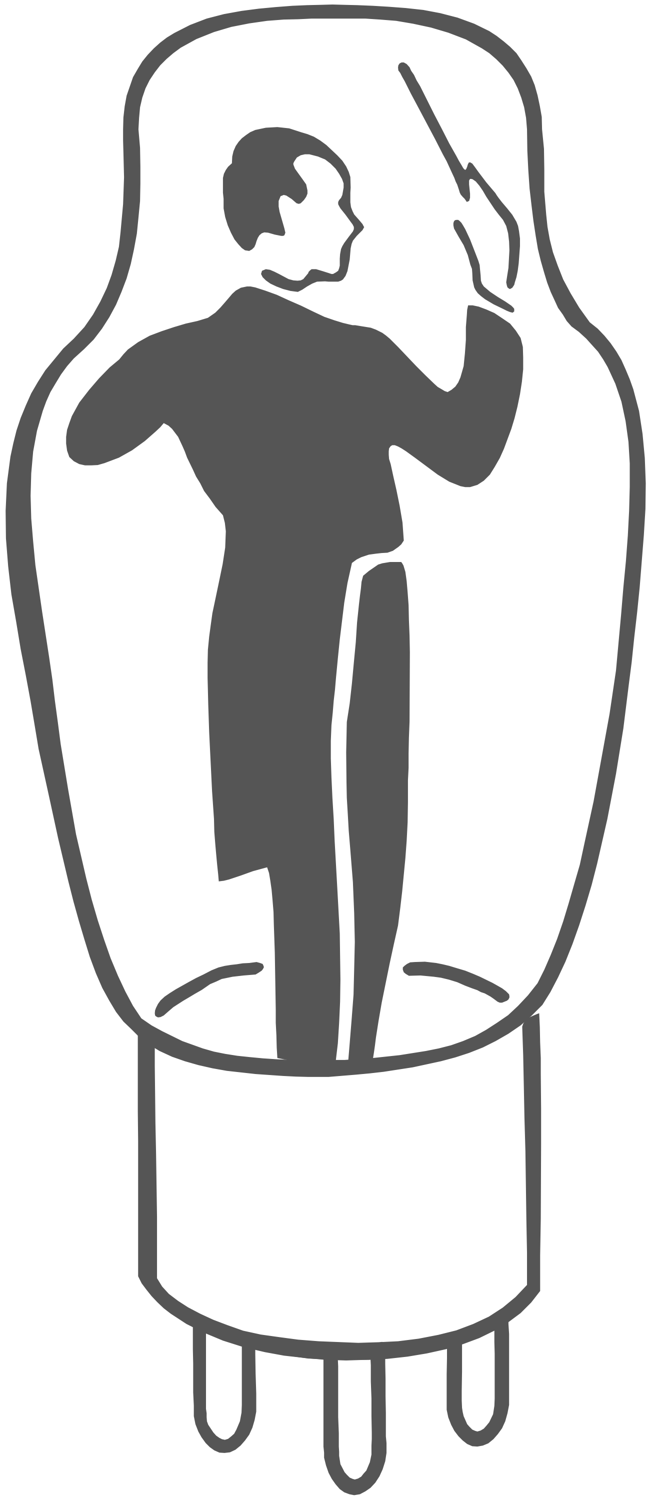
LTspice Working Paper

for

Tube-Amplifier Simulations



Version: 2.8

kurtblum.com

Preface

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.

Kurt Blum Switzerland

Contents

Keyboard Shortcuts Table

LTspice Keyboard Shortcuts

LTspice HotKeys

	Schematic	Symbol	Waveform	Netlist
Modes	ESC - Exit Mode	ESC - Exit Mode		
	F3 – Draw Wire			
	F5 – Delete	F5 – Delete	F5 – Delete	
	F6 – Duplicate	F6 – Duplicate		
	F7 – Move	F7 – Move		
	F8 – Drag	F8 – Drag		
	F9 – Undo	F9 – Undo	F9 – Undo	F9 – Undo
	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo
View	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	
	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	
	Space – Zoom Fit		Ctrl+E – Zoom Extents	
	Ctrl+G – Toggle Grid		Ctrl+G – Toggle Grid	Ctrl+G – Goto Line #
	U – Mark Unncon. Pins	Ctrl+W – Attribute Window	'0' - Clear	
	A – Mark Text Anchors	Ctrl+A – Attribute Editor	Ctrl+A – Add Trace	
	Atl+Click - Power		Ctrl+Y – Vertical Autorange	Ctrl+R – Run Simulation
	Ctrl+Click - Attr. Edit		Ctrl+Click - Average	
	Ctrl+H – Halt Simulation		Ctrl+H – Halt Simulation	Ctrl+H – Halt Simulation
Place	R – Resistor	R – Rectangle		
	C – Capacitor	C – Circle		
	L – Inductor	L – Line		
	D – Diode	A – Arc		
	G – GND			
	S – Spice Directive			
	T – Text	T – Text		
	F2 – Component			
	F4 – Label Net			
	Ctrl+E – Mirror	Ctrl+E – Mirror		
	Ctrl+R – Rotate	Ctrl+R – Rotate		

Command Line Switches

Flag	Short Description
-ascii	Use ASCII .raw files. (Degrades performance!)
-b	Run in batch mode.
-big or -max	Start as a maximized window.
-encrypt	Encrypt a model library.
-FastAccess	Convert a binary .raw file to Fast Access Format.
-netlist	Convert a schematic to a netlist.
-nowine	Prevent use of WINE(Linux) workarounds.
-PCBnetlist	Convert a schematic to a PCB netlist.
-registry	Store user preferences in the registry.
-Run	Start simulating the schematic on open.
-SOI	Allow MOSFET's to have up to 7 nodes in subcircuit.
-uninstall	Executes one step of the uninstallation process.
-wine	Force use of WINE(Linux) workarounds.

Simulator Directives - Dot Commands

Command	Short Description
.AC	Perform a Small Signal AC Analysis
.BACKANNO	Annotate the Subcircuit Pin Names on Port currents
.DC	Perform a DC Source Sweep Analysis
.END	End of Netlist
.ENDS	End of Subcircuit Definition
.FOUR	Compute a Fourier Component
.FUNC	User Defined Functions
.FERRET	Download a File Given the URL
.GLOBAL	Declare Global Nodes
.IC	Set Initial Conditions
.INCLUDE	Include another File
.LIB	Include a Library
.LOADBIAS	Load a Previously Solved DC Solution
.MEASURE	Evaluate User-Defined Electrical Quantities
.MODEL	Define a SPICE Model
.NET	Compute Network Parameters in a .AC Analysis
.NODESET	Supply Hints for Initial DC Solution
.NOISE	Perform a Noise Analysis
.OP	Find the DC Operating Point
.OPTIONS	Set Simulator Options
.PARAM	User-Defined Parameters
.SAVE	Limit the Quantity of Saved Data
.SAVEBIAS	Save Operating Point to Disk
.STEP	Parameter Sweeps
.SUBCKT	Define a Subcircuit
.TEMP	Temperature Sweeps
.TF	Find the DC Small-Signal Transfer Function
.TRAN	Do a Nonlinear Transient Analysis
.WAVE	Write Selected Nodes to a .WAV file

Suffix		Suffix	
		f	1e-15
T	1e12	p	1e-12
G	1e9	n	1e-9
Meg	1e6	u	1e-6
K	1e3	M	1e-3
		Mil	25.4e-6

Constants	
E	2.7182818284590452354
Pi	3.14159265358979323846
K	1.3806503e-23
Q	1.602176462e-19
TRUE	1
FALSE	0

Simulation

Examples of Simulation

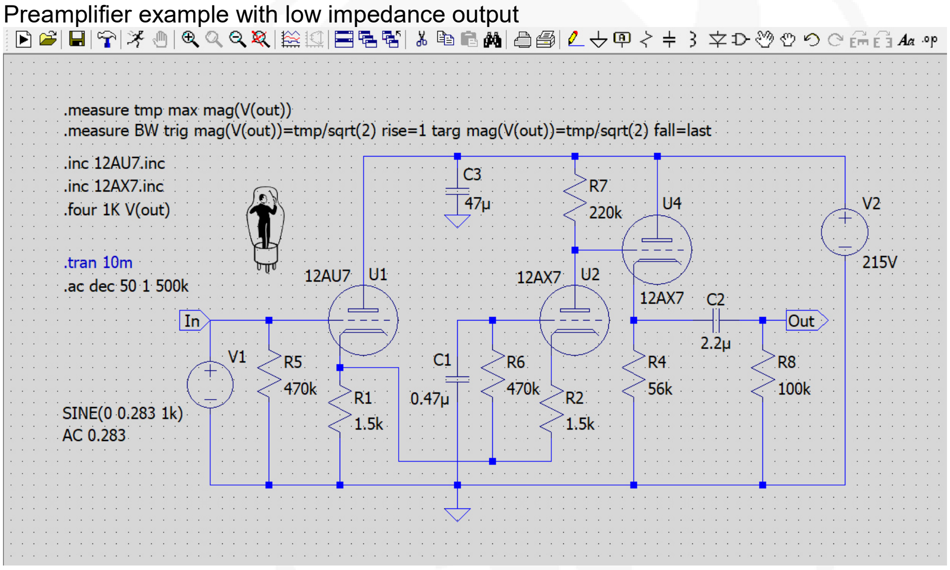
Examples to learn how LTspice is working with tube models and audio transformers
Hmm, of course there is still room to improve these examples to turn it into the world's best amplifier out of it !

- 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: $2{,}0V_{RMS} = 5.657V_{pp} = 2.828_p$

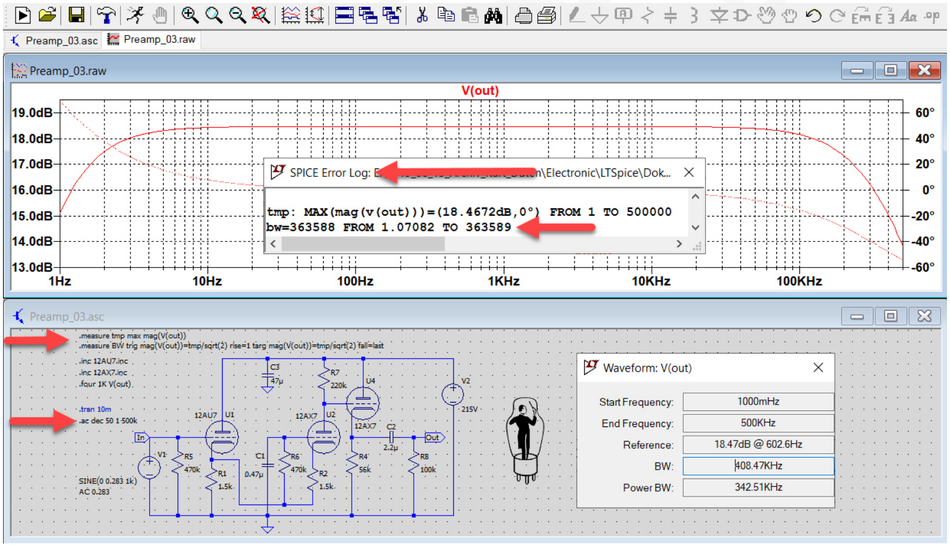
Preamplifier Examples

1. SE Preamplifier

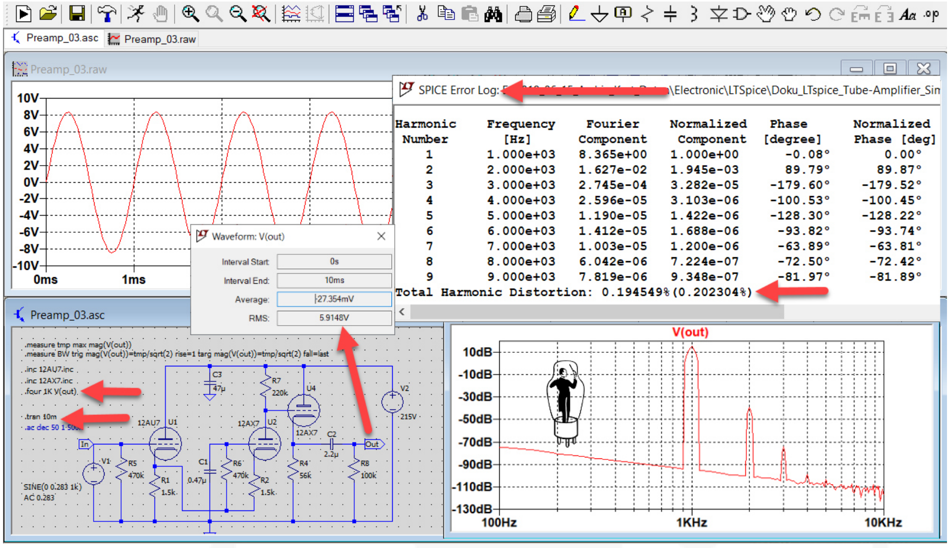
- Preamplifier example with low impedance output



- 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 !

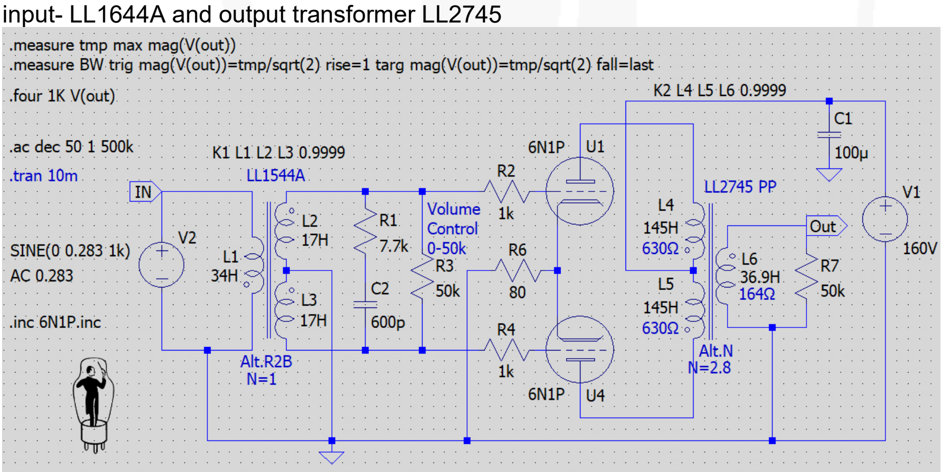


- Simulation: .tran 10m
- Input: sinus 1kHz and 200mV$_{RMS}$, output 5.915V$_{RMS}$,
- Gain: $20\log\frac{V(out)}{V(in)}$ = 29,4dB
- Harmonic distortion: THD = 0.19%; THD+N = 0.202% !

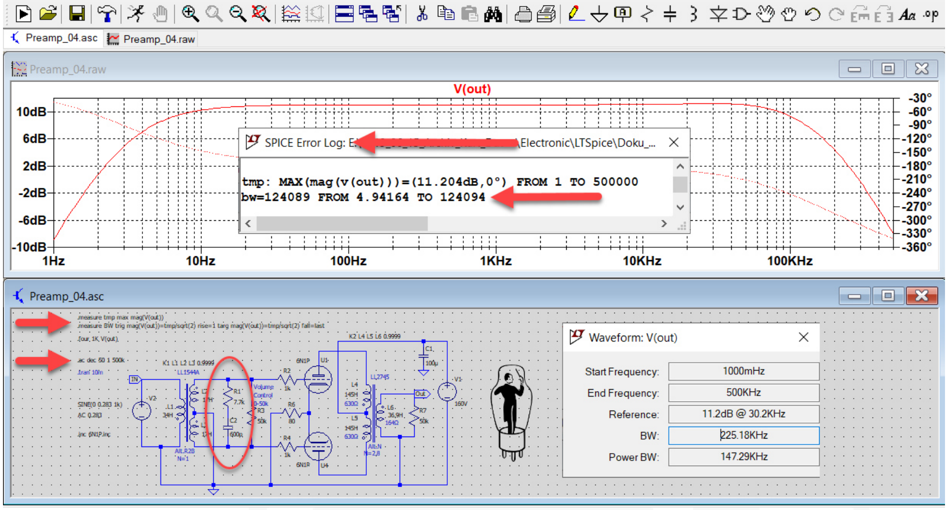


2. PP Preamplifier

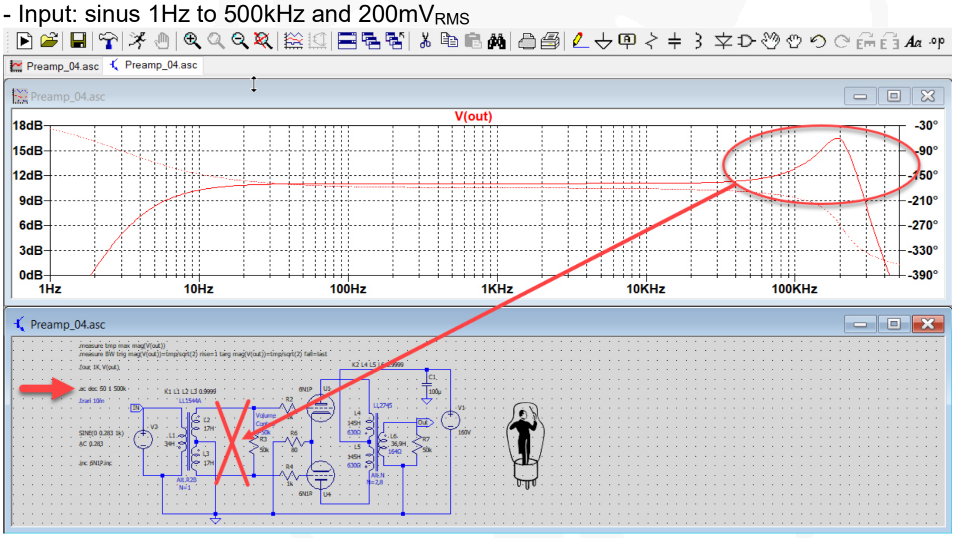
- Example of push pull preamplifier with tube 6N1P, volume control 0-50kΩ, zobel network input- LL1644A and output transformer LL2745



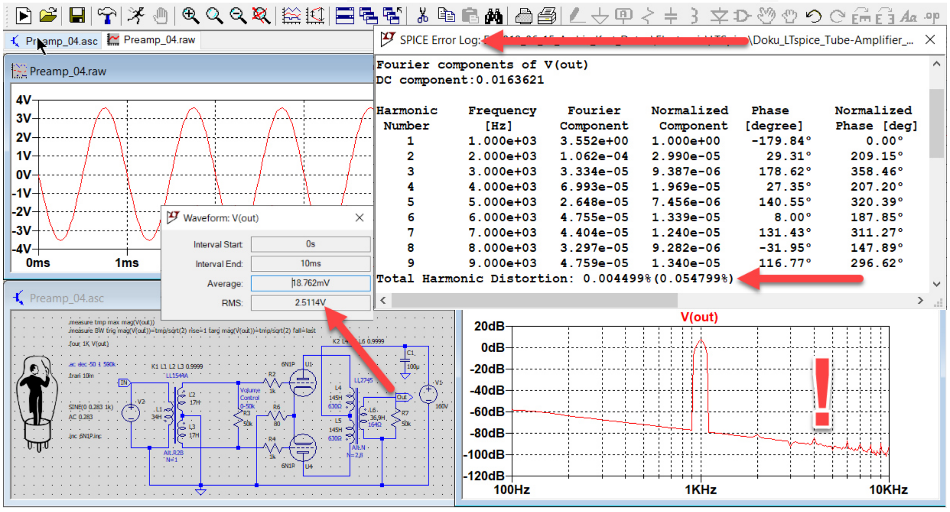
- Simulation: .ac dec 50 1 500k with zobel network
- Input: sinus 1Hz to 500kHz and 200mV$_{RMS}$
- Bandwidth: -3dB = 124.1kHz, from 4.94Hz to 124.1kHz !



- Simulation: .ac dec 50 1 500k without zobel network
- Input: sinus 1Hz to 500kHz and 200mV$_{RMS}$



- Simulation: .tran 10m
- Input: sinus 1kHz and 200mV$_{RMS}$, output 2.51V$_{RMS}$,
- Gain: $20\log\frac{V_{(out)}}{V_{(in)}}$ = 22,0dB
- Harmonic distortion: THD = 0.00449% !, THD+H = 0.054799% !

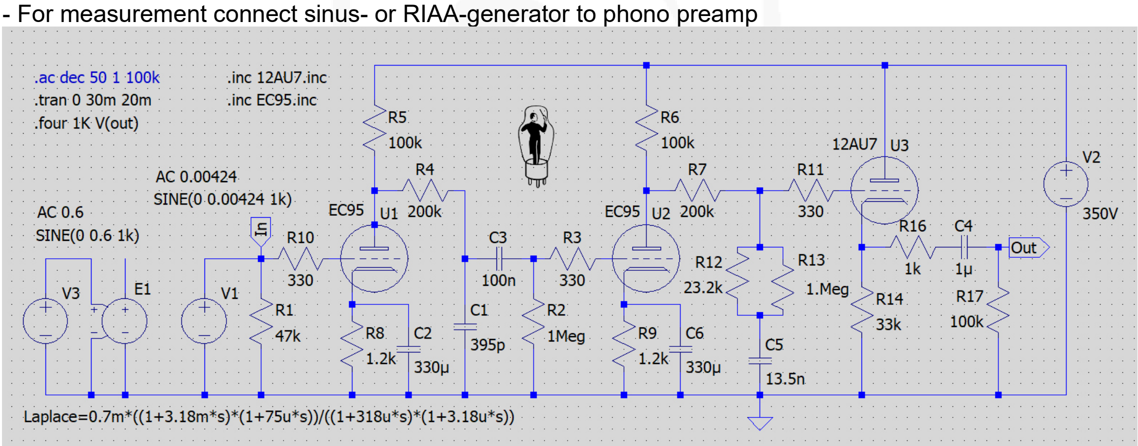


Phono Preamplifier Example

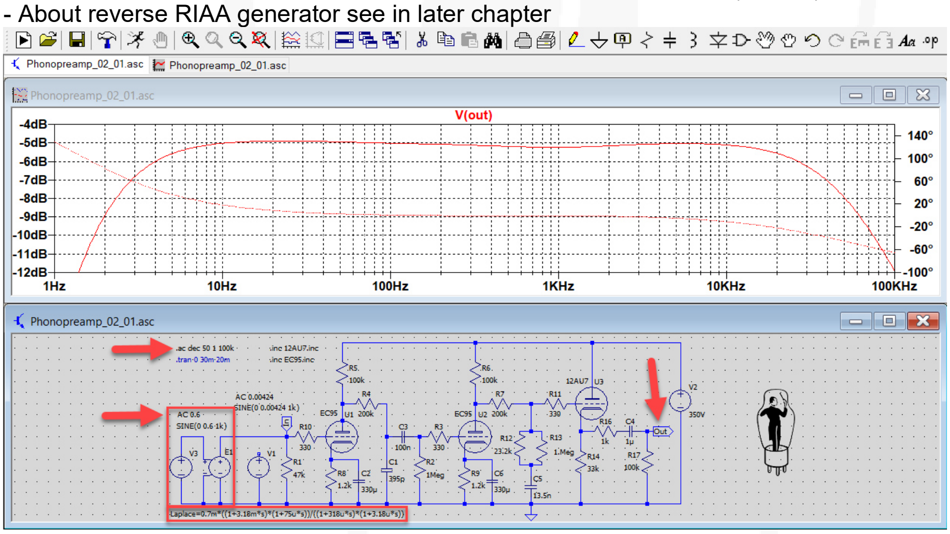
1. Phono Preamplifier with passive RIAA

- 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

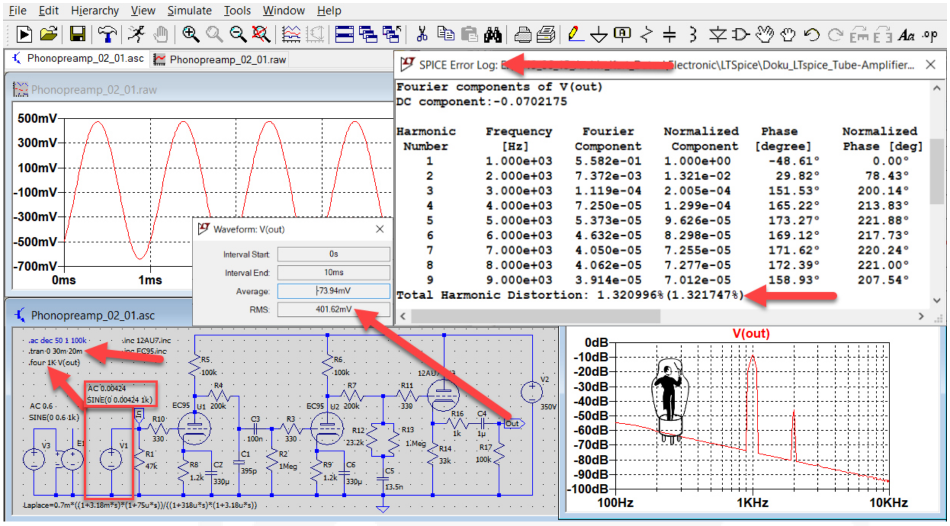
- Example with tube EC95 / 6ER and ECC83 / 12AU7
  - For measurement connect sinus- or RIAA-generator to phono preamp



- Simulation: .ac dec 50 10 100k
- Input: sinus 10Hz to 100kHz reverse RIAA and $3mV_{RMS} = 0.00424V_p$ ($0.6V_p$ at the RIAA Generator)
- About reverse RIAA generator see in later chapter



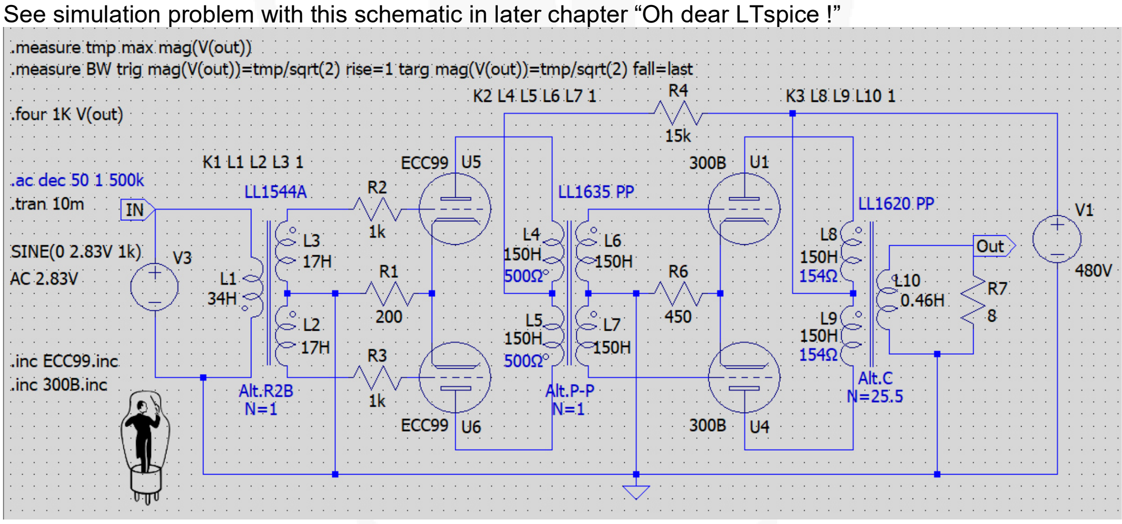
- Simulation: .tran 030m 20m
- Input: sinus 1kHz and 3mV$_{RMS}$, output 401.6V$_{RMS}$,
- Gain: $20 \log \frac{V_{(out)}}{V_{(in)}}$ = 42,5dB
- Total harmonic distortion: 1.32%



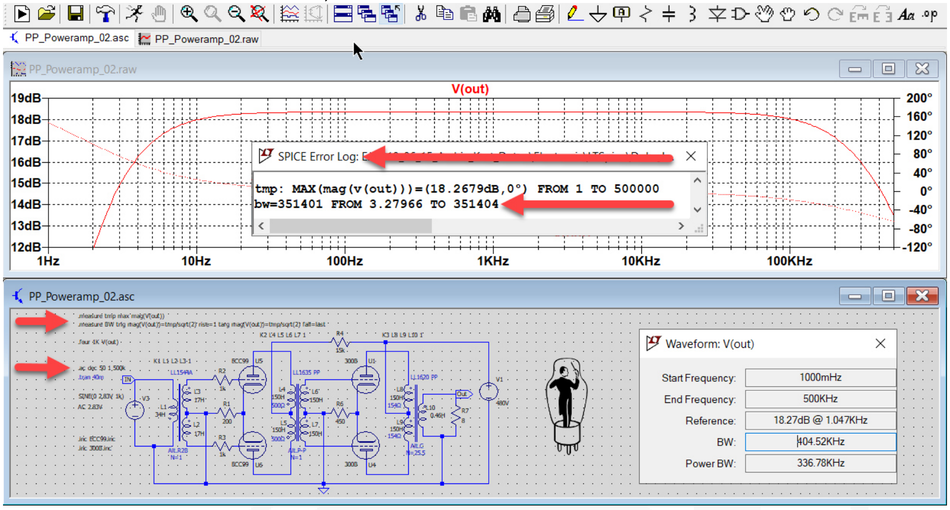
Power Amplifier

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 !”

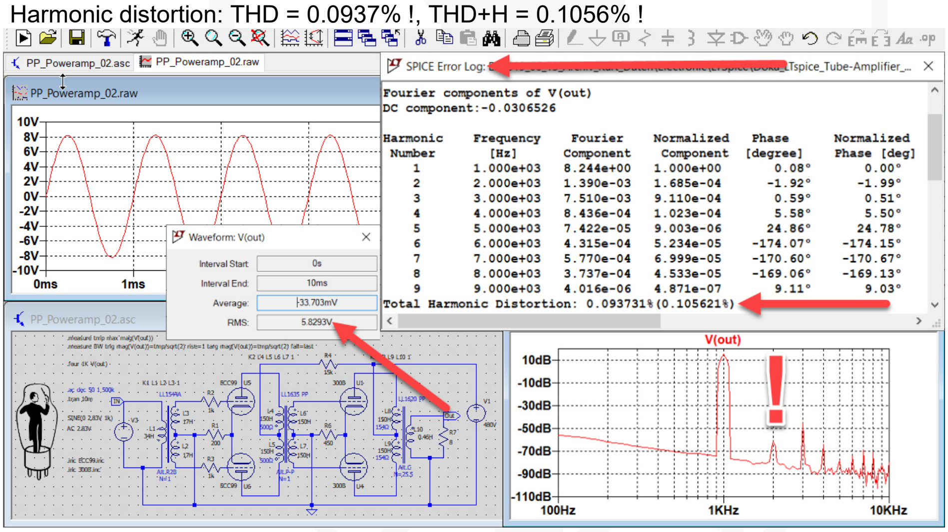


- - Simulation: .ac dec 50 1 500k, coupling factor of all transformers K = 1.0
  - Input: sinus 1Hz to 500kHz and 200mV$_{RMS}$
  - Bandwidth: -3dB = 351.4kHz, from 3.3Hz to 351.4kHz !



  - Simulation: .tran 10m
  - Input: sinus 1kHz and 200mV$_{RMS}$, output 5.83V$_{RMS}$,
  - Output Power: $P = \frac{U^2}{R}$ = 4.25 Watt

  Harmonic distortion: THD = 0.0937% !, THD+H = 0.1056% !

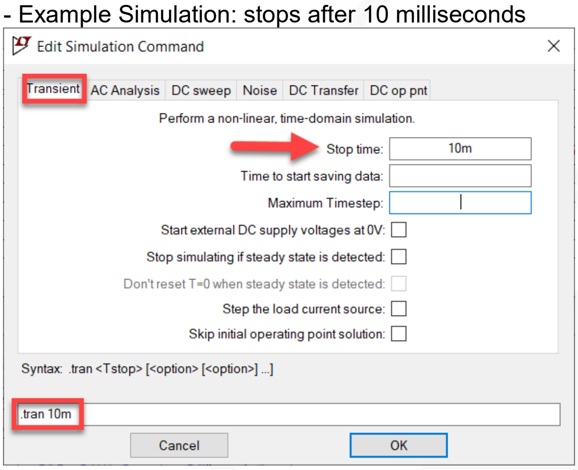


SPICE Directives Examples

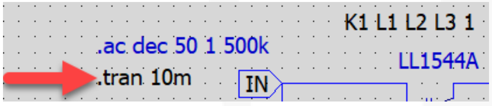
1. Simulation SPICE Directives

- 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

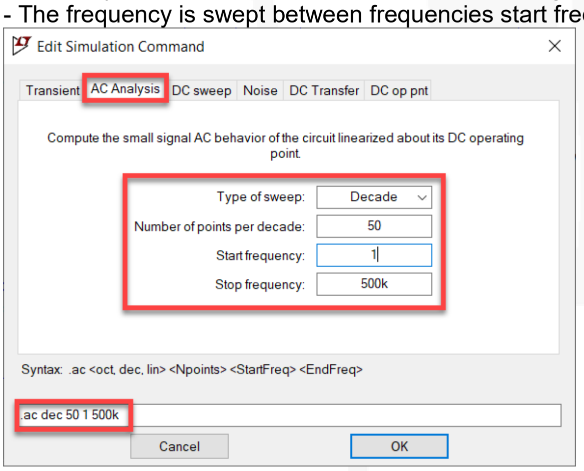


  - The active ".tran" simulation directive is in black color on the schematics

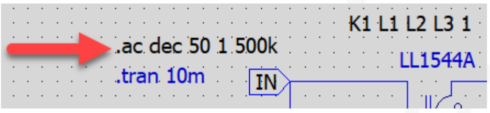


- AC Analysis Simulation

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



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



2. SPICE Dot Commandes

- Harmonic Distortion THD and THD+H Check

.four 1K V(out)

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 Error Log: [redacted]

```
Fourier components of V(out)
DC component:-0.0308528

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 Harmonic Distortion: 0.093731%(0.105621%)
```

- Bandwidth minus 3dB Check

.measure tmp max mag(V(out))

This two dot commands are only computing a result after ".ac" simulation
- 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

SPICE Error Log: [redacted]

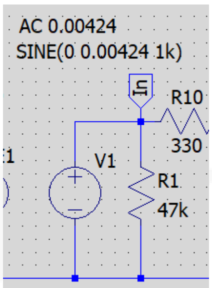
```
tmp: MAX(mag(v(out)))=(18.3604dB,0°) FROM 1 TO 500000
bw=351401 FROM 3.27966 TO 351404
```

- RIAA Forward Curve of Phono Preamp Check

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

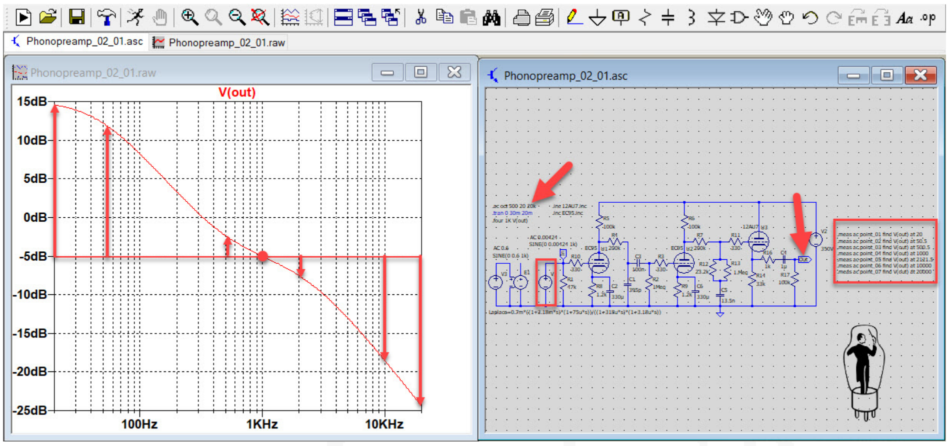
plus



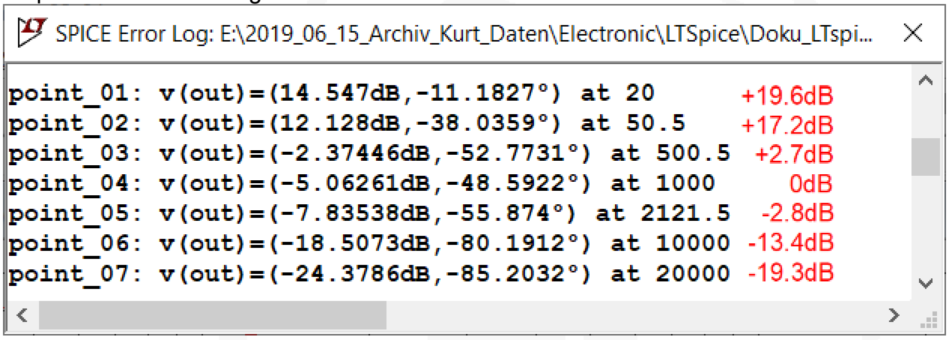
plus

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

- ".ac oct" simulation



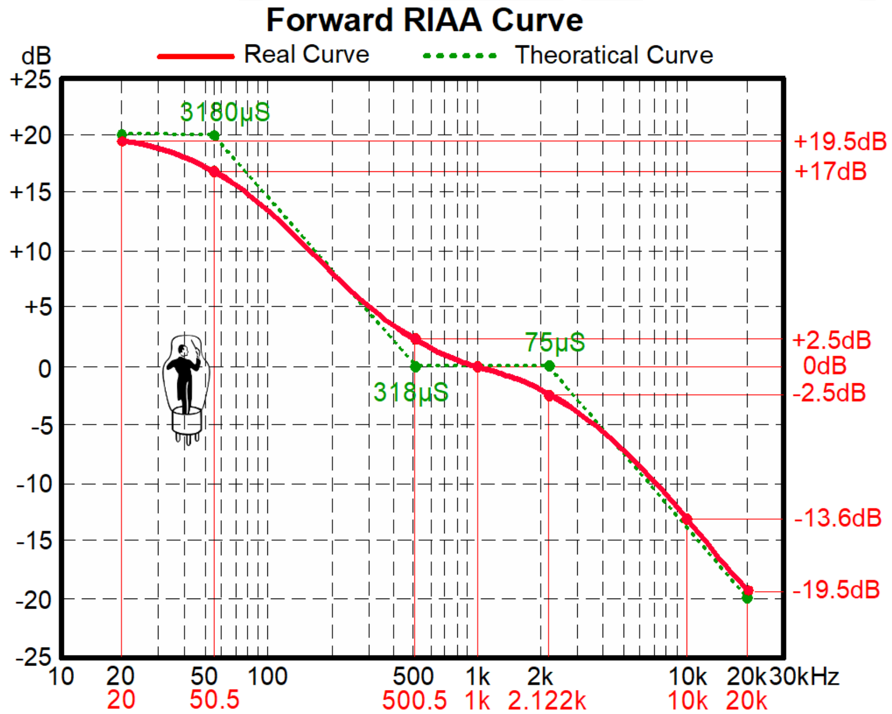
- Open SPICE Error Log file



SPICE Error Log: E:\2019_06_15_Archiv_Kurt_Daten\Electronic\LTSpice\Doku_LTspi...

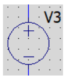
```
point_01: v(out)=(14.547dB,-11.1827°) at 20         +19.6dB
point_02: v(out)=(12.128dB,-38.0359°) at 50.5       +17.2dB
point_03: v(out)=(-2.37446dB,-52.7731°) at 500.5    +2.7dB
point_04: v(out)=(-5.06261dB,-48.5922°) at 1000     0dB
point_05: v(out)=(-7.83538dB,-55.874°) at 2121.5    -2.8dB
point_06: v(out)=(-18.5073dB,-80.1912°) at 10000    -13.4dB
point_07: v(out)=(-24.3786dB,-85.2032°) at 20000    -19.3dB
```

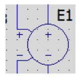
- The standard forward RIAA filter curve



- RIAA Inverse Curve Generator check

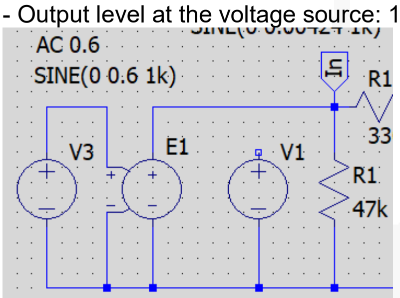
  - The RIAA Generator is the combination of two components:

  “normal voltage source” 

plus “voltage controlled voltage source” (VCVS) 

  plus transfer function of the VCVS Laplace=0.7m*((1+3.18m*s)*(1+75u*s))/((1+318u*s)*(1+3.18u*s))

  - “.ac” simulation: 20Hz to 20kHz
  - Sinus level at output VCVS: $3m_{VRMS}$ = $4.24mV_P$
  - Output level at the voltage source: 1.2 V_{PP}



plus

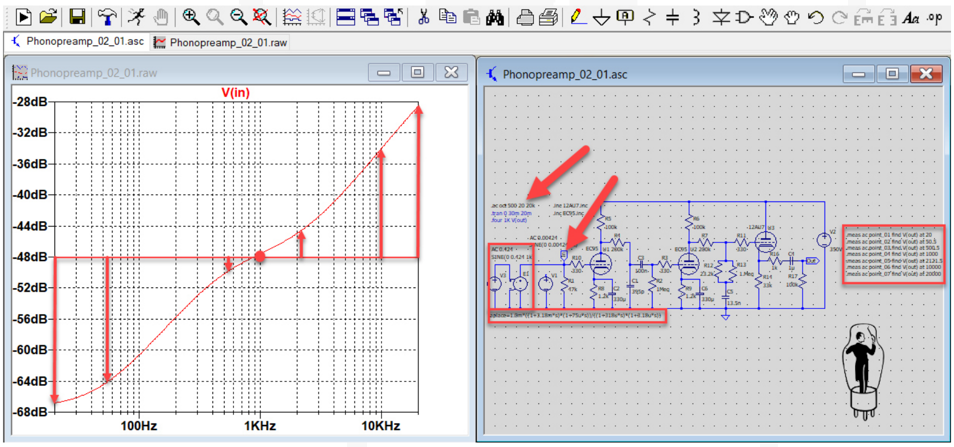
  - “Laplace” = Multiplier coefficient between command and output transfer function of VCVS

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

  plus

  “.ac oct” simulation

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

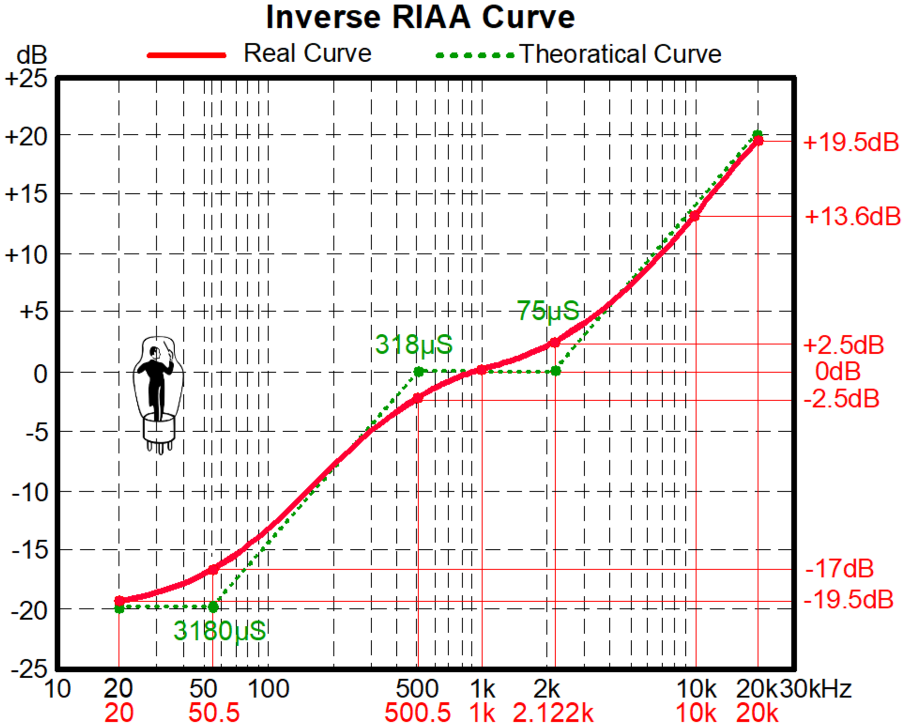


  - Open SPICE Error Log file

SPICE Error Log: E:\2019_06_15_Archiv_Kurt_Daten\Electronic\LTSpice\Doku_LTspi...

```
point_01: v(in)=(-66.8156dB,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        0dB
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
```

- The standard inverse RIAA filter curve



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 → lib → sym → misc. folder and the tube-models into a separate folder
Find tube-symbols on the internet, here are some different sources:

- http://www.intactaudio.com/forum/viewtopic.php?t=24 tube-symbols and tube-models from Stephie Bench and/or Duncan’s

- http://www.dmitrynizh.com/dmitry_composites.zip tube-models from Dmitry Nizehegorodev save only the tube-model library: dmitry_composites.lib

- http://www.normankoren.com/Audio/Tubemods.zip tube-models from Norman Koren save only the tube-model library: Tube97.lib; tube1.lib; Tube.lib

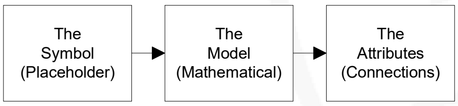
- https://www.dos4ever.com/uTracer3/TubeLib.inc tube-models are created with the “ExtractModel” program of Derk Reefman and are based on real tube data measured with “μTracer” curve tracer from Ronald Dekker https://www.dos4ever.com

- http://ayumi.cava.jp/audio/tubemodel_3.20_win.zip
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

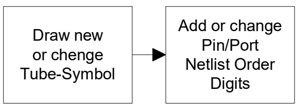
- Workflow



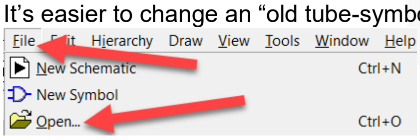
- The symbol (placeholder of the tube-model on the schema)
- The model (mathematical description of the component)
- The attributes and connection-number (LTspice instruction and link between tube-symbol and tube-model)

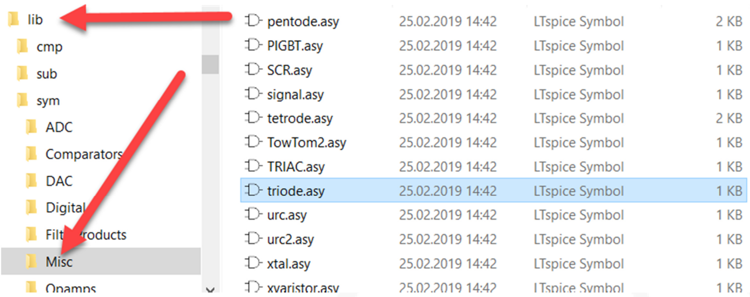
2. Tube-Symboles (Placeholder)

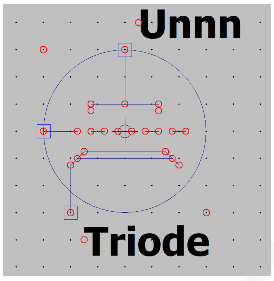
- Workflow



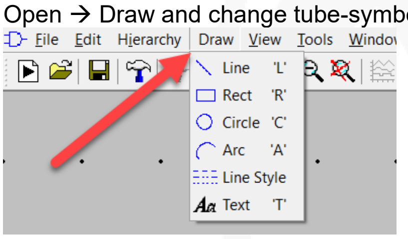
- Open File → Open.. →“Existing Symbol”
- It’s easier to change an “old tube-symbol” then to create a completely new one



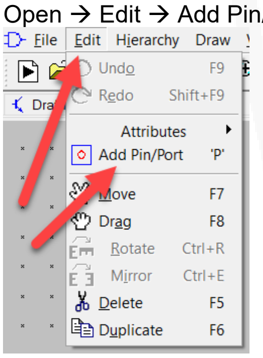




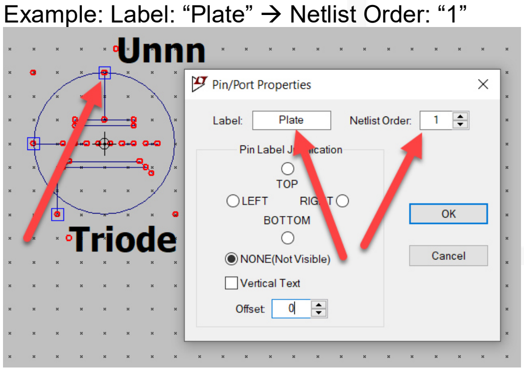
- Open → Draw and change tube-symbol



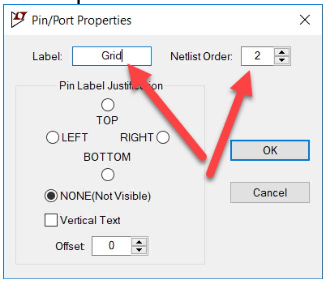
- Open → Edit → Add Pin/Port for add a new or extra Pin/Port



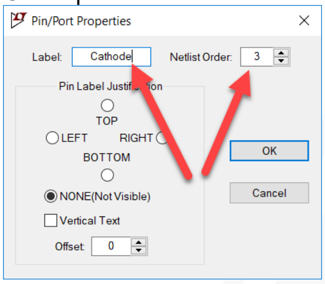
- Change Pin/Port numbering, click with right mouse button on Pin/Port
- Netlist Order “digit” has to correspond with the tube-model
- Example: Label: “Plate” → Netlist Order: “1”



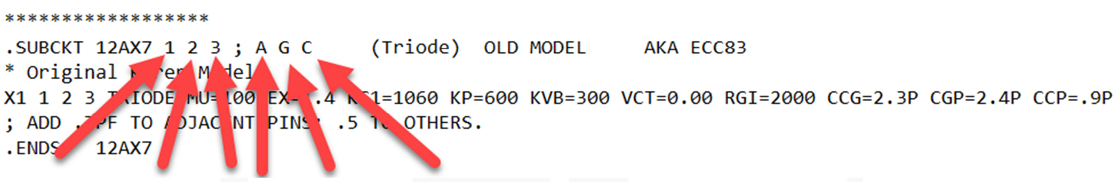
- Correspondent connection for the grid



- Correspondent connection for the cathode

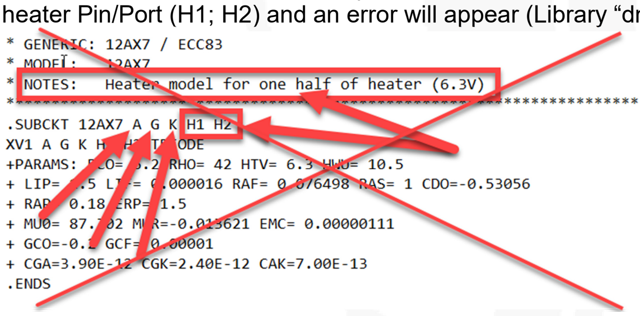


- Example: the correspondent link from the tube-symbol to the tube-model (Library “Koren_Tubes.inc”)



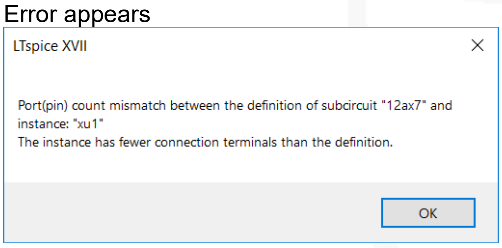
```
******************
.SUBCKT 12AX7 1 2 3 ; A G C      (Triode)  OLD MODEL     AKA ECC83
* Original Koren Model
X1 1 2 3 TRIODE MU=100 EX=1.4 KG1=1060 KP=600 KVB=300 VCT=0.00 RGI=2000 CCG=2.3P CGP=2.4P CCP=.9P
; ADD .7PF TO ADJACENT PINS; .5 TO OTHERS.
.ENDS 12AX7
```

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



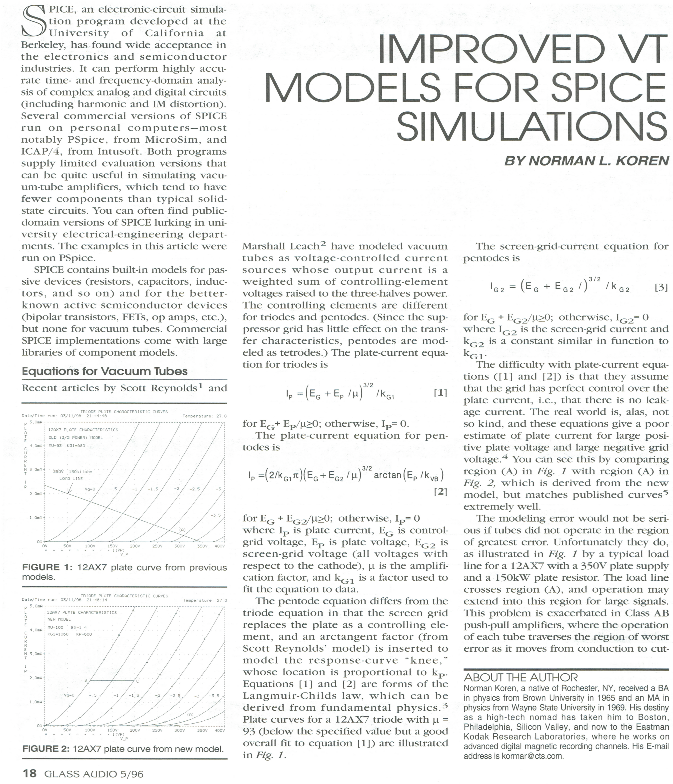
```
* GENERIC: 12AX7 / ECC83
* MODEL:    12AX7
* NOTES:    Heater model for one half of heater (6.3V)
***********************************************************
.SUBCKT 12AX7 A G K H1 H2
XV1 A G K H1 H2 TRIODE
+PARAMS: RCO= 6.2 RHO= 42 HTV= 6.3 HWU= 10.5
+ LIP= 1.5 LTF= 0.000016 RAF= 0.076498 RAS= 1 CDO=-0.53056
+ RAP= 0.18 ERP= 1.5
+ MU0= 87.702 MUR=-0.013621 EMC= 0.00000111
+ GCO=-0.2 GCF= 0.00001
+ CGA=3.90E-12 CGK=2.40E-12 CAK=7.00E-13
.ENDS
```

- Error appears



3. Tube-Model Code (Mathematical)

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



IMPROVED VT MODELS FOR SPICE SIMULATIONS

BY NORMAN L. KOREN

SPICE, an electronic-circuit simulation program developed at the University of California at Berkeley, has found wide acceptance in the electronics and semiconductor industries. It can perform highly accurate time- and frequency-domain analysis of complex analog and digital circuits (including harmonic and IM distortion). Several commercial versions of SPICE run on personal computers—most notably PSpice, from MicroSim, and ICAP/4, from Intusoft. Both programs supply limited evaluation versions that can be quite useful in simulating vacuum-tube amplifiers, which tend to have fewer components than typical solid-state circuits. You can often find public-domain versions of SPICE lurking in university electrical-engineering departments. The examples in this article were run on PSpice.

SPICE contains built-in models for passive devices (resistors, capacitors, inductors, and so on) and for the better-known active semiconductor devices (bipolar transistors, FETs, op amps, etc.), but none for vacuum tubes. Commercial SPICE implementations come with large libraries of component models.

Equations for Vacuum Tubes

Recent articles by Scott Reynolds[1] and Marshall Leach[2] have modeled vacuum tubes as voltage-controlled current sources whose output current is a weighted sum of controlling-element voltages raised to the three-halves power. The controlling elements are different for triodes and pentodes. (Since the suppressor grid has little effect on the transfer characteristics, pentodes are modeled as tetrodes.) The plate-current equation for triodes is

$$I_P = \left(E_G + E_P/\mu\right)^{3/2}/k_{G1} \quad [1]$$

for $E_G + E_P/\mu \geq 0$; otherwise, $I_P = 0$.

The plate-current equation for pentodes is

$$I_P = \left(2/k_{G1}\pi\right)\left(E_G + E_{G2}/\mu\right)^{3/2}\arctan\left(E_P/k_{VB}\right) \quad [2]$$

for $E_G + E_{G2}/\mu \geq 0$; otherwise, $I_P = 0$ where I_P is plate current, E_G is control-grid voltage, E_P is plate voltage, E_{G2} is screen-grid voltage (all voltages with respect to the cathode), μ is the amplification factor, and k_{G1} is a factor used to fit the equation to data.

The pentode equation differs from the triode equation in that the screen grid replaces the plate as a controlling element, and an arctangent factor (from Scott Reynolds' model) is inserted to model the response-curve "knee," whose location is proportional to k_P. Equations [1] and [2] are forms of the Langmuir-Childs law, which can be derived from fundamental physics.[3] Plate curves for a 12AX7 triode with μ = 93 (below the specified value but a good overall fit to equation [1]) are illustrated in *Fig. 1*.

The screen-grid-current equation for pentodes is

$$I_{G2} = \left(E_G + E_{G2}/\right)^{3/2}/k_{G2} \quad [3]$$

for $E_G + E_{G2}/\mu \geq 0$; otherwise, $I_{G2} = 0$ where I_{G2} is the screen-grid current and k_{G2} is a constant similar in function to k_{G1}.

The difficulty with plate-current equations ([1] and [2]) is that they assume that the grid has perfect control over the plate current, i.e., that there is no leakage current. The real world is, alas, not so kind, and these equations give a poor estimate of plate current for large positive plate voltage and large negative grid voltage.[4] You can see this by comparing region (A) in *Fig. 1* with region (A) in *Fig. 2*, which is derived from the new model, but matches published curves[5] extremely well.

The modeling error would not be serious if tubes did not operate in the region of greatest error. Unfortunately they do, as illustrated in *Fig. 1* by a typical load line for a 12AX7 with a 350V plate supply and a 150kW plate resistor. The load line crosses region (A), and operation may extend into this region for large signals. This problem is exacerbated in Class AB push-pull amplifiers, where the operation of each tube traverses the region of worst error as it moves from conduction to cut-

FIGURE 1: 12AX7 plate curve from previous models.

FIGURE 2: 12AX7 plate curve from new model.

ABOUT THE AUTHOR
Norman Koren, a native of Rochester, NY, received a BA in physics from Brown University in 1965 and an MA in physics from Wayne State University in 1969. His destiny as a high-tech nomad has taken him to Boston, Philadelphia, Silicon Valley, and now to the Eastman Kodak Research Laboratories, where he works on advanced digital magnetic recording channels. His E-mail address is kormar@cts.com.

18 GLASS AUDIO 5/96

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

- 12AX7 Spice-Example out of Norman L. Koren Library "Tube1.lib"
- 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)}
```

```
Ga    A  K  VALUE {V(EL)}
Egf   GF 0  VALUE {PWR(LIMIT(V(G,K)-{GCO},0,1E6),1.5)*{GCF}}
Gg    G  K  VALUE {(V(GF)+V(LD))}
CM1             G              K              {CGK}
CM2             A              G              {CGA}
CM3             A              K              {CAK}
RF1             A              0              1000MEG
RF2             G              0              1000MEG
RF3             K              0              1000MEG
.ENDS
************************************************************************
* GENERIC: 12AX7 / ECC83
* MODEL:   NH12AX7
* NOTES:   No heater model

************************************************************************
.SUBCKT NH12AX7 A G K
XV1 A G K TRIODENH
+PARAMS: LIP= 1.5 LIF= 0.000016 RAF= 0.076498 RAS= 1 CDO=-0.53056
+ RAP= 0.18 ERP= 1.5
+ MU0= 87.302 MUR=-0.013621 EMC= 0.00000111
+ GCO=-0.2 GCF= 0.00001
+ CGA=3.90E-12 CGK=2.40E-12 CAK=7.00E-13
.ENDS
************************************************************************
```

- 12AX7 Spice Example out of Ronald Dekker / Rolf Refman Library “TubeLib.inc

```
******************************************************
.SUBCKT 12AX7 1 2 3; A G C;
X1 1 2 3 ECC83
.ENDS
******************************************************
******************************************************
.SUBCKT ECC83 1 2 3; A G C;
* ExtractModel V .998
* Model created: 09-Dec-13
X1 1 2 3 TriodeK MU=108.93 EX= .988 KG1= 389.8 KP= 677.7 KVB=10751. RGI=2000
+ CCG=1.6P  CGP=1.6P CCP=0.33P  ;
.ENDS
******************************************************
******************************************************
.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
******************************************************
```

- 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
1 2 3	= number of the or Pin/Port
A G C	= 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 .998
* Model created: 09-Dec-13
X1 1 2 3 TriodeK = call of the subcircuit "TriodeK"
MU=108.93 EX= .988 KG1= 389.8 KP= 677.7 KVB=10751. RGI=2000
= the „Phenomenological Equations Factors" are the most important part of the tube-model, with this factors LTspice calculates the tube-curves

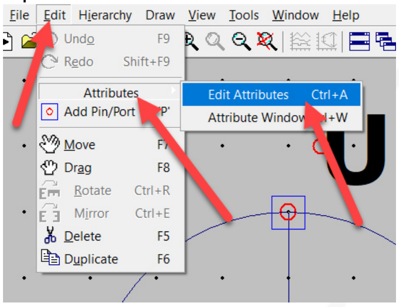
MU=108.93	= amplification factor (μ)
EX= .988	= calculated factor out of tube data
KG1= 389.8	= out of grid- and anode-voltage calculated factor
KG2=	= additional factor only for pentode-model
KP= 677.7	= calculated factor by high negative grid-voltage and small plate voltage
KVB=10751.	= different twist 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

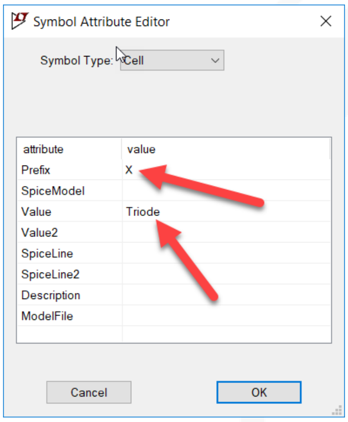
```
.SUBCKT TriodeK 1 2 3; A G C
RE1 7 0 1G
E1 7 0 VALUE=
+{V(1,3)/KP*LOG(1+EXP(KP*(1/MU+V(2,3)/SQRT(KVB+V(1,3)*V(1,3)))))}
G1 1 3 VALUE={0.5*(PWR(V(7),EX)+PWRS(V(7),EX))/KG1}
RCP 1 3 1G    ; TO AVOID FLOATING NODES IN MU-FOLLOWER
C1 2 3 {CCG}  ; CATHODE-GRID
C2 2 1 {CGP}  ; GRID-PLATE
C3 1 3 {CCP}  ; 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)

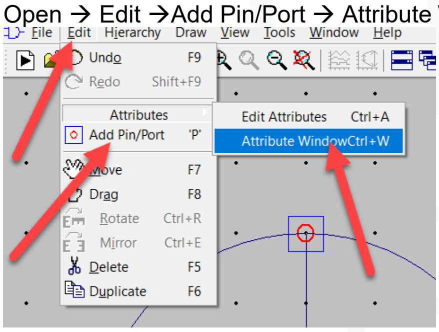
- The attributes includes the work information for the symbol (placeholder).
- Open → Edit → Attributes → Edit Attributes



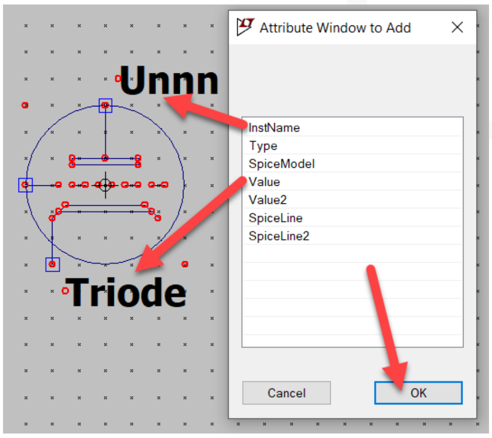
- Edit → Prefix: "X"
- Edit → Value: "Triode"



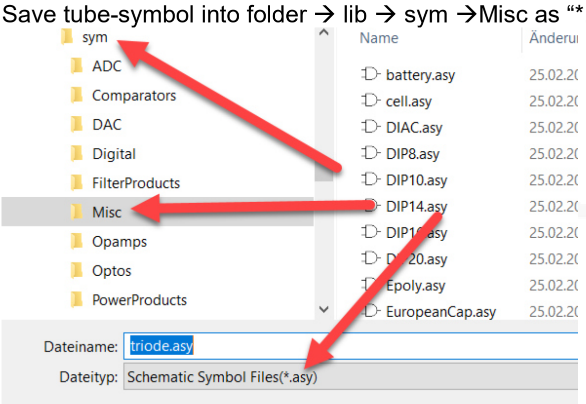
- Open → Edit →Add Pin/Port → Attribute Window



- 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"



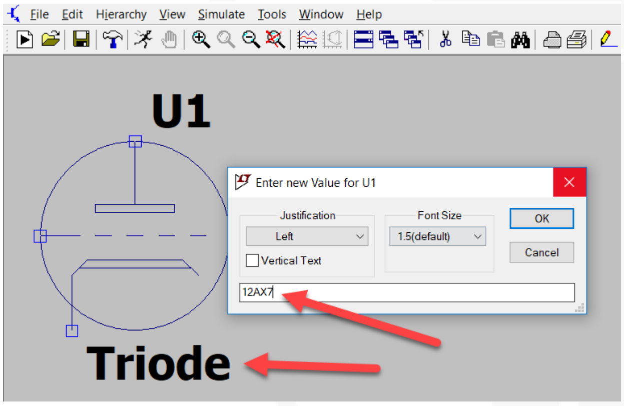
- Save tube-symbol into folder → lib → sym →Misc as "*.asy" file



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

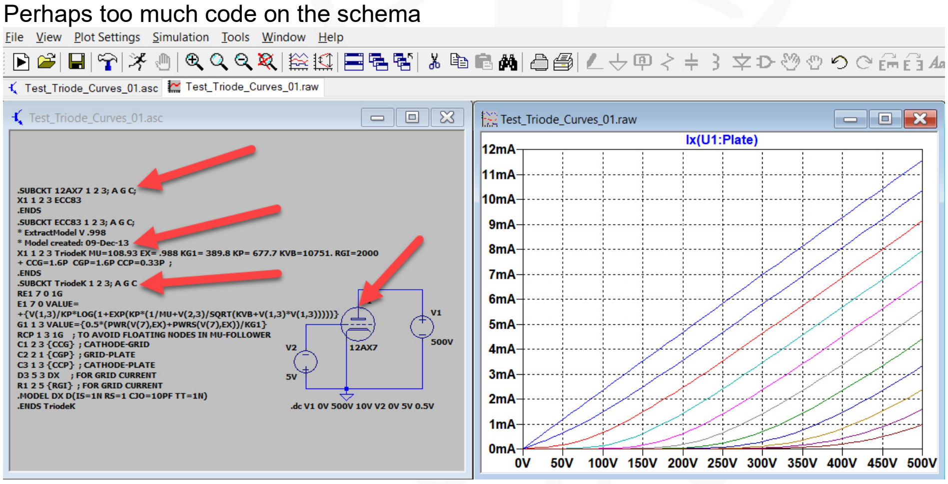
1. Tube-Symbol

- Place tube-symbol on the desktop
- Click name "Triode" and enter "Tube-Number"



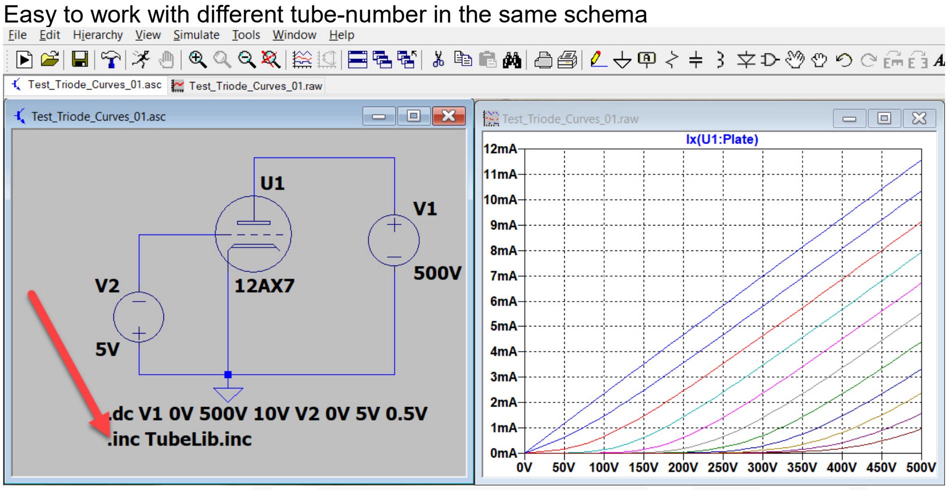
2. Place Tube-Model Code into the Schema

- When copy schema, all the information are included
- Perhaps too much code on the schema



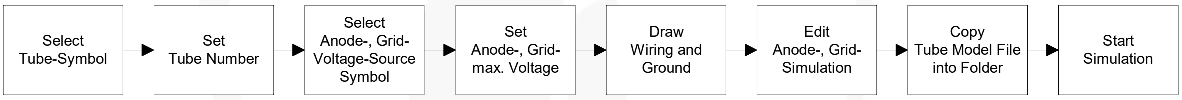
3. Link Tube-Symbol and Tube-Library together

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

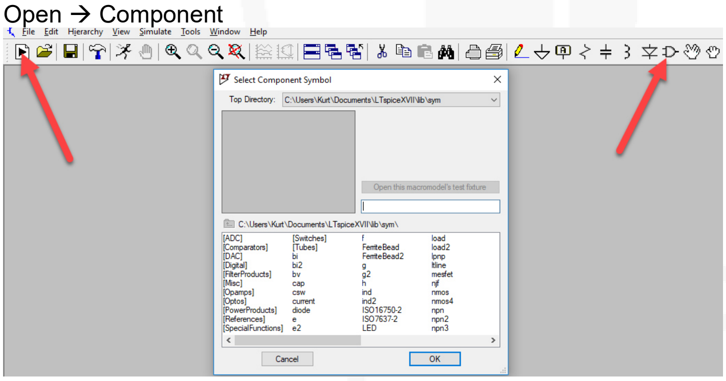


4. Check Tube-Model with LTspice

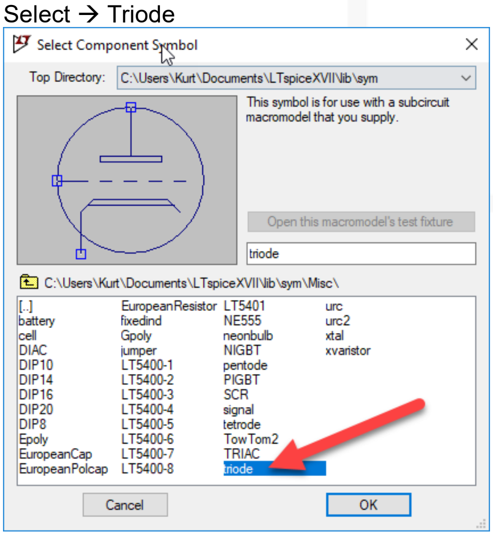
- Workflow



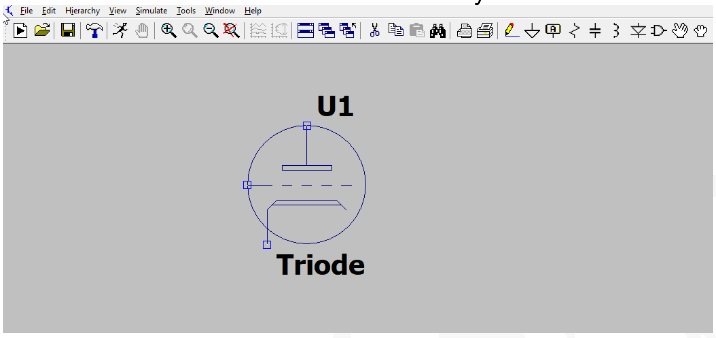
- Open → New Schematic
- Open → Component



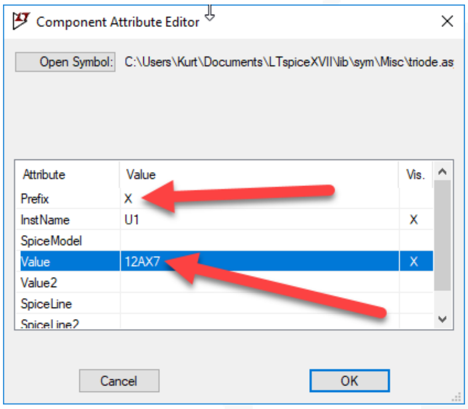
- Select → Component Symbol
- Select → Triode



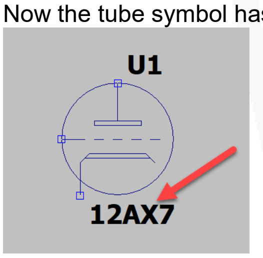
- Place “Triode Symbol” on new schema
- Click with left mouse button on the “Triode Symbol”



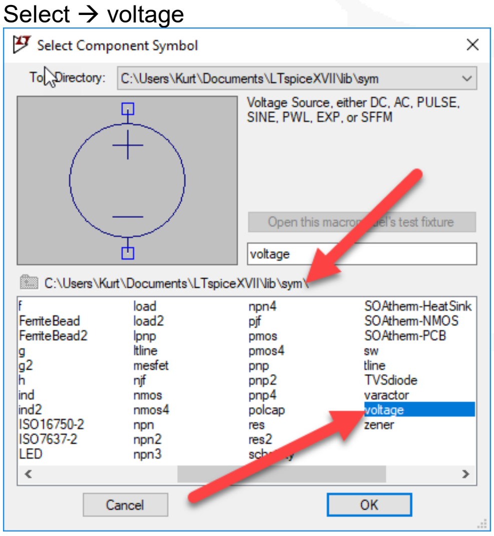
- Write into the field Prefix: x
- Write into the field Value: the tube number



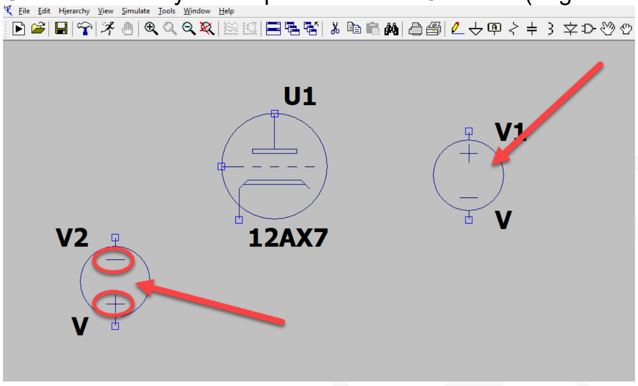
- Now the tube symbol has an number



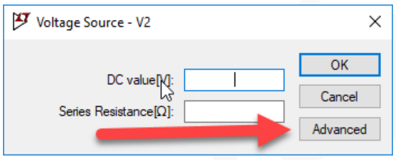
- Open → Component Symbol
- Select → \lib\sim
- Select → voltage



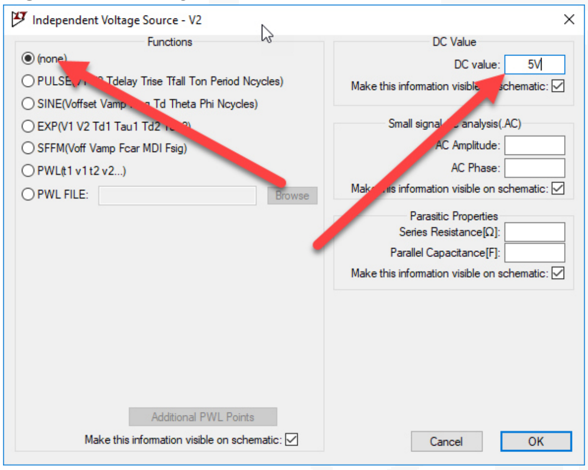
- Place two “voltage symbol on the schema
- Click on → V2 symbol tap two times → Contrl+R (negative connector is now above)



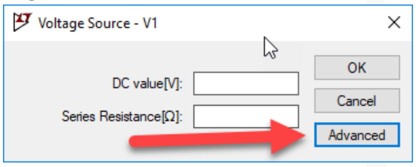
- Click → right mouse button on symbol V2
- Click → Advanced Button



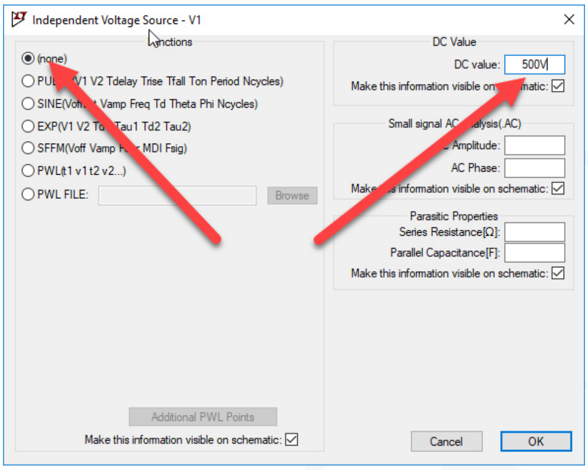
- Select → button (none)
- DC Value: write 5V



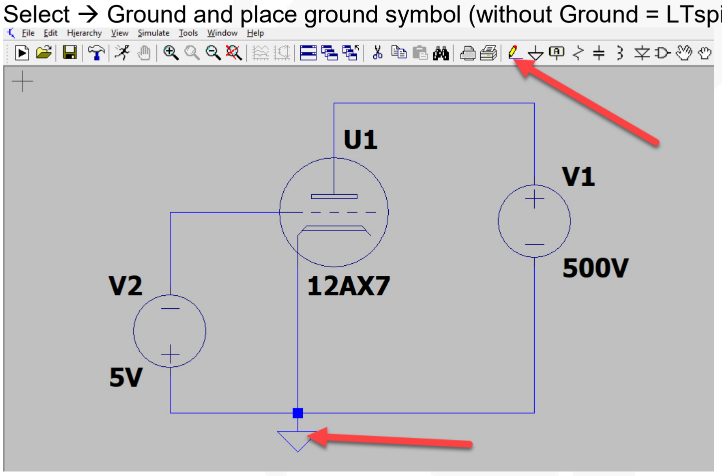
- Click →right mouse button on symbol V1
- Click → Advanced Button



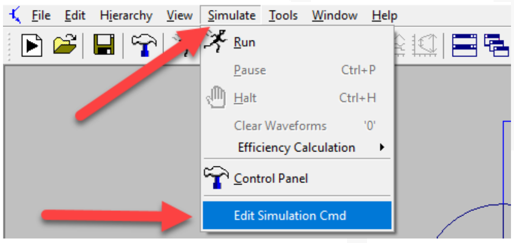
- Select → button (none)
- DC Value: write 500V



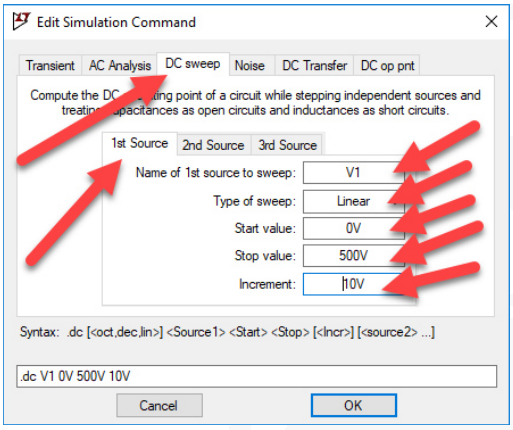
- Select → Wire
- Design wiring like on the schema
- Select → Ground and place ground symbol (without Ground = LTspice can't simulate)



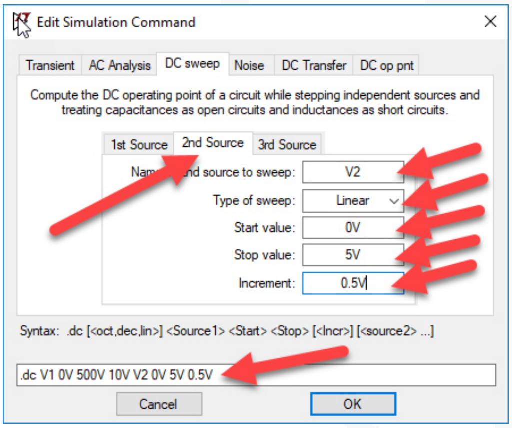
- Select → Simulate
- Select → Edit Simulation Cmd



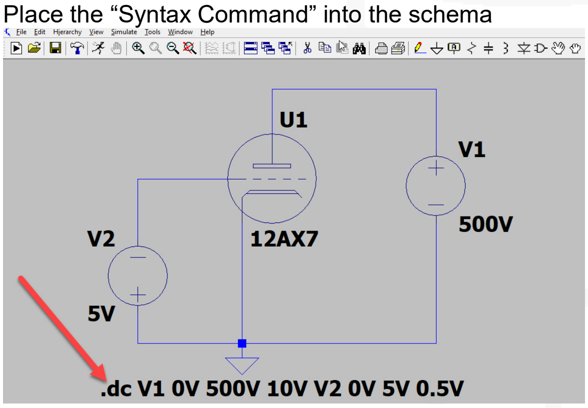
- Edit command for dynamic simulation
- Select →DC sweep → Select → 1st Source
- Name of 1st source to sweep: enter V1 (anode power supply)
- Type of sweep: enter Linear
- Start value: enter 0V
- Stop Value: enter 500V
- Increment: enter 10V



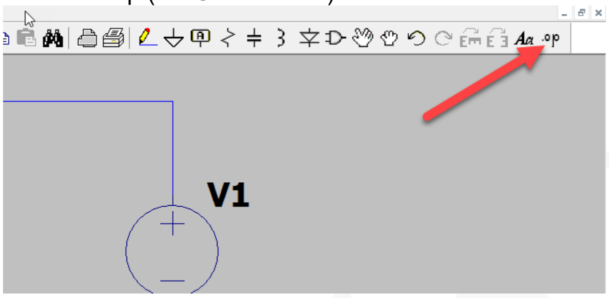
- Select → 2st Source
- Name of 1st source to sweep: enter V2 (grid power supply)
- Type of sweep: enter Linear
- Start value: enter 0V
- Stop value: enter 5V
- Increment: enter 0.5V
- Click → OK button



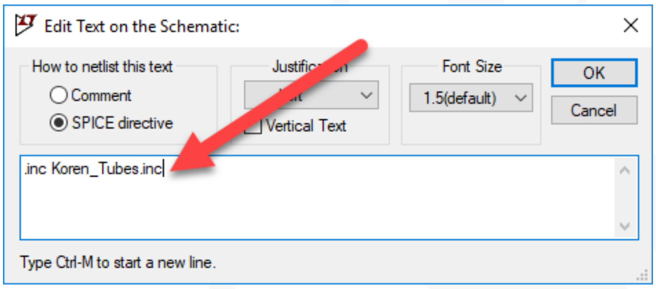
- Place the "Syntax Command" into the schema



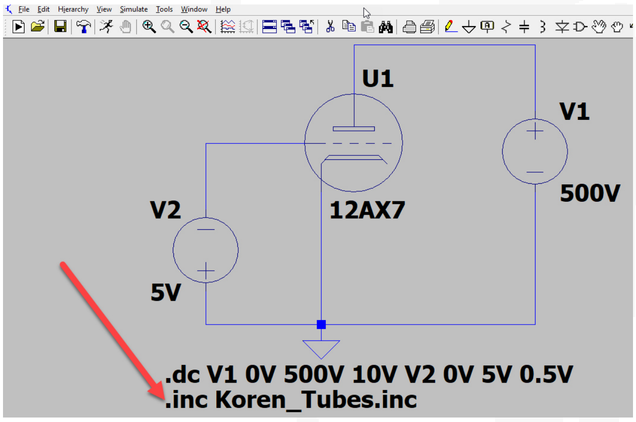
- Create the “link to the tube library”
- Select → .op (SPICE Directive)



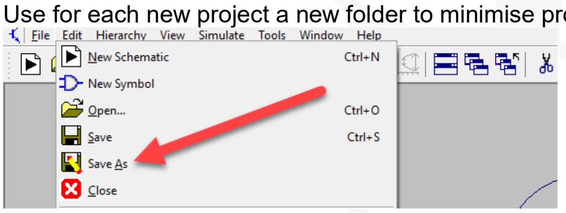
- Write the SPICE directive: inc. Koren_Tubes.inc
- Click → button OK



- Place the “SPICE directive” into the schema



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

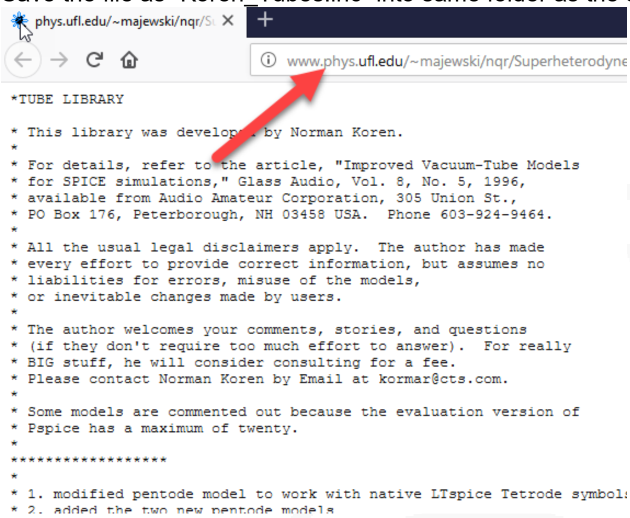


- The saved file looks like this

Test_Triode_Curves_01.asc   30.06.2019 16:36   LTspice Schematic   1 KB

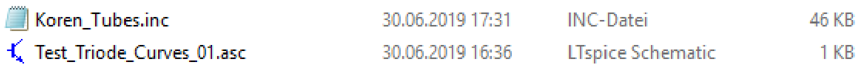
- To simulate the schema, LTspice needs the tube model information
- Find the tube library file on the web:
  http://www.intactaudio.com/forum/download.php?id=1905

- Save the file as “Koren_Tubes.inc” into same folder as the schema

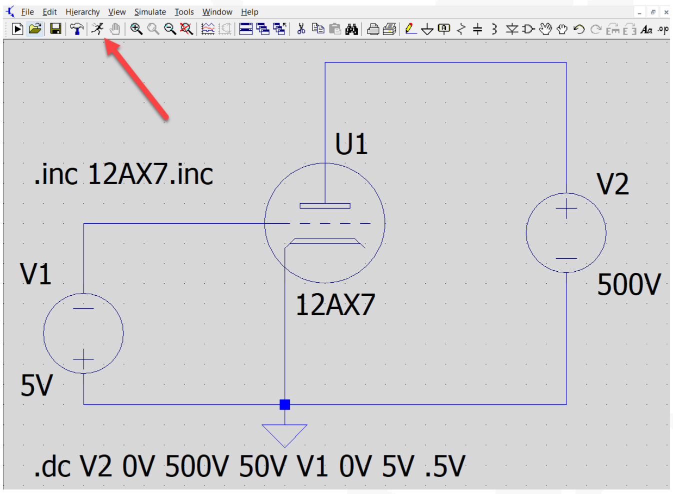



```
*TUBE LIBRARY

* This library was developed by Norman Koren.
*
* For details, refer to the article, "Improved Vacuum-Tube Models
* for SPICE simulations," Glass Audio, Vol. 8, No. 5, 1996,
* available from Audio Amateur Corporation, 305 Union St.,
* PO Box 176, Peterborough, NH 03458 USA.  Phone 603-924-9464.
*
* All the usual legal disclaimers apply.  The author has made
* every effort to provide correct information, but assumes no
* liabilities for errors, misuse of the models,
* or inevitable changes made by users.
*
* The author welcomes your comments, stories, and questions
* (if they don't require too much effort to answer).  For really
* BIG stuff, he will consider consulting for a fee.
* Please contact Norman Koren by Email at kormar@cts.com.
*
* Some models are commented out because the evaluation version of
* Pspice has a maximum of twenty.
*
*****************
*
* 1. modified pentode model to work with native LTspice Tetrode symbol:
* 2. added the two new pentode models
```

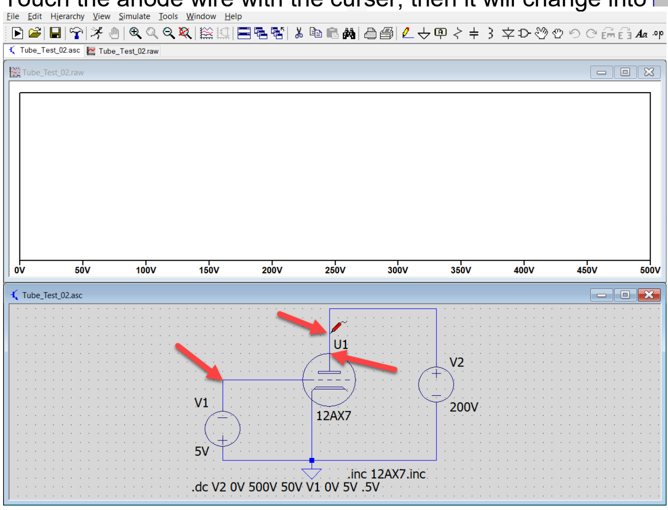
- The two files are saved in the same folder



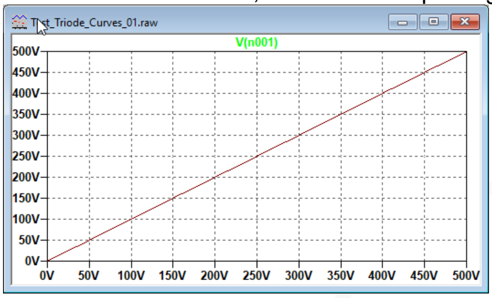
- Now start the simulation
- Press → Run button



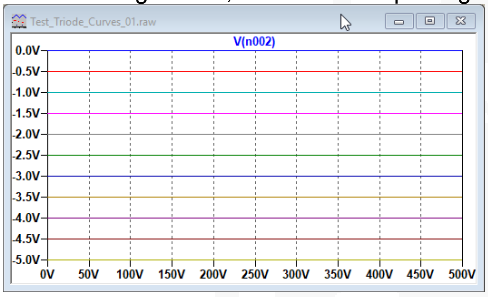
- The desktop looks like this
- Touch the anode wire with the curser, then it will change into  “voltage probe”



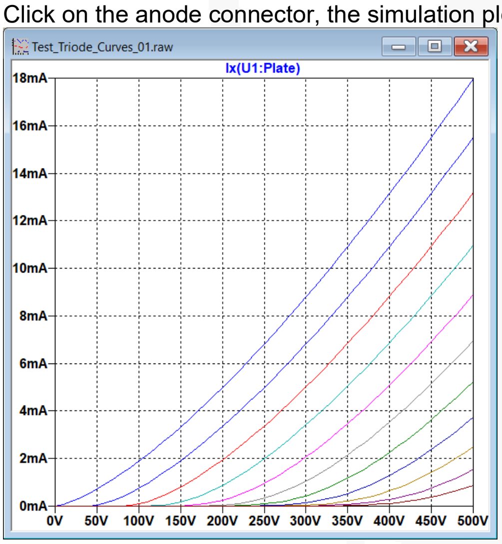
- Click on the anode wire, the simulation plotting of the anode change voltage appears



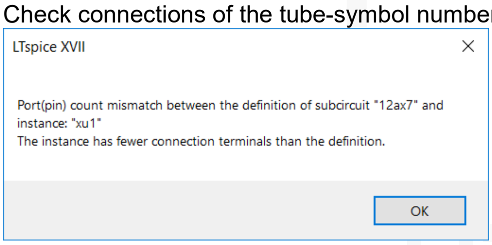
- Click on the grid wire, the simulation plotting of the grid step voltage appears



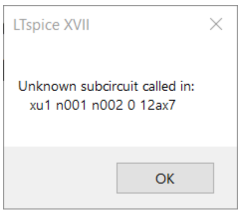
- Touch the tube anode connector the curser will change into  “current probe”
- Click on the anode connector, the simulation plotting of the anode curves appears



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



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



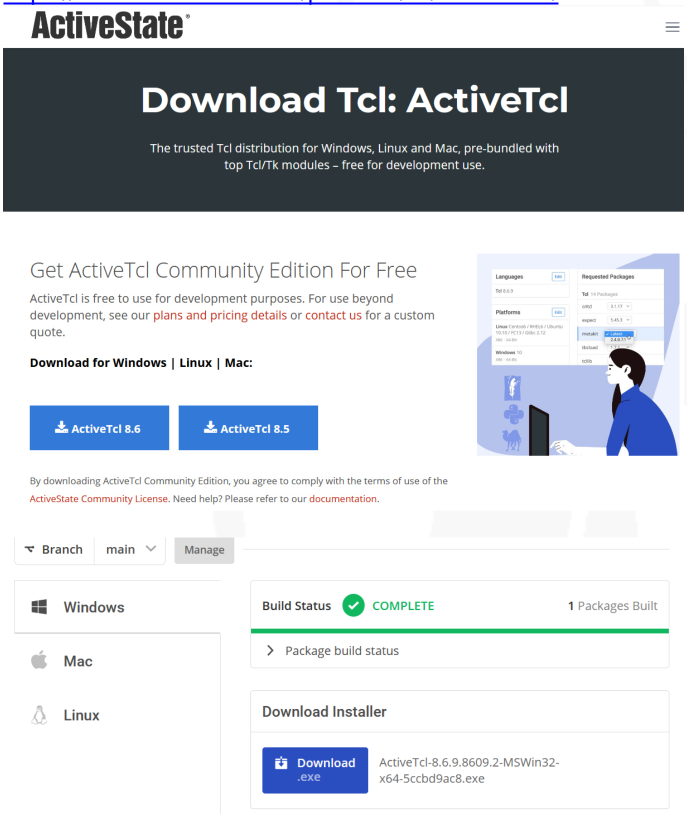
Tube-Model Creation-Software

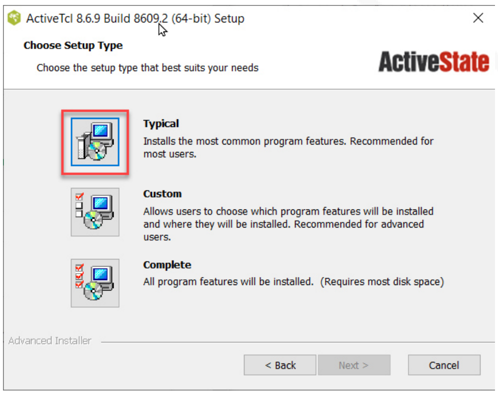
1. Triode Tube-Models created with "Curve Captor" of (Andrei Frolov)

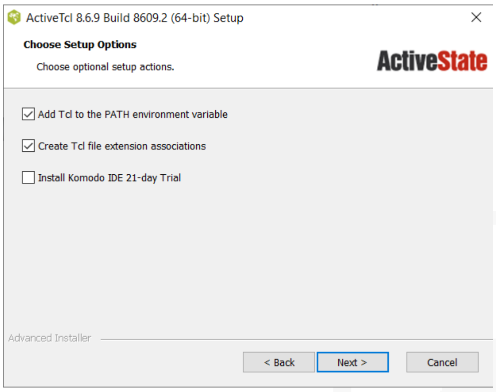
- This program creates in the most simplest way excellent tube models incl. additional information, but is still the beta version since 2013 and has some stumbling blocks.

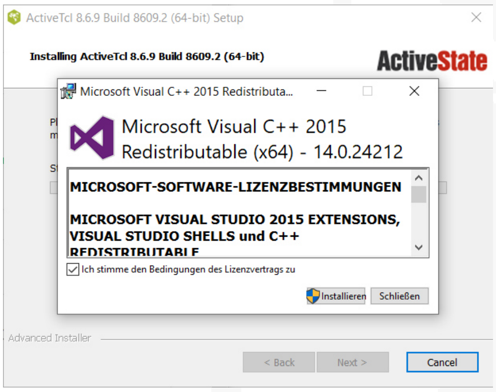
- On YouTube you will find also a Film how to use this program https://www.youtube.com/watch?v=tGPOx9GtXlM

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

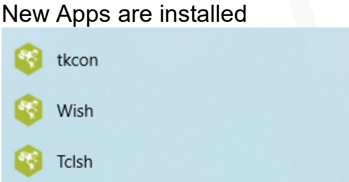




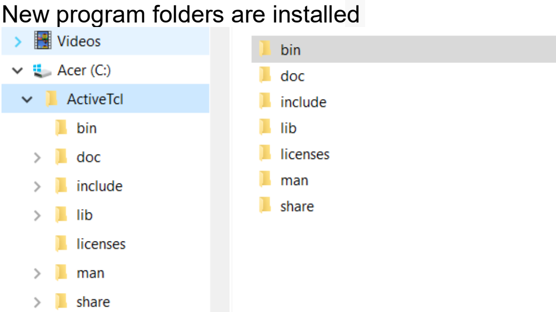




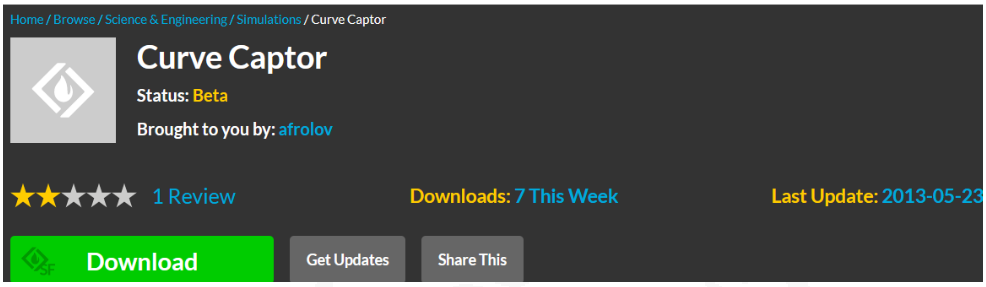
- New Apps are installed

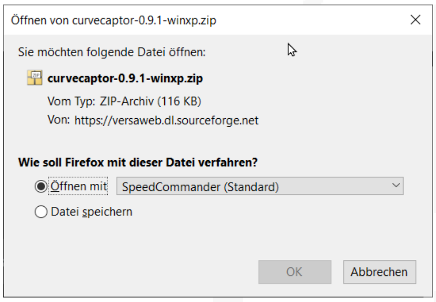


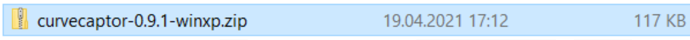
- New program folders are installed



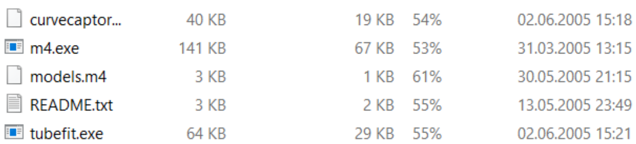
- Download the open source program “Curve Captor” from SOURCDFORGE https://sourceforge.net/projects/curvecaptor/



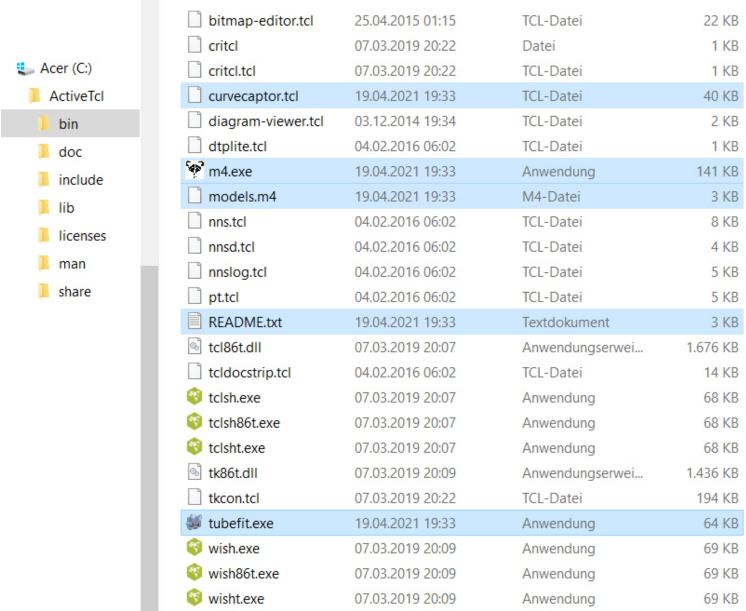




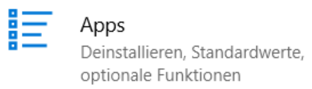
Open zip-folder



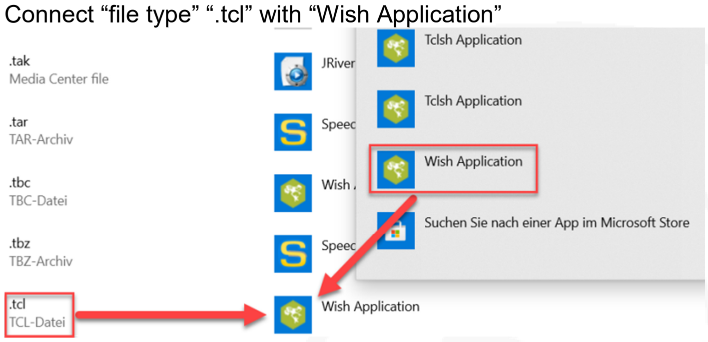
- Copy files into → bin folder



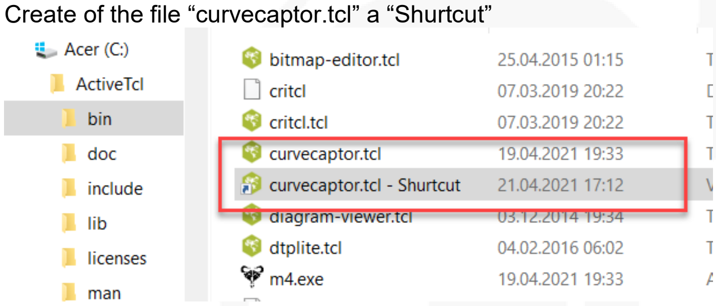
- Open Settings → Apps



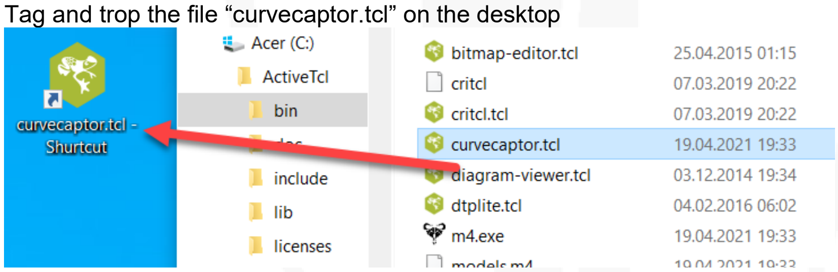
- Connect “file type” “.tcl” with “Wish Application”



- Create of the file “curvecaptor.tcl” a “Shortcut”

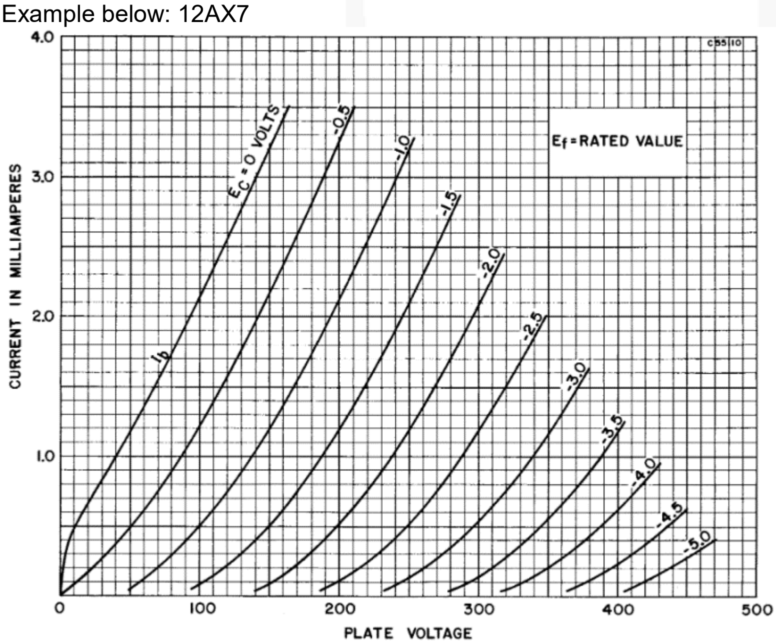


- Tag and trop the file “curvecaptor.tcl” on the desktop

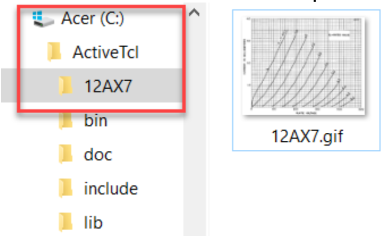


- Now you can start “Curve Captor” from the desktop

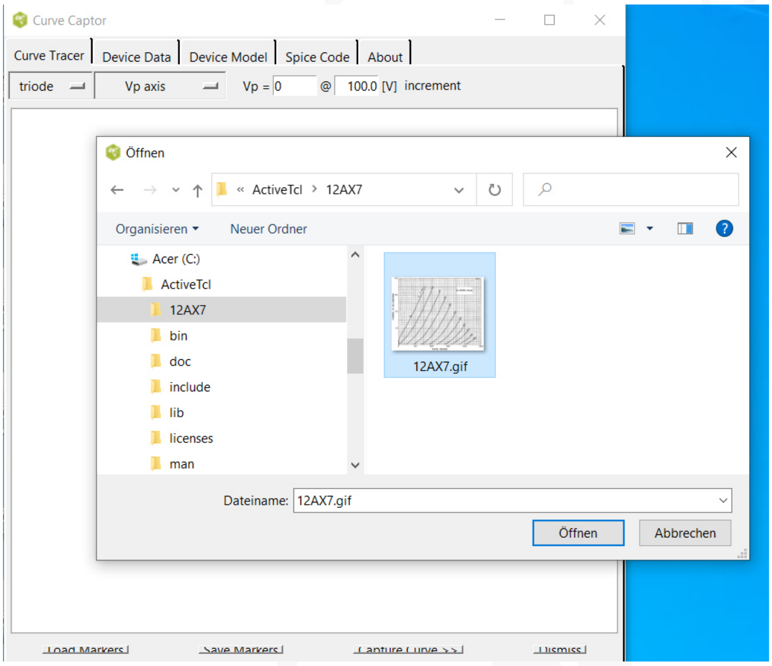
- Prepare a tube curve copy from the manufacture data sheet
max. 750 x 600 pix and as XXX.gif format (only this format accepts Curve Capture)
Example below: 12AX7



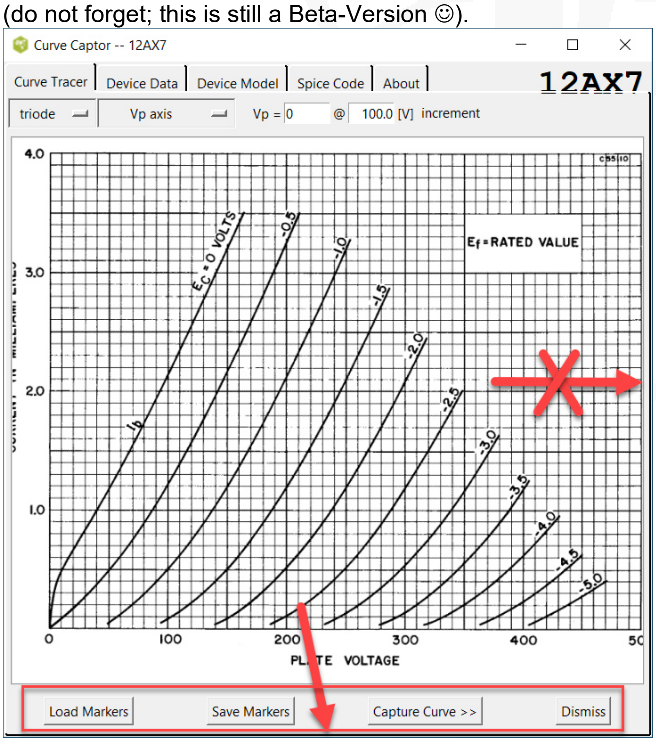
- Save the created file into the path of ActiveTcl



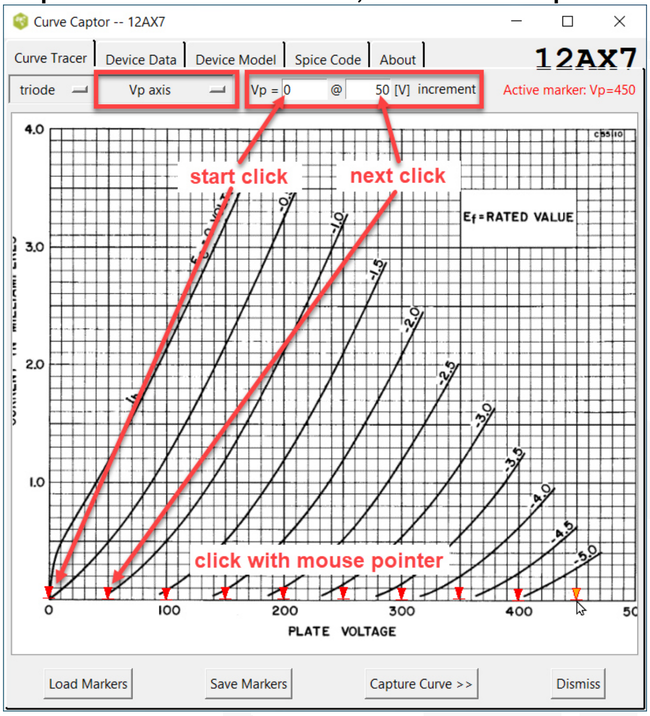
- Start Curve Capture and load the prepared curve copy



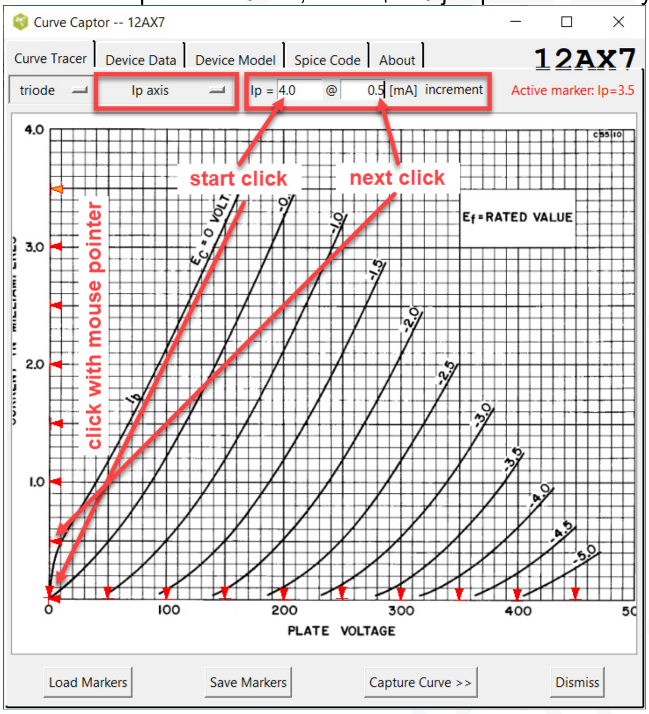
- Never enlarge the window with the left handle or you will start again
- With the bottom handle you can enlarge the window, so you can see the buttons (do not forget; this is still a Beta-Version ☺).

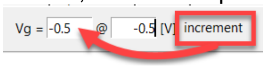


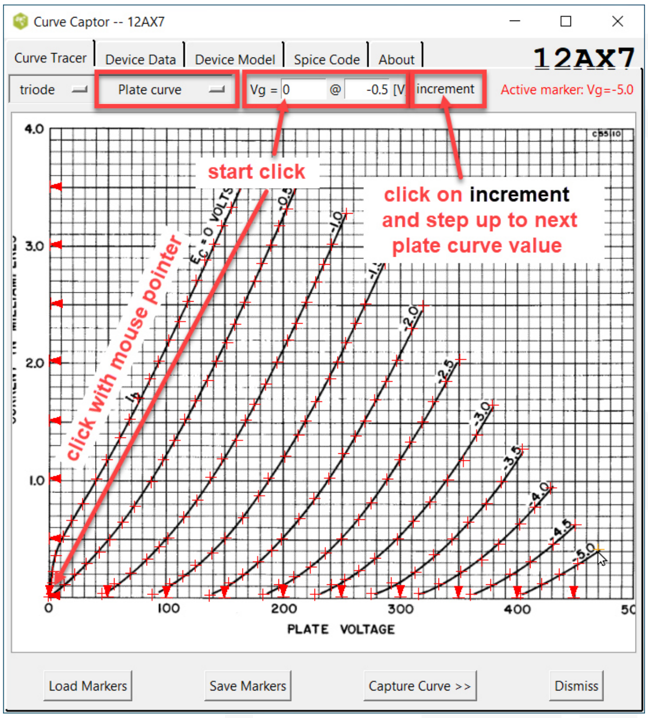
- Set start point for “Volt-Plate Axis (V_P)” “Example below: V_P = 0 @ 50[V] increment”
- Start click with mouse pointer on Plate Voltage = 0 Volt,
  after marker placed to 0 Volt, value V_P = 0 jumps automatically to value V_P = 50 → 100 → 150 → etc.
- **Place the marker precis, if not, no chance to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



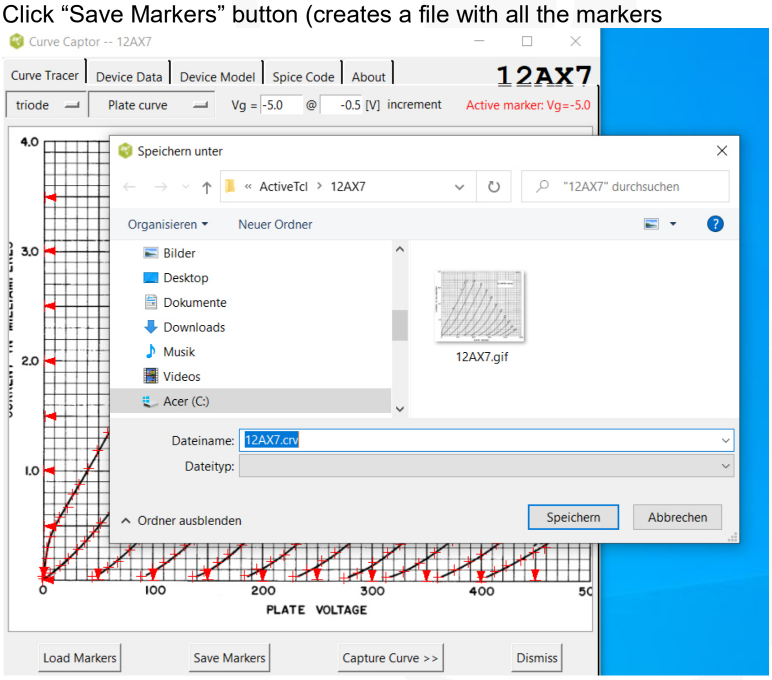
- Set start point for “Current-Plate Axis (I_P)” “Example below: I_P = 0 @ 5[mA] increment”
- Start click with mouse pointer on Plate Current = 0 mA,
  after marker placed to 0 mA, value I_P = 0 jumps automatically to value I_P = 0.5 → 1.0 → 1.5 → etc.



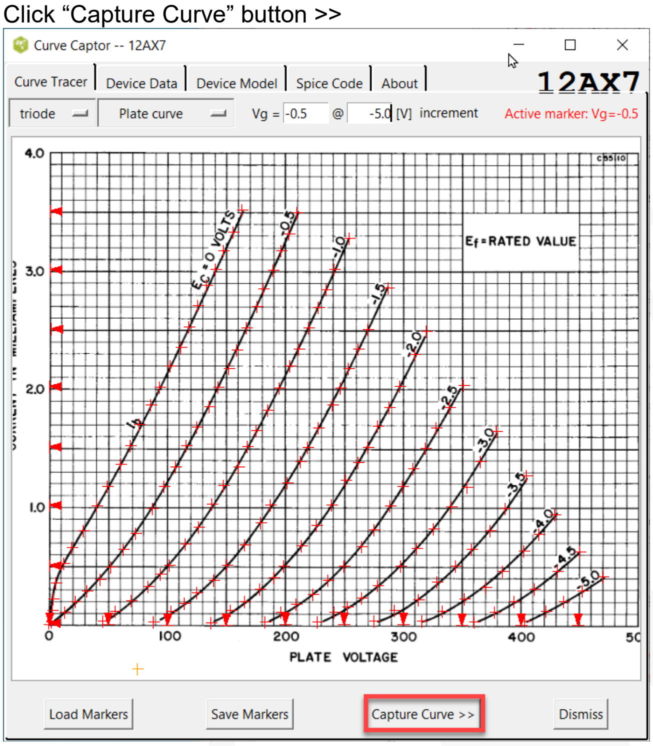
- Set start point for “Plate Curve (V_g)” “Example below: V_g = 0 @ -0.5[V] increment”
- Start click with mouse pointer on Plate Curve = 0 V, after marks placed on plate curve 0V until the end, click on “increment” for the next plate curve  and value of V_g = 0 jumps to value V_g = -0.5 → -1.0 → -1.5 → -2.0→ etc.



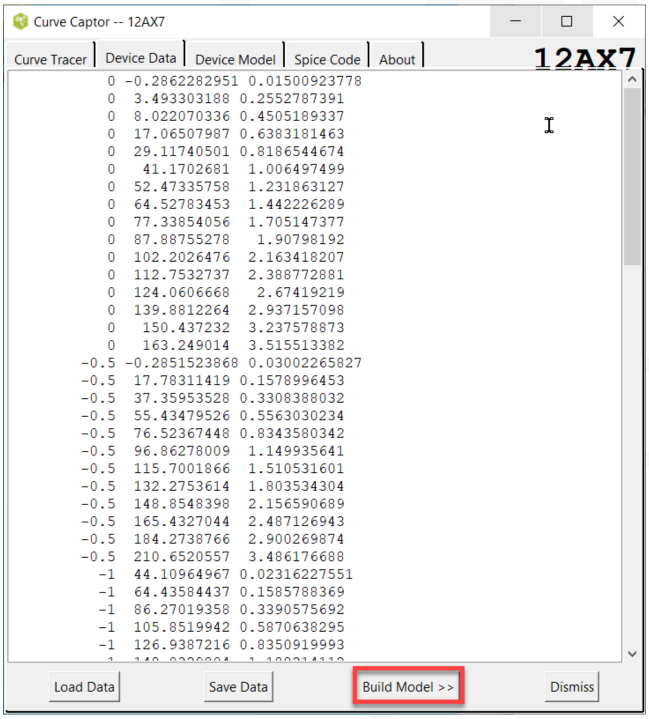
- Click “Save Markers” button (creates a file with all the markers



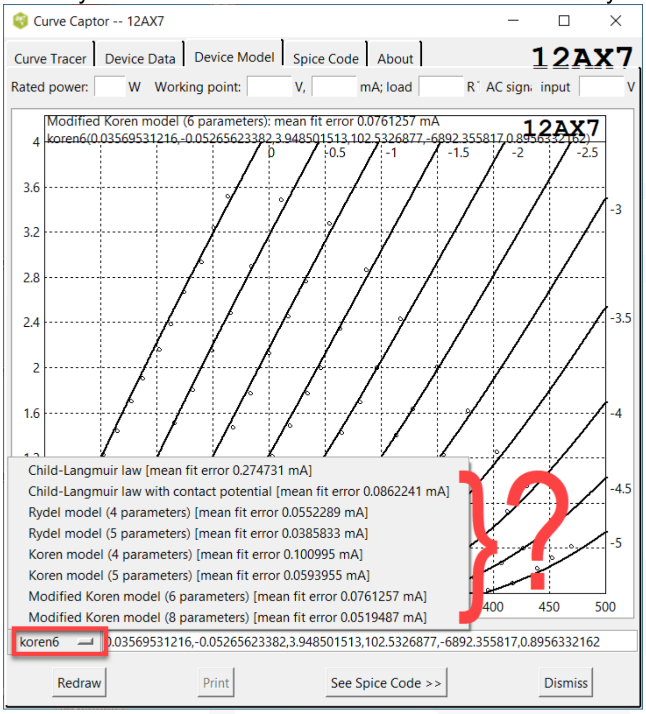
- Click “Capture Curve” button >>



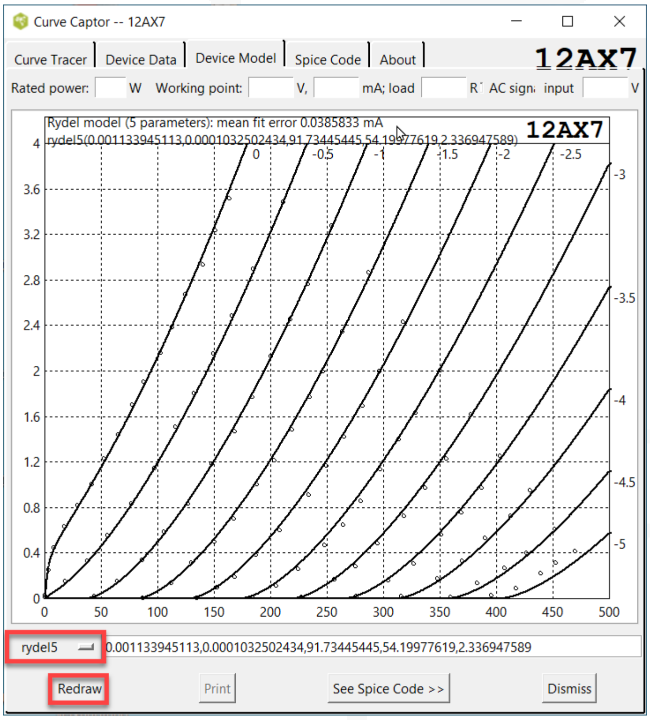
- Curve Capture displays the saved data points
- Cllick “Build Model >>>” button



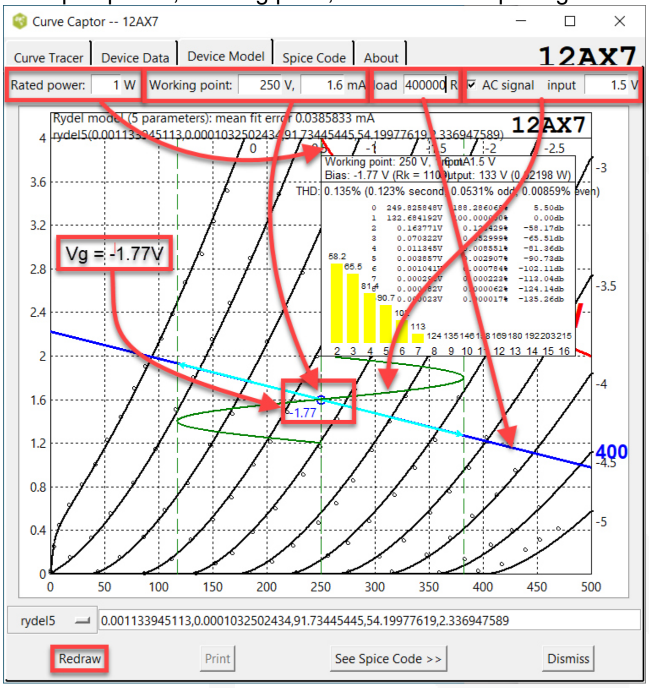
- Click “rydel5” button and check which “Device Model fits your original curves the best



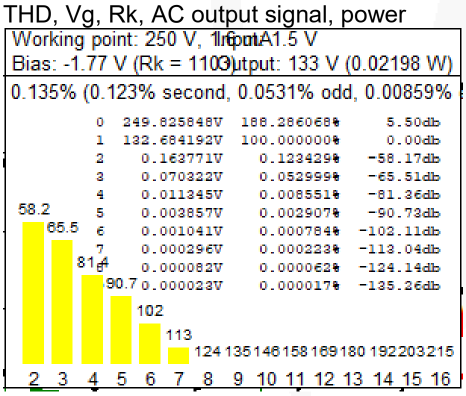
- Click button “Redraw” and see “bad fit” of model “korean6”



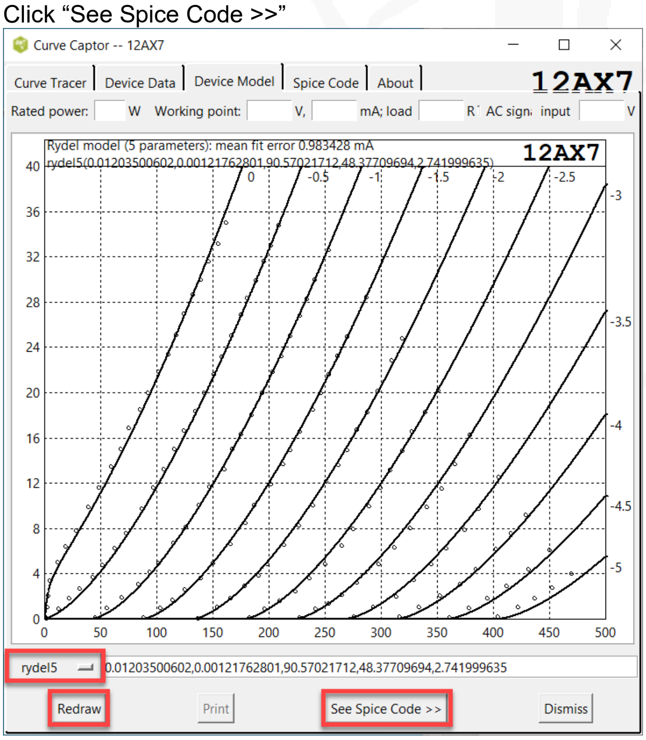
- Get additional information, when enter:
  max. plat power, working point, load and AC input signal



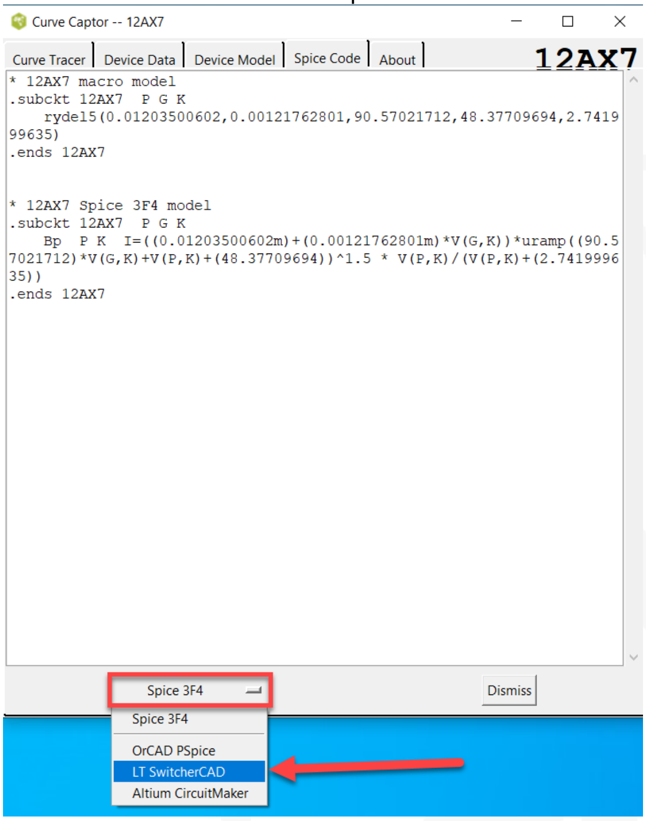
- Get additional calculated information of desired working point, load and input signal
  THD, Vg, Rk, AC output signal, power



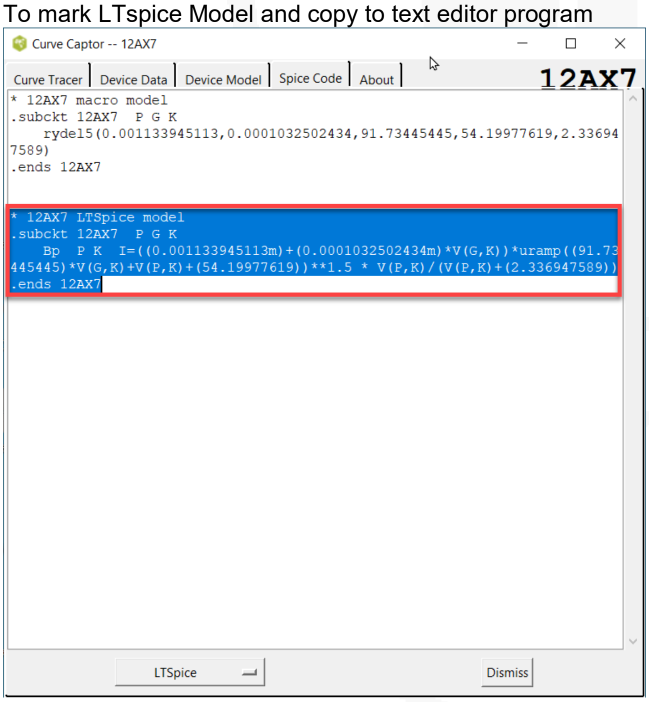
- Best fit of curves is model "rydel5"
- Click "See Spice Code >>"



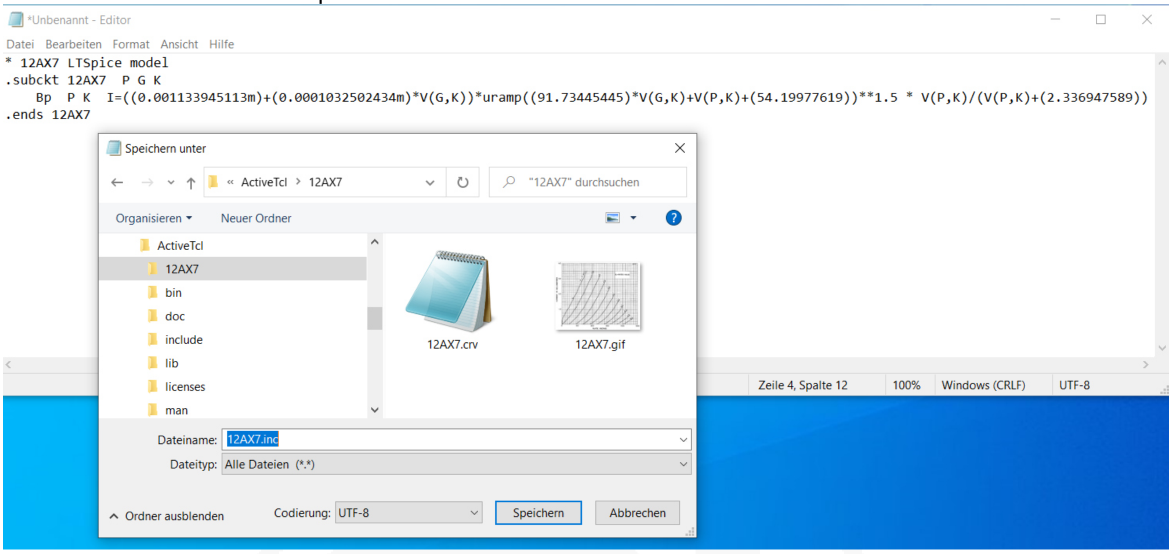
- Klick “Spice SF4” button
- Select “ LT SwitcherCAD” = LTspice ☺



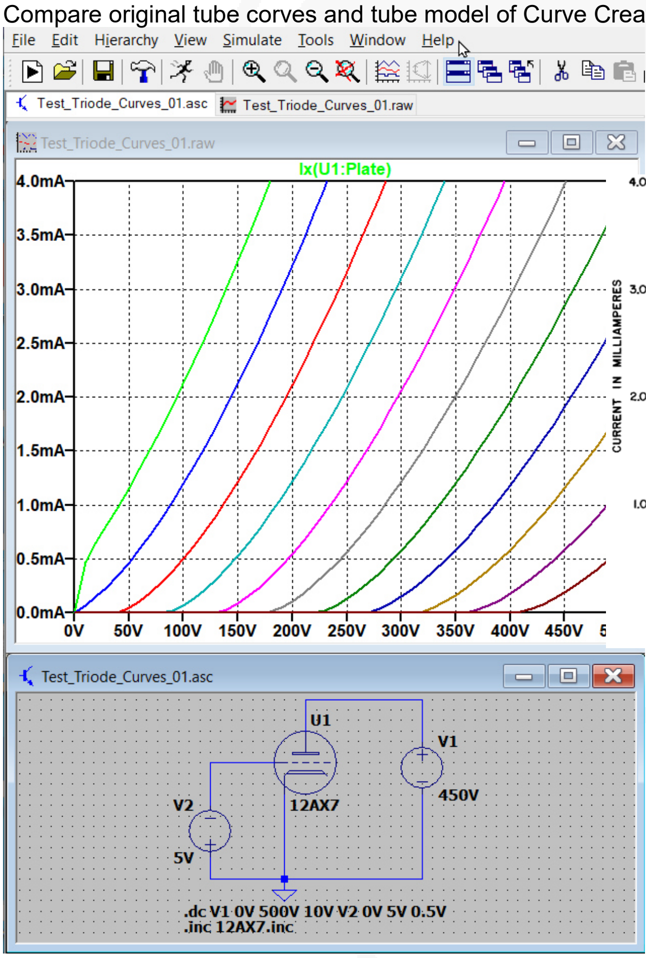
- To mark LTspice Model and copy to text editor program

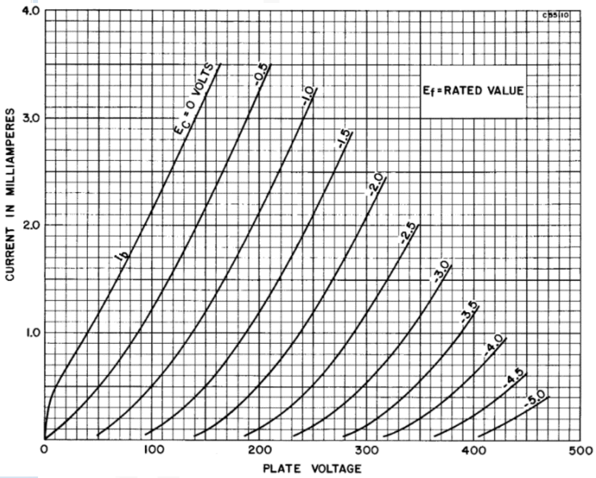


- Save tube model file as example XXXX.inc



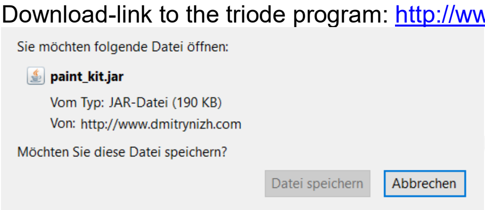
- Compare original tube corves and tube model of Curve Creator





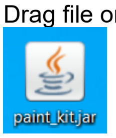
2. Triode Tube-Models created with “Model Paint Tools” (Dmitry Nizhegorodov)

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

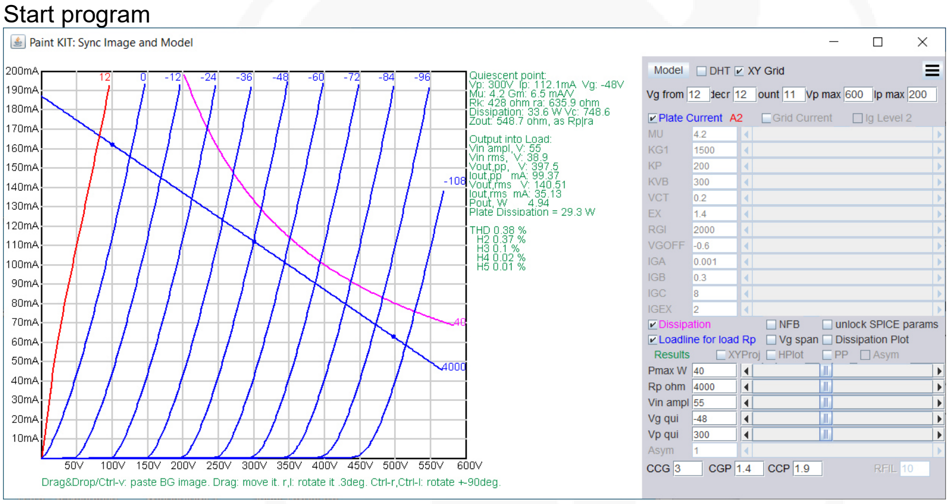


- Find program file in “Downloads-Folder” (if possible, virus check !)

- Drag file on desktop for easy start up



- Start program



- Tube specification (KR 300BXLS)

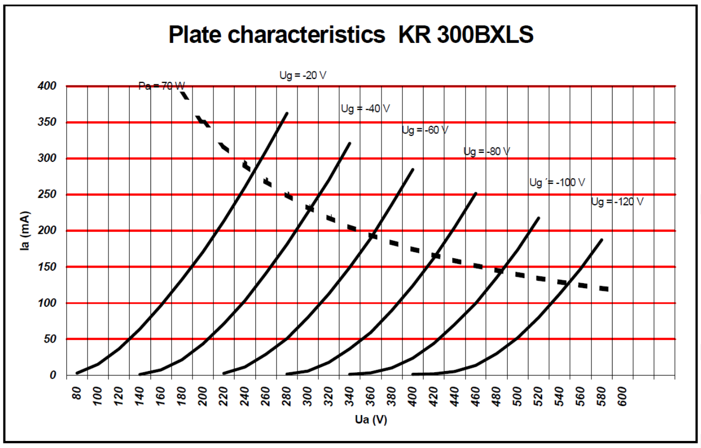
Power Amplifier class A1

Maximum

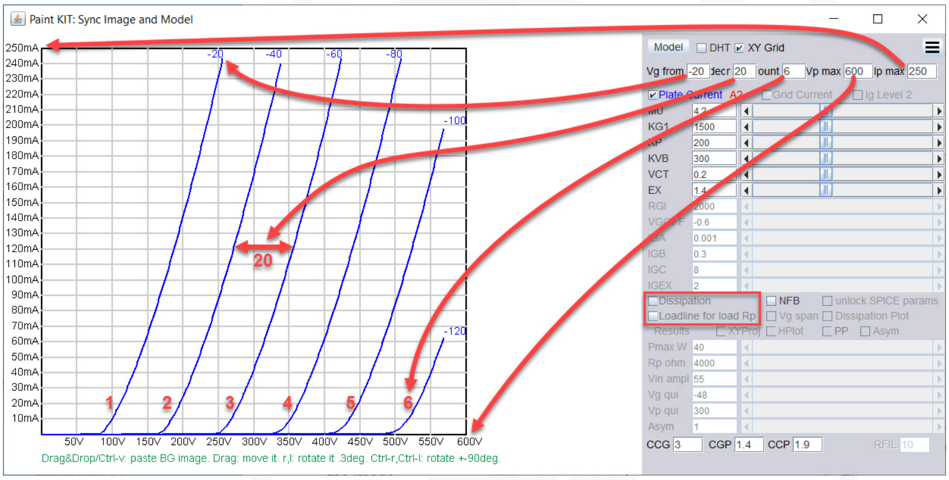
DC Plate Voltage	600	V
Dissipation	70	W
DC Plate Current	160	mA

DC Plate Voltage	450	V
DC Plate Current	100	mA
Grid Voltage	-94	V
Plate Resistance	650	ohm
Transconductance	5,7	mA/V

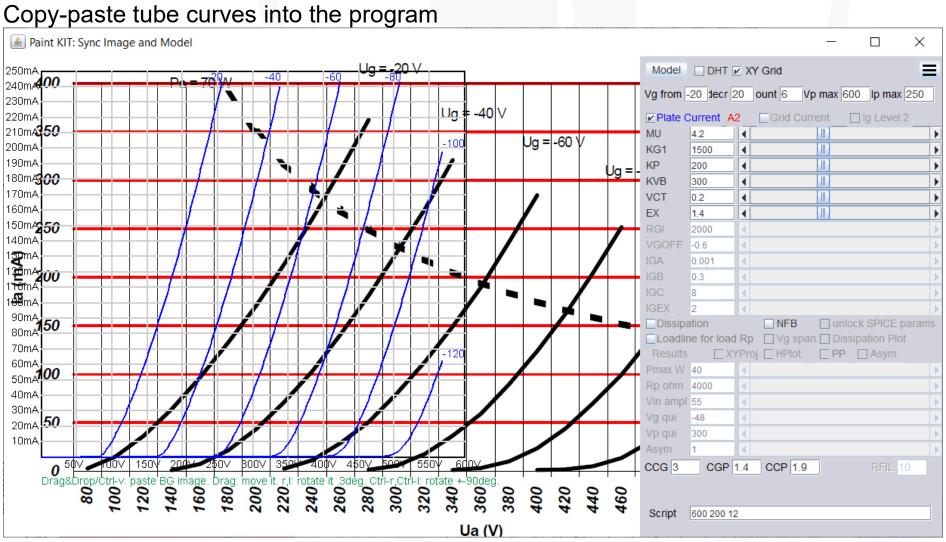
- Capture tube curves from datasheet



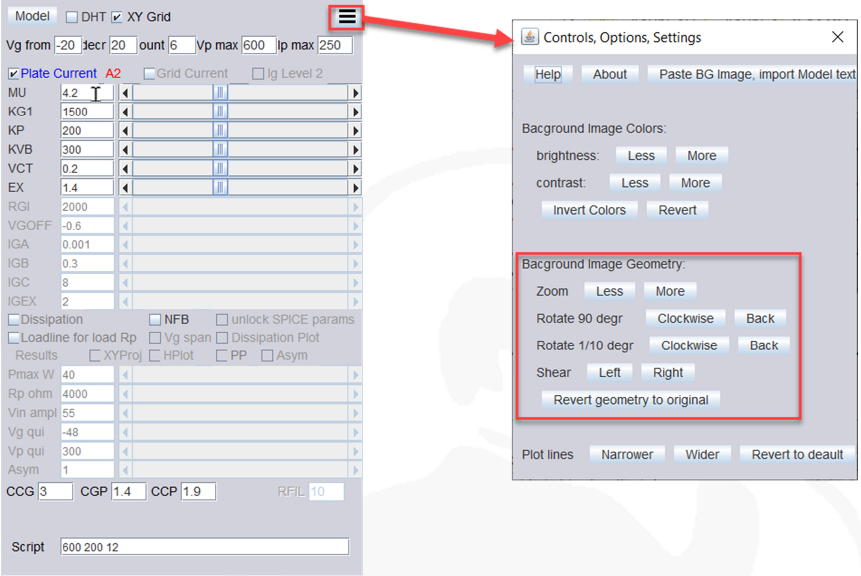
- Enter tube curve datasheet values into the boxes



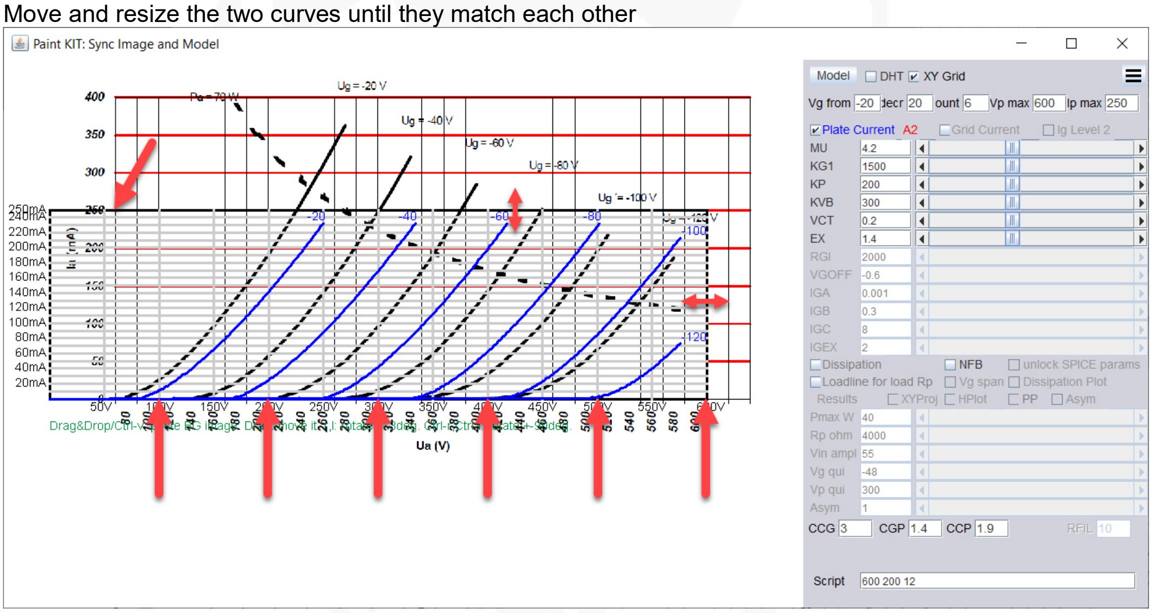
- Copy-paste tube curves into the program



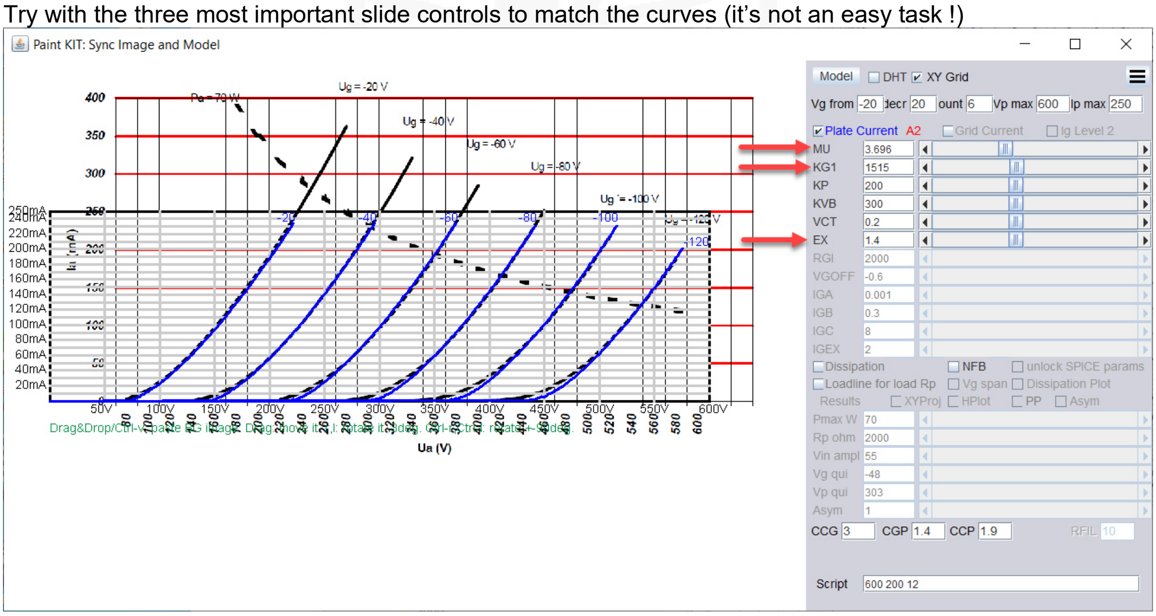
- Open “Control, Options, Settings” page
- Zoom in the past curves with “Zoom Less Button”



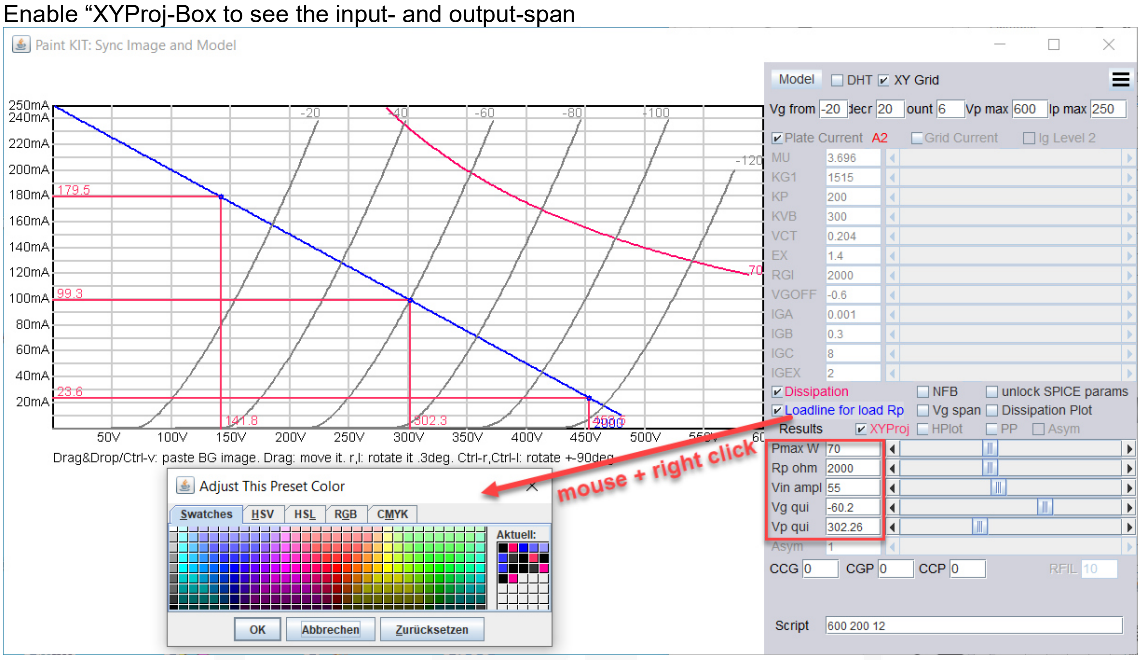
- Move and resize the two curves until they match each other



- Try with the three most important slide controls to match the curves (it’s not an easy task !)



- 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

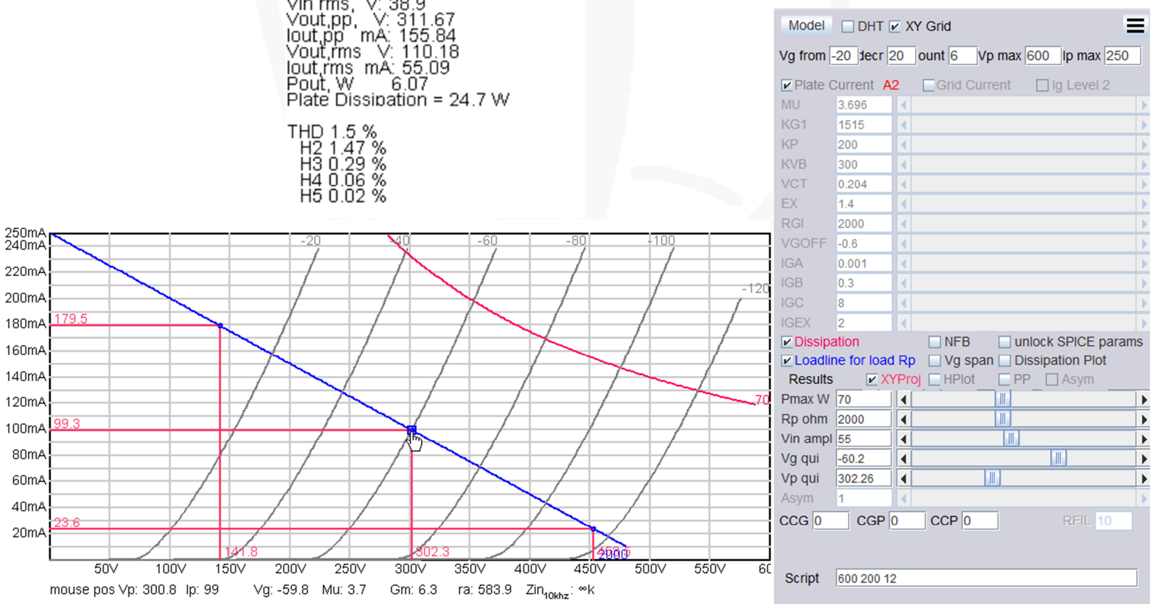


- Check the calculated tube specification, working point and distortion parameters

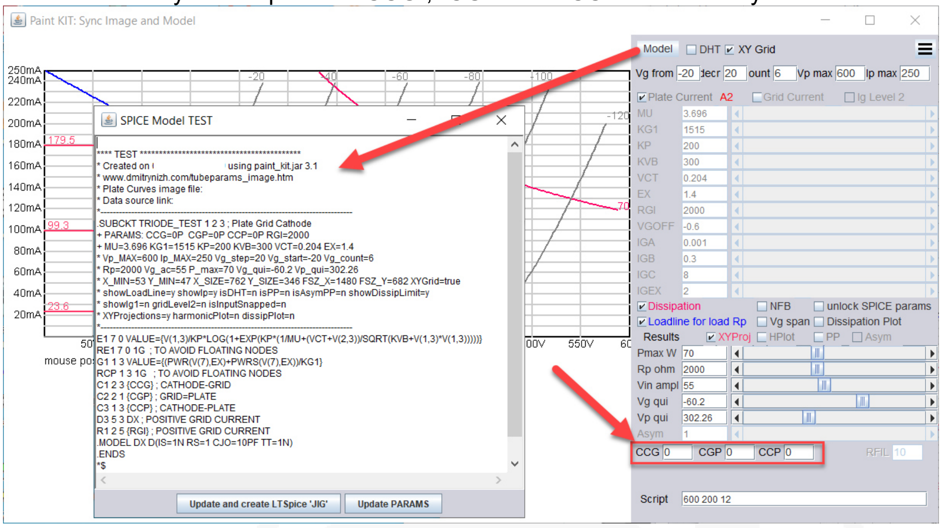
Quiescent point:
Vp: 302.3V Ip: 99.3mA Vg: -60.2V
Mu: 3.6 Gm: 6.2 mA/V
Rk: 606.5 ohm ra: 584.4 ohm
Dissipation: 30 W Vc: 500.8
Zout: 452.2 ohm, as Rp|ra

Output into Load:
Vin ampl, V: 55
Vin rms, V: 38.9
Vout,pp, V: 311.67
Iout,pp mA: 155.84
Vout,rms V: 110.18
Iout,rms mA: 55.09
Pout, W 6.07
Plate Dissipation = 24.7 W

THD 1.5 %
H2 1.47 %
H3 0.29 %
H4 0.06 %
H5 0.02 %



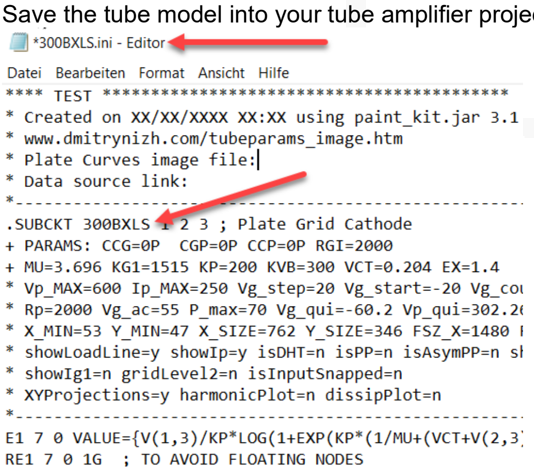
- Create the LTspice “tube model” of the tube KR 300B XLS
- Enter the tube system capacities “CCG”, “CGP” and “CCP” if necessary



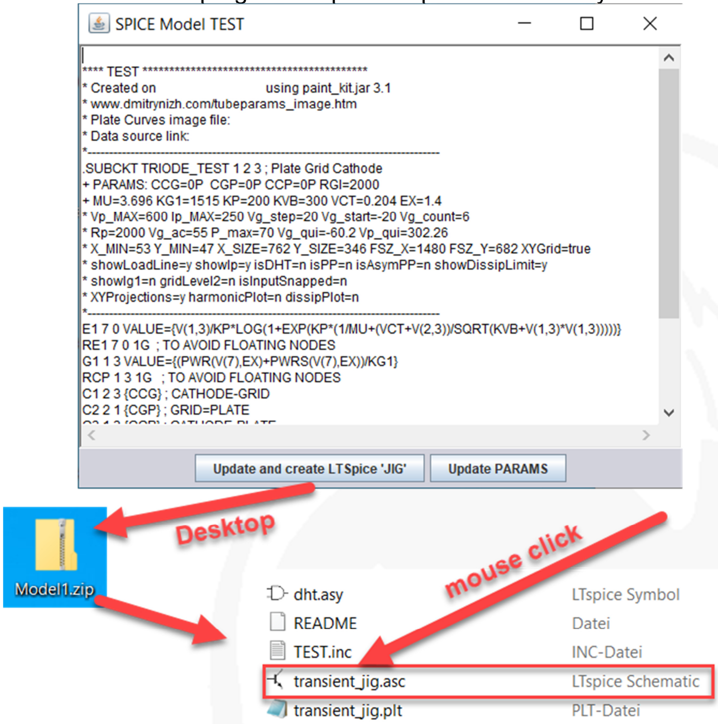
- Copy the “tube model” text into a “editor program”
- Enter the correct tube model name “300BXLS”, so LTspice can find it when simulate the amplifier

```
**** TEST ******************************************
* 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 Ip_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
* 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))/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
*$
```

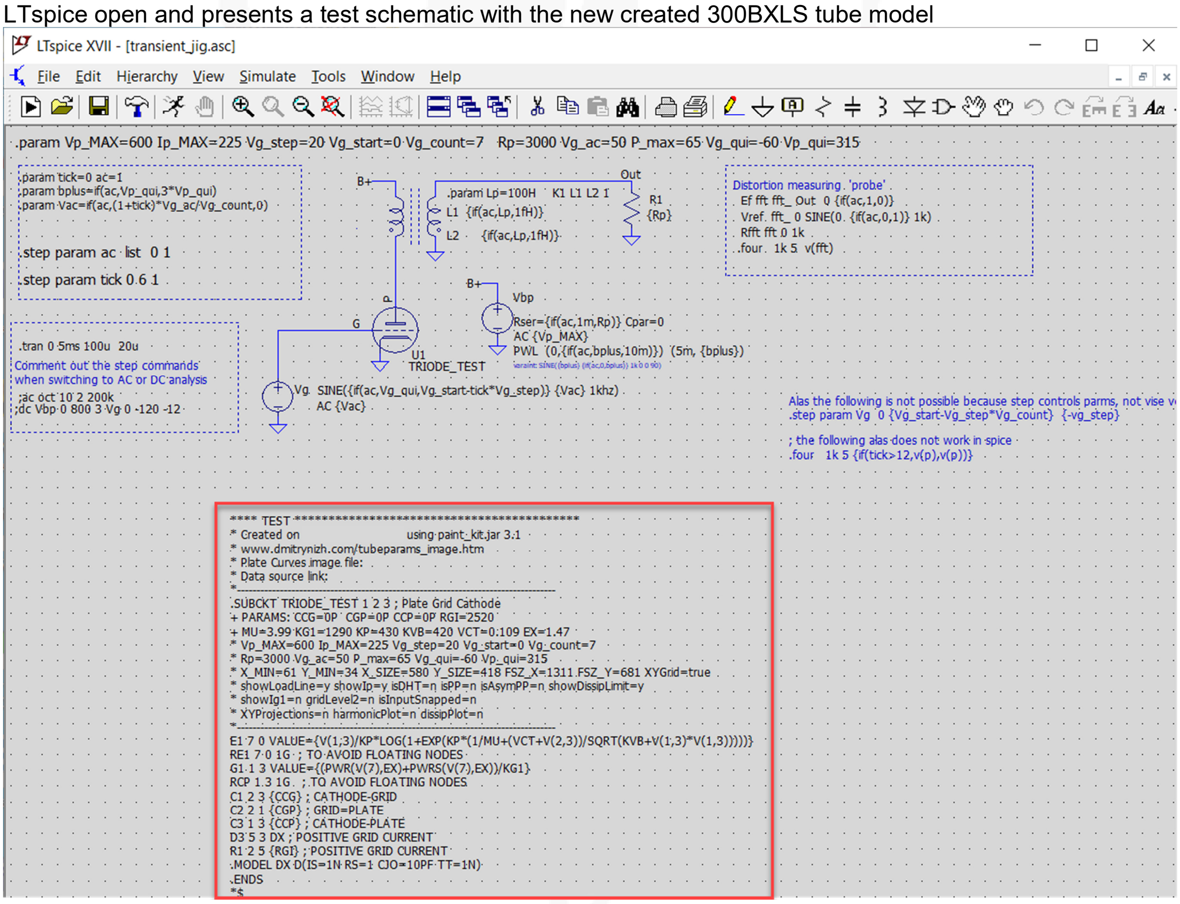
- Save the tube model into your tube amplifier project folder



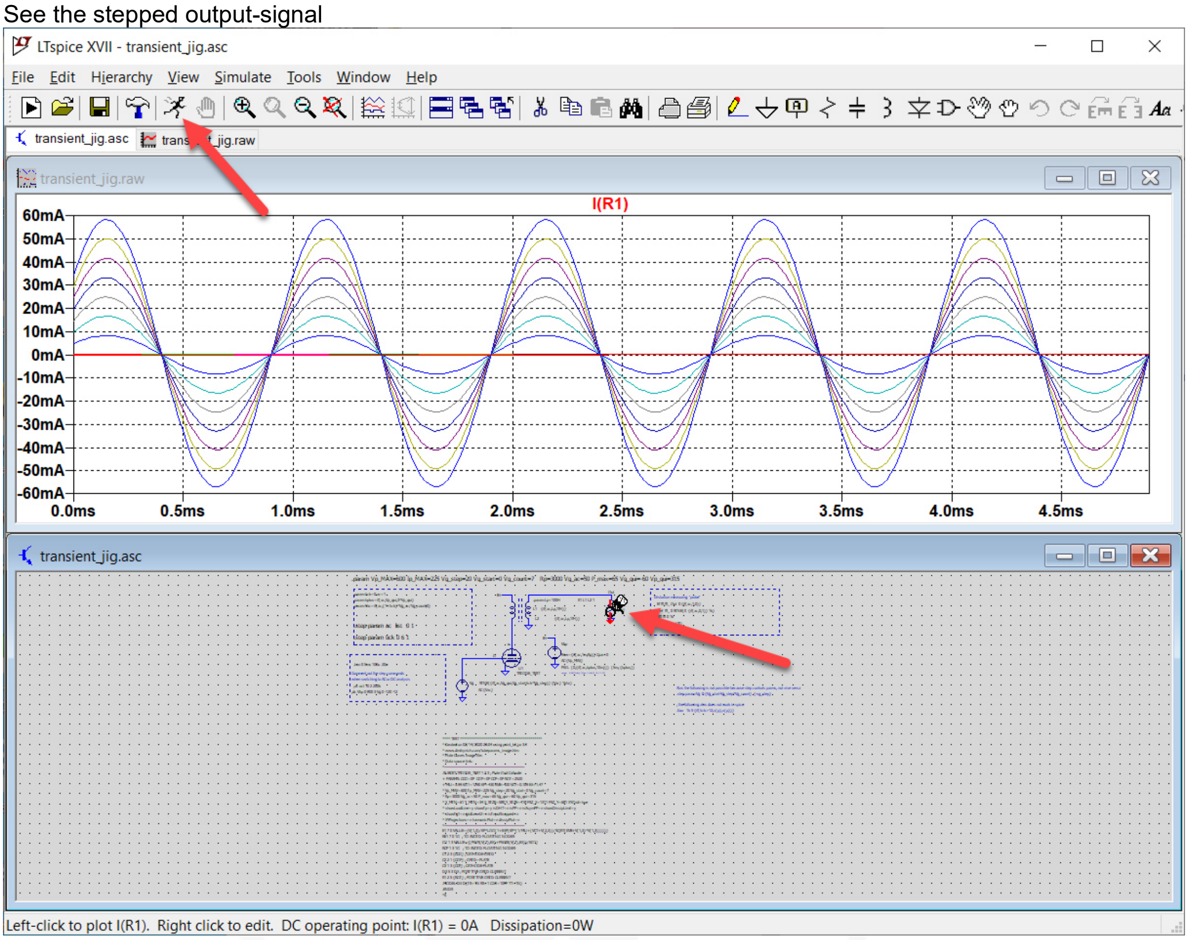
- Push button “Update and create LTspice ”JIG”
- Creates an icon on the desktop
- Click the icon and program “Explorer” open automatically and click the file “transient_jig.asc



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

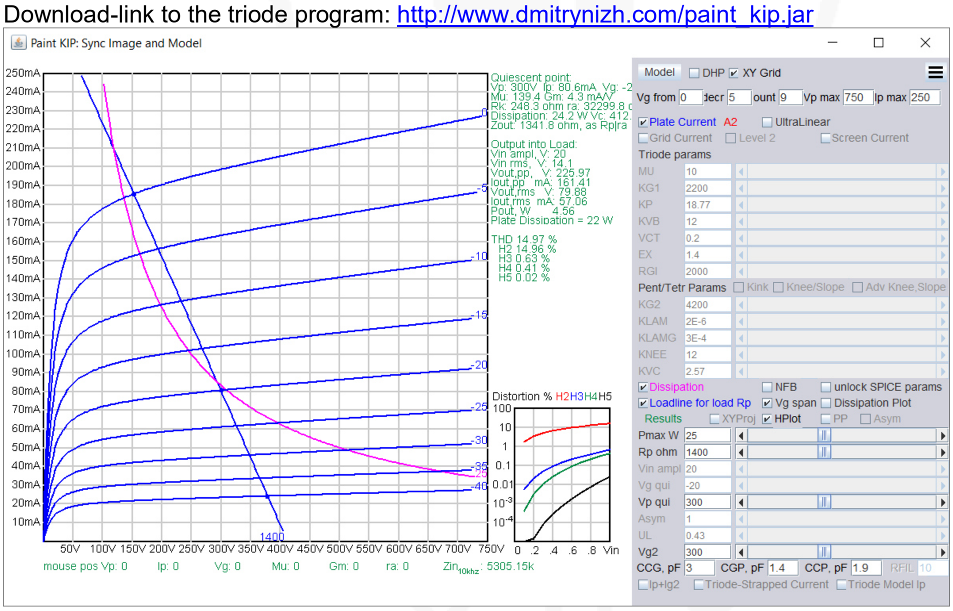


- Click “Simulation Button” and click the out coil with the mouse
- See the stepped output-signal



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

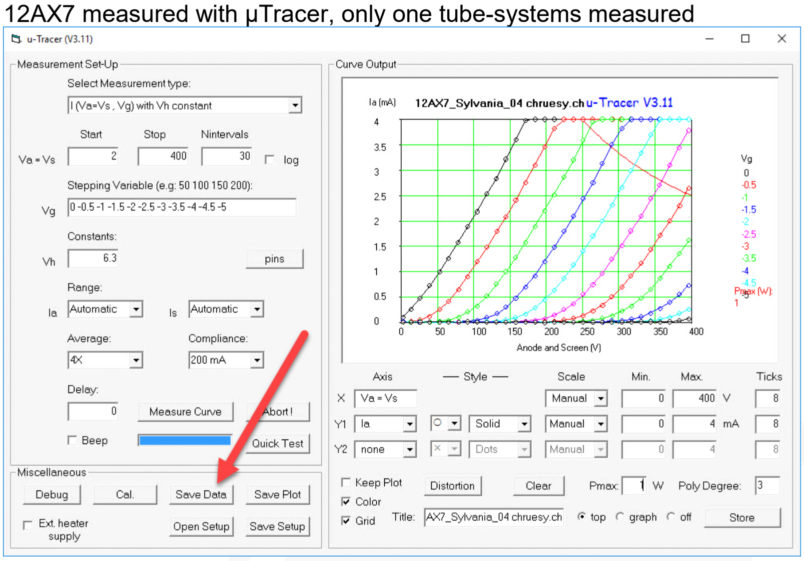
- Download-link to the triode program: http://www.dmitrynizh.com/paint_kip.jar



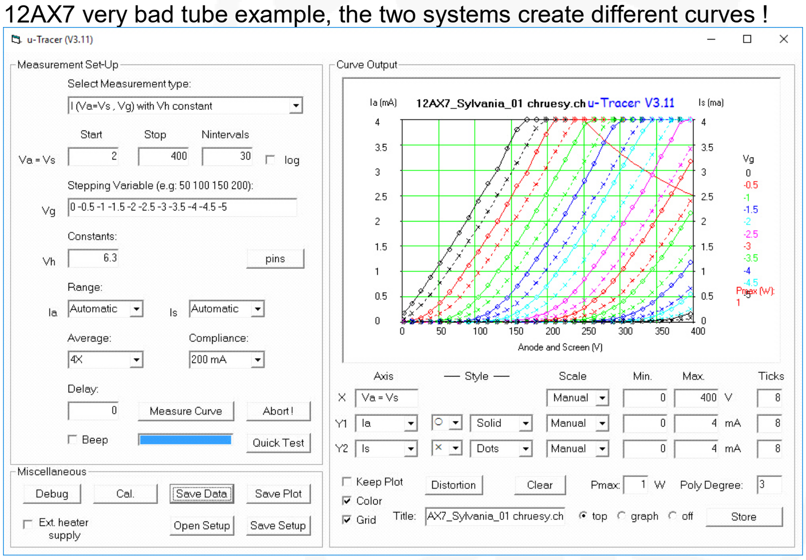
- Sorry, I tried many times to create pentode curves for KT88, EL84, EL34 etc. but didn’t get it, maybe you do can !

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

- 12AX7 measured with µTracer, only one tube-systems measured



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



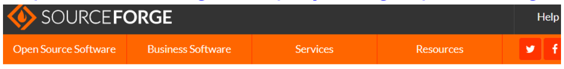
- 12AX7 measured Data of µTracer

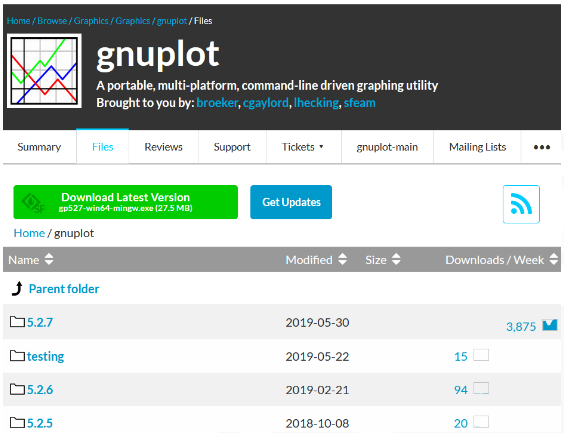
12AX7_Sylvania_04_single.utd - Editor

Datei Bearbeiten Format Ansicht ?

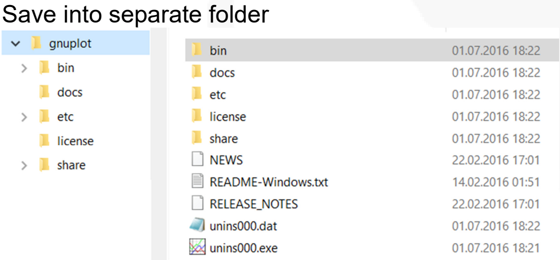
Point	Curve	Ia (mA)	Is (mA)	Vg (V)	Va (V)	Vs (V)	Vf (V)
1	1	0.11	0.01	0	2.91	3.07	6.3
2	1	0.26	0.01	0	13.71	13.95	6.3
3	1	0.48	0.01	0	25.82	27.02	6.3
4	1	0.73	0.01	0	39.22	40.51	6.3
5	1	0.99	0.01	0	52.62	53.58	6.3
6	1	1.27	0.01	0	65.16	66.2	6.3
7	1	1.58	0.01	0	78.56	79.27	6.3
8	1	1.89	0.01	0	91.96	93.2	6.3
9	1	2.18	0.01	0	104.49	105.83	6.3
10	1	2.54	0.01	0	117.89	119.32	6.3
11	1	2.84	0.01	0	131.72	132.39	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

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

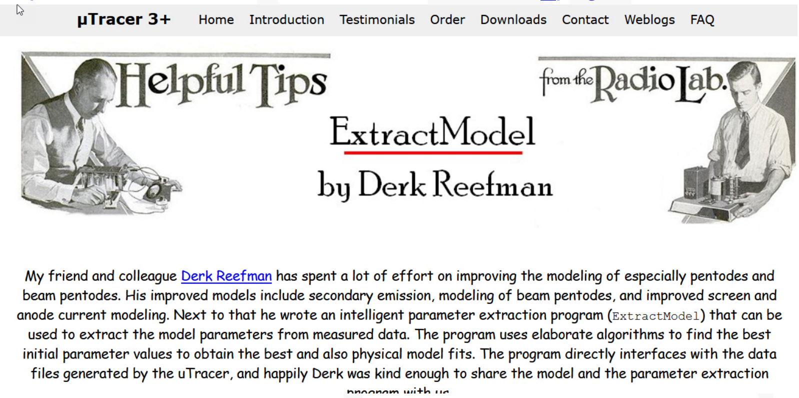




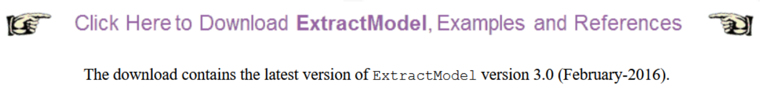
- Save into separate folder



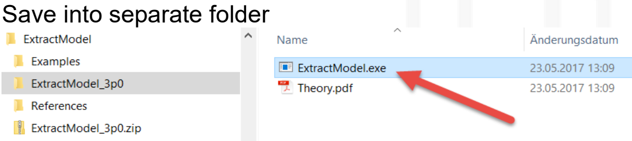
- Open ExtractModel webpage from Derk Reefman
https://www.dos4ever.com/uTracer3/uTracer3_pag14.html



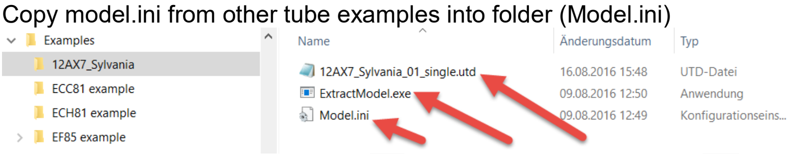
- Download ExtractModel



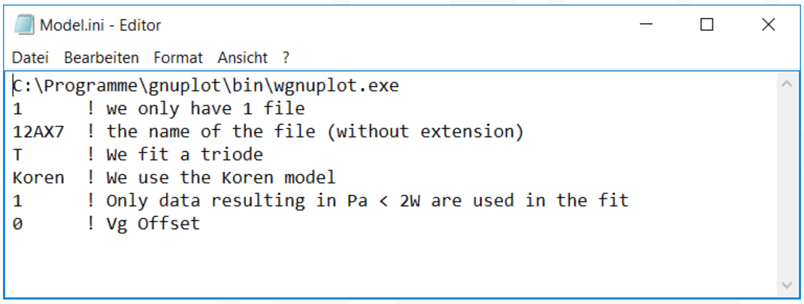
- Save into separate folder



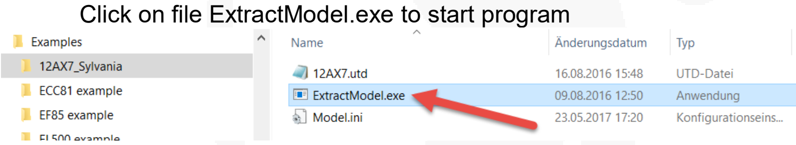
- Create new folder for tube-module 12AX7
- Copy tube-curve µTracer data into folder (XXXXXXX.utd)
- Copy Extract Model program into folder (ExtractModel.exe)
- Copy model.ini from other tube examples into folder (Model.ini)



- 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



- After start of program press “Return”

- Program-Information -→ just press “Return”

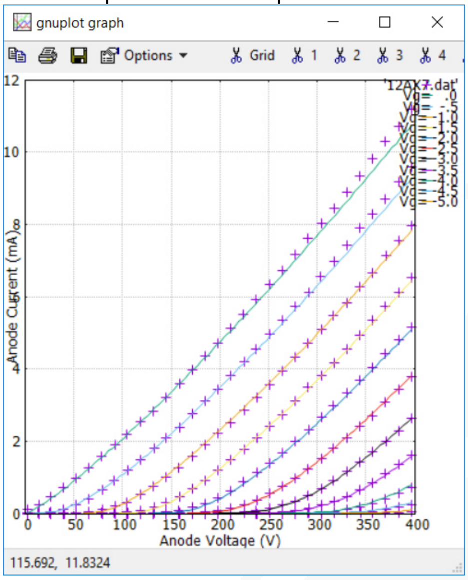
E:\Electronic\Computer\Programme\Curve Captor\ExtractModel\Examples\12AX7_Sylvania\ExtractModel.exe

```
**************************
      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.

!!!!!!
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.

Saturation effects detected for datafile 1 for Vg =    .0.

Saturation effects detected.
Fit will continue - but results may be untrustworthy.
Hit return to continue.
```

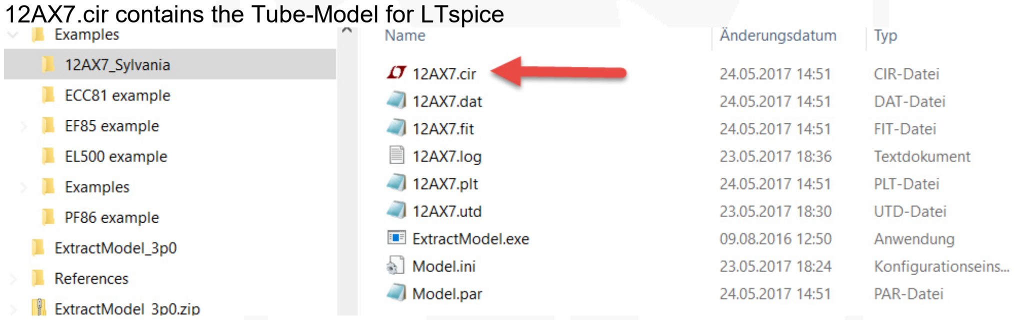
- ExtractProgram calculate the Korean LTspice code

```
**************************
      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.

!!!!!!
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.

Saturation effects detected for datafile 1 for Vg =    .0.

Saturation effects detected.
Fit will continue - but results may be untrustworthy.
Hit return to continue.
Initial estimates:

kVB =       8.468101
kG1 =     562.428500
kp  =      50.000000
ex  =       1.226632
mu  =      89.178250

Before refinement R2 =    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 add capacitances yourself!

Elapsed time:   5.0s
Final estimates:

kVB =    2278.915000
kG1 =     524.414900
kp  =     868.710400
ex  =       1.193594
mu  =      93.103400

Standard variances (in %) for each parameter:
kVB =       5.452899
kG1 =   3.963895E-01
kp  =       1.198544
ex  =   3.832978E-01
mu  =   1.493413E-01
```

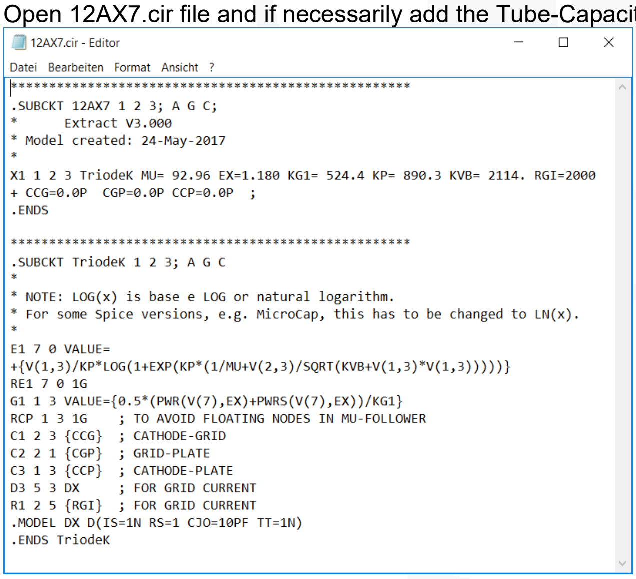
- The “Gnuplot-Windows” open and shows the difference between the µTracer data and the tube-model data



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

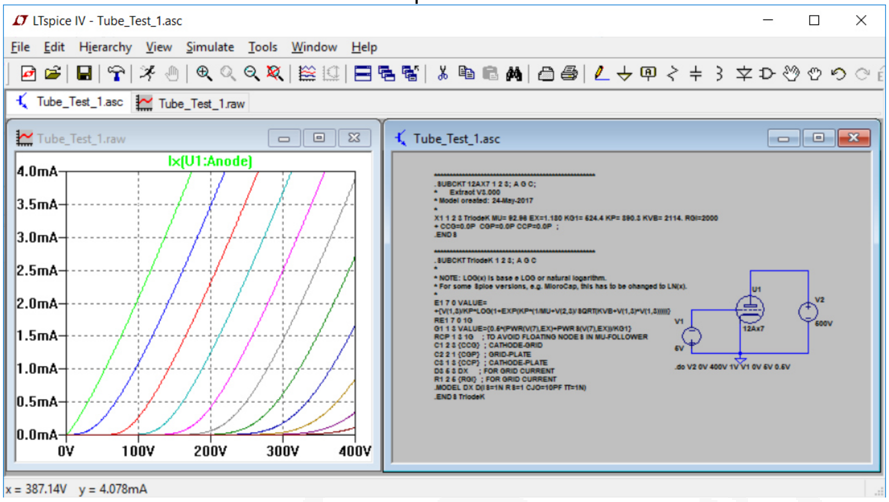


- Open 12AX7.cir file and if necessarily add the Tube-Capacities




```
*****************************************************
.SUBCKT 12AX7 1 2 3; A G C;
*      Extract V3.000
* Model created: 24-May-2017
*
X1 1 2 3 TriodeK MU= 92.96 EX=1.180 KG1= 524.4 KP= 890.3 KVB= 2114. RGI=2000
+ CCG=0.0P  CGP=0.0P CCP=0.0P  ;
.ENDS

*****************************************************
.SUBCKT TriodeK 1 2 3; A G C
*
* NOTE: LOG(x) is base e LOG or natural logarithm.
* For some Spice versions, e.g. MicroCap, this has to be changed to LN(x).
*
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
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
```

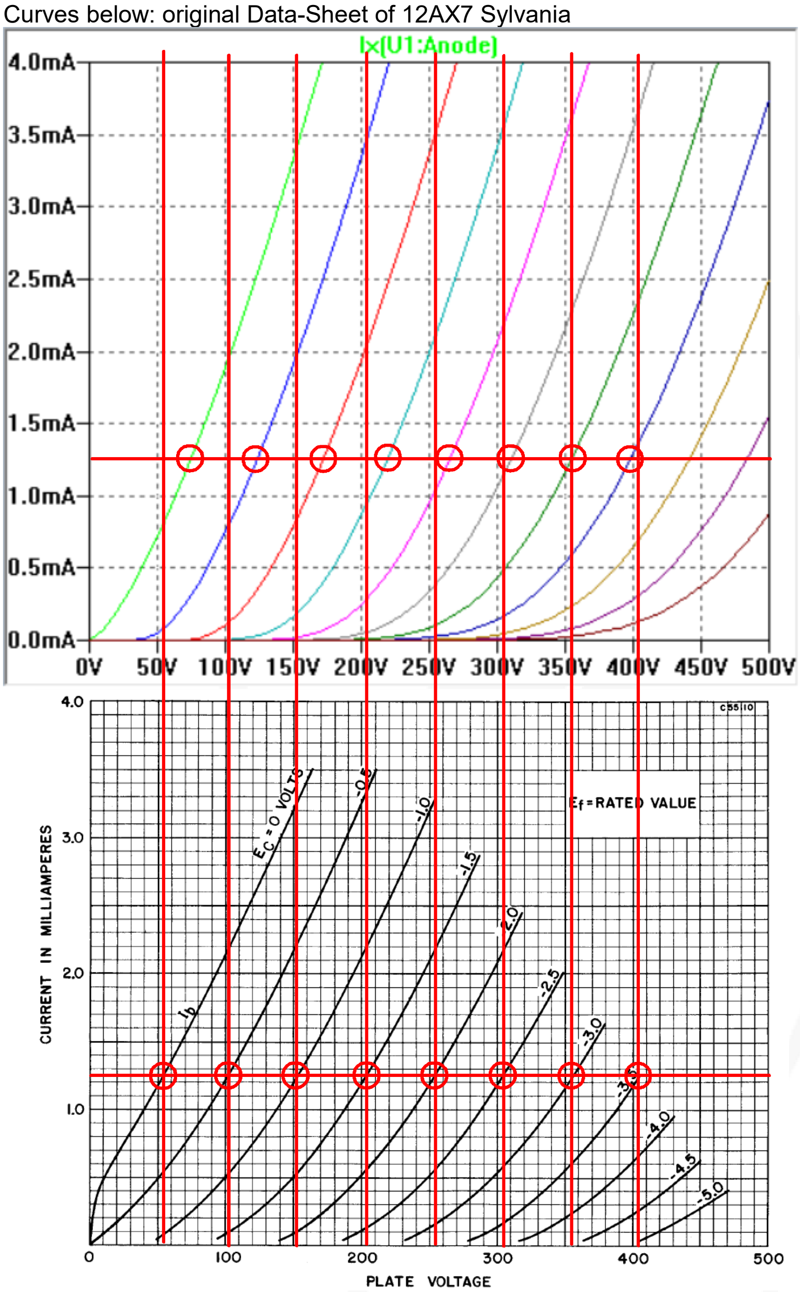
- 12AX7 ExtractModel checked with LTspice



Tube-Curve Contest

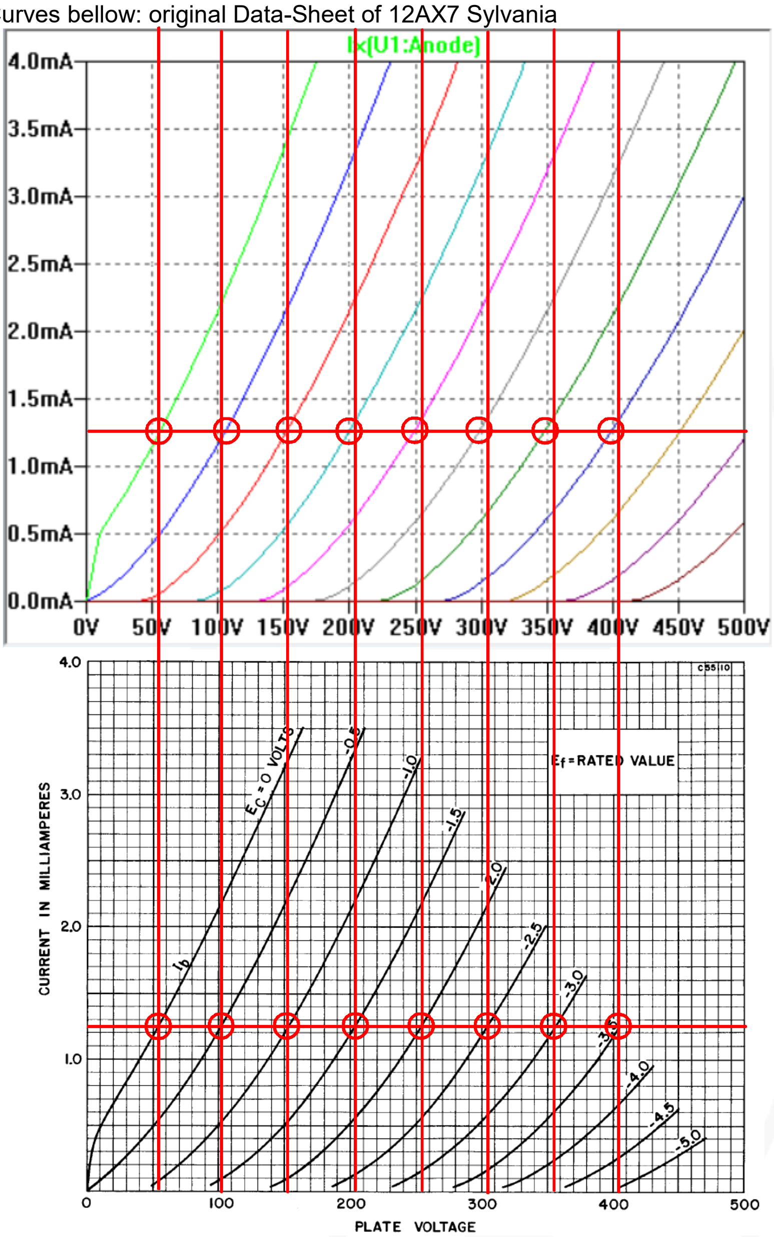
1. Norman L. Koren

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



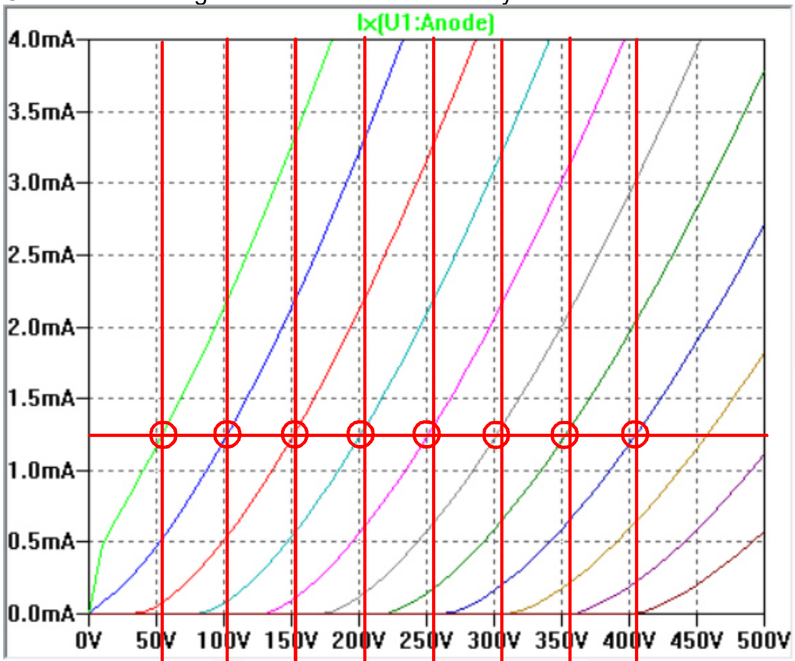
2. S. Bench/Ducan

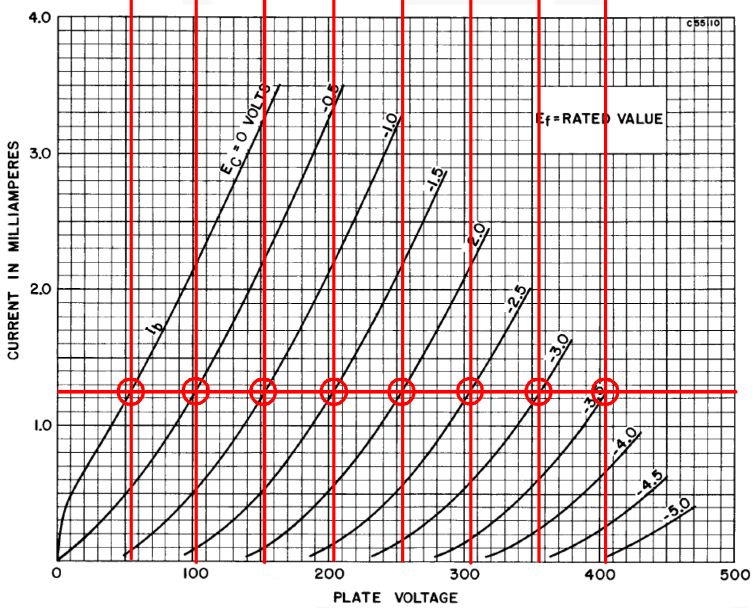
- Curves up: 12AX7 Library „triode_nh.inc“ (Curves created with LT-Spice)
- Curves bellow: original Data-Sheet of 12AX7 Sylvania



3. Rydel

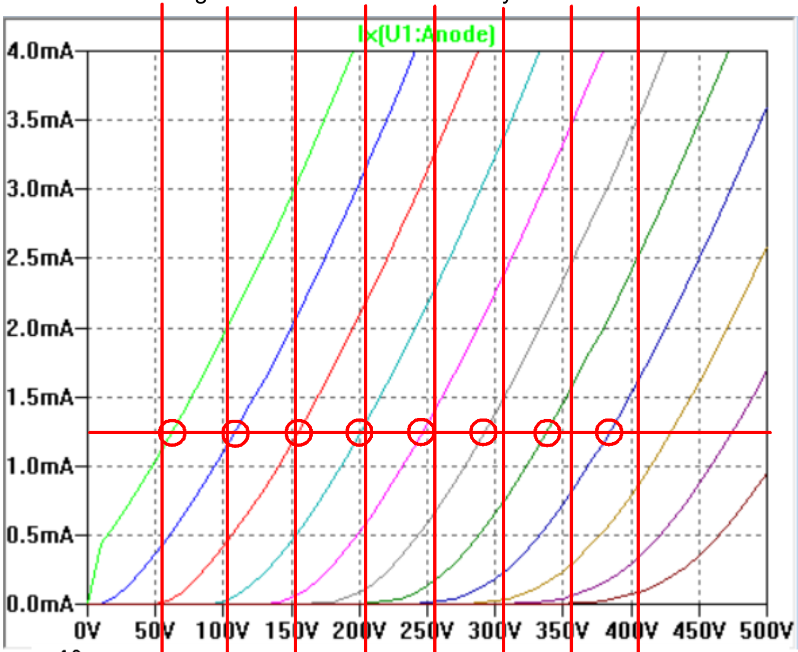
- Curves up: 12AX7 Tube-Model created with "Curve Captor Program" (Andrei Frolov) according to the "Rydel5 Tube-Model" and original Data-Sheet of 12AX7 Sylvania (Curves created with LT-Spice)
- Curves bellow: original Data-Sheet of 12AX7 Sylvania

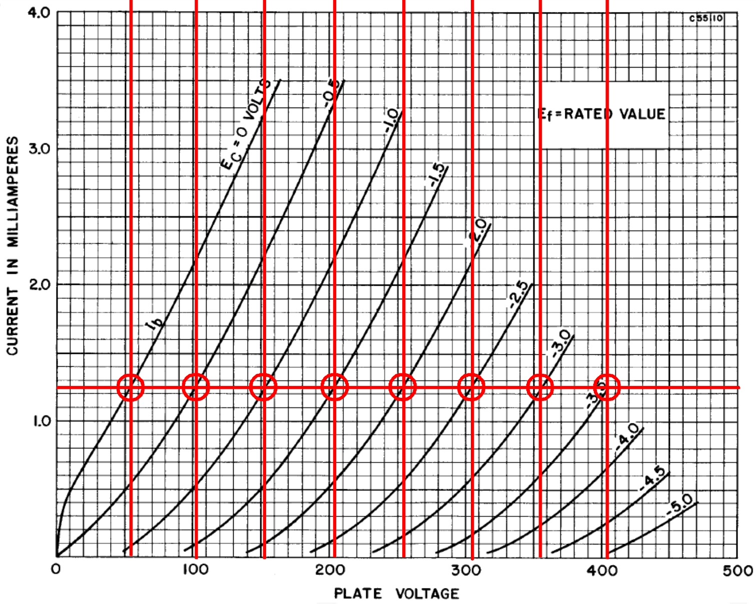




4. Dimitry Nizhegorodov

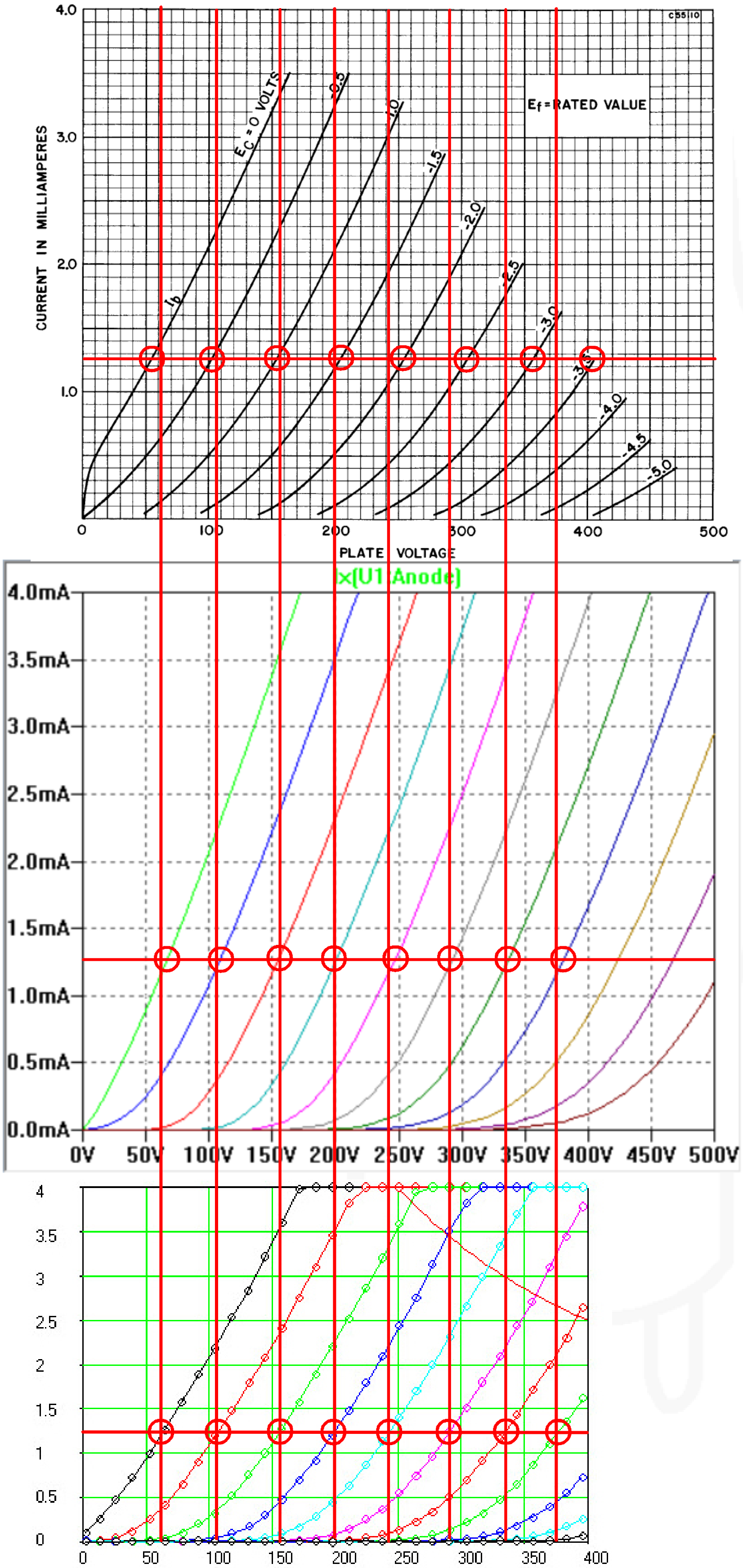
- Curves up: 12AX7 Library „Dmitry_composites.lib“ (Curves created with LT-Spice)
- Curves bellow: original Data-Sheet of 12AX7 Sylvania





5. Ronald Dekker, Derk Refman

- 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.

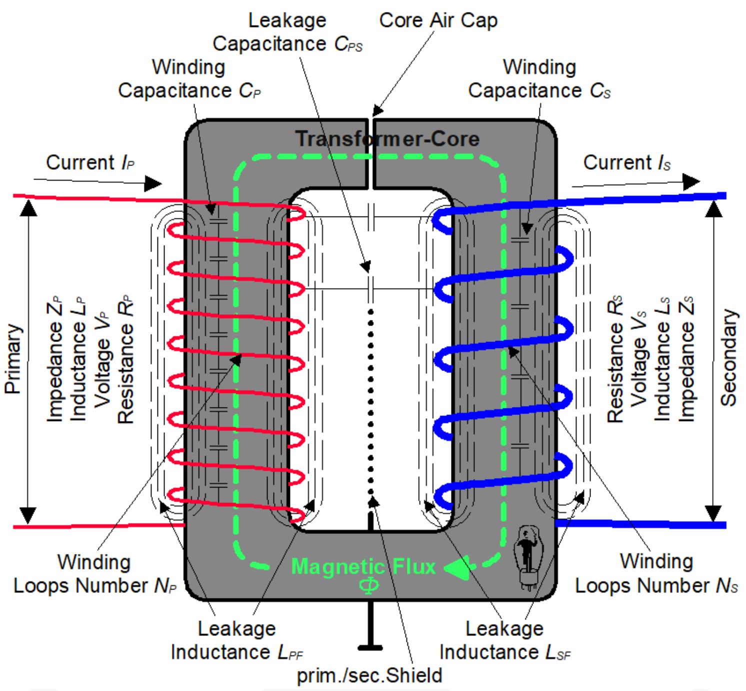
- By itself, the output transformer has no impedance. Simply it reflects the secondary impedance back to the primary. This means the impedance between the loudspeaker and the tube(s) and vice versa. So the most important parameter is the Turns Ratio *N*.
- The primary inductance L_p value has a direct 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 inter-winding 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

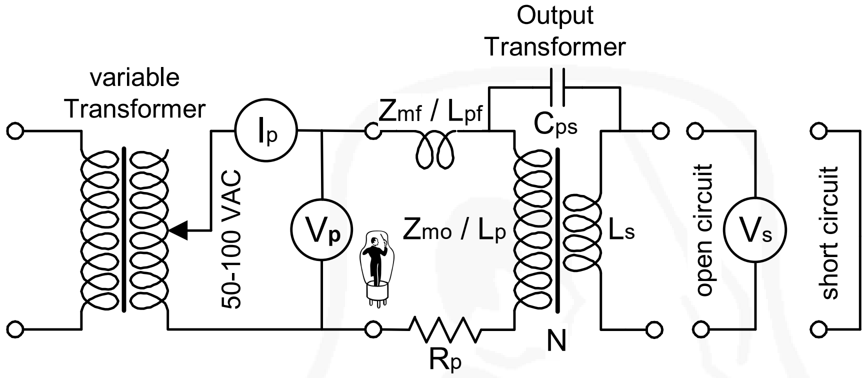
2. Elements of a simple Audio Transformer



- **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.
- **I_s** = 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.
- **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.
  - $\boldsymbol{V_p}$ and $\boldsymbol{I_p}$ = measure V_p and I_p and V_s (measure I_s it's not necessary).
  - Short the secondary side of the transformer.
  - $\boldsymbol{V_p}$ and $\boldsymbol{I_p}$ = measure again V_p and I_p.
  - $\boldsymbol{R_p}$ = measure primary winding DC-resistor if possible with a 4-wire bridge-meter.
  - $\boldsymbol{C_{ps}}$ = 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.

    $Z_{mo} = \frac{V_p}{I_p}$
  - **Z_{mf}** = Leakage Impedance (Ω) primary winding measured with short circuit secondary winding(s).

    $Z_{mf} = \frac{V_p}{I_p}$ (this value is usually 100th to 1000th of Z_{mo})
  - **N** = Turns Ratio = the quotient of the primary and secondary windings

    $N = \frac{V_p}{V_s}$
  - **K** = Coupling Coefficient from primary to secondary windings

    $K = \sqrt{1 - \frac{Z_{mf}}{Z_{mo}}}$
  - **Z_{pf}** = primary leakage Impedance (Ω)

    $Z_{pf} = (1 - K) \cdot Z_{mo}$
  - **Z_p** = primary Impedance (Ω)

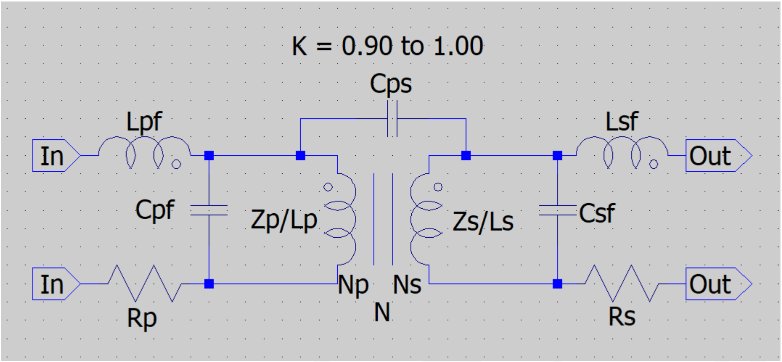
    $Z_p = K \cdot Z_{mo}$
  - **Z_s** = secondary Impedance (Ω)

    $Z_s = K \cdot Z_{mo} \cdot N^2$
  - **Lp** = primary Inductance (H)

    $L_p = \frac{Z_p}{2\pi \cdot f}$ (use measured and calculated Z_p and 50Hz (60Hz)
  - **Ls** = secondary Inductance (H)

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

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
  $N = \frac{V_p}{V_s} = \frac{N_p}{N_s} = \frac{I_s}{I_p}$ or $N = \sqrt{\left(\frac{L_P}{L_S}\right)}$ or $N = \sqrt{\left(\frac{Z_P}{Z_S}\right)}$
  $L_s = \frac{L_p}{N^2}$
  $L_p = N^2 \cdot L_s$

- **Z_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.

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

- **L_P** 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_{pf}, 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.

  $K = \sqrt{1 - \left(\frac{L_{mf}}{L_{mo}}\right)}$ or $\sqrt{1 - \left(\frac{L_{pf}}{L_p}\right)}$

  $L_{pf} = (1 \cdot K^2) \cdot L_p$

  K = Coupling Coefficient
  L_{mo} = measured primary inductance (open secondary winding)
  L_{mf} = measured leakage inductance (short circuit secondary winding)
  L_p = 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.

  $Z_s = \frac{V_s}{I_s}$

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

- **L_s** 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

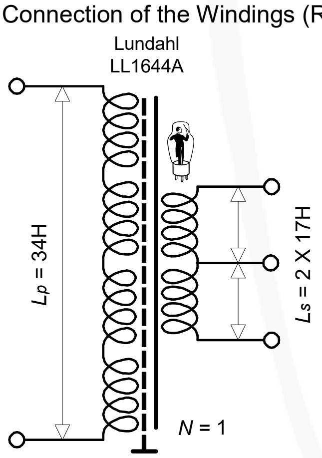
- Please read first the chapter "Calculation of Transformer Parameters"

- Example: Lundahl LL1544
- Specification out of the data sheet

Termination Alternative	Turns ratio	Copper Resistance Prim/sec	Idle impedance @40 Hz, 0dBU	Suggested Use	THD < 0.5%@50 Hz primary level / real source impedance
R4B / R4U : L4B / L4U	1:1	520Ω / 520Ω	80kΩ / 80kΩ	10 kΩ / 10 kΩ	20 dBU / 600Ω

- Calculation:
  $N = \frac{N_P}{N_S} = \frac{1}{1} = 1$
  $L_S = \frac{L_P}{N^2} = \frac{34}{1^2}$ = 34H 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"

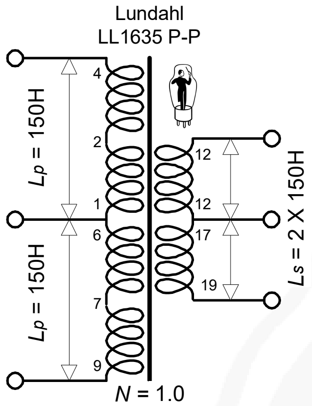
- Example: Lundahl LL1635 P-P
- Specification out of the data sheet

	LL1635 P-P
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
Frequency response, primaries in series	5 Hz - 60 kHz
(Source 4 kΩ for PP and 5mA, 2 kΩ 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:
  $N = \frac{N_p}{N_s} = \frac{2}{2} = 1$
  $L_s = \frac{L_P}{N^2} = \frac{300}{1^2} = 300H$
  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 P-P



3. Line Output Transformer

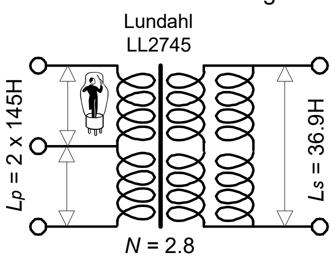
- Please read first the chapter ““Calculation of Transformer Parameters”

- Example: Lundahl LL2745/PP
  Specification out of the data sheet

Type	**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 (*) Secondaries open	15kΩ
Max sec. voltage @ 30 Hz	190V r.m.s.

- Calculation:
  $N = \frac{N_p}{N_s} = \frac{2 \cdot 2.8}{2} = 2.8$
  $L_s = \frac{L_P}{N^2} = \frac{290}{2.8^2} = 36.9$ 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”

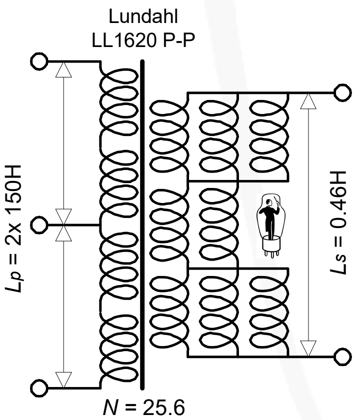
- Example: Lundahl LL1620 P-P
  Specification out of the data sheet

	LL1620	
Turns ratio:	4 x 19.2 : 8 x 1	
Static resistance of primary (all in series)	308 Ω (4 • 77 Ω)	
Static resistance of each secondary (average)	0.4 Ω	
Primary leakage inductance (all in series)	11 mH	
Max. recommended primary DC current (heat dissip. 7W)	150 mA	
Max. primary signal voltage r.m.s. at 30 Hz (all in series)	Push-Pull 860V	Single End 380V

	Secondary connection for 4/8/16 Ω (See next page)			**DC current for 0.9 Tesla (rec. operating point) Primary Inductance**
	-/B/C	B/C/D	C/D/E	
	Primary Load Impedance (transformer copper resistance included)			
LL1620	11.5 kΩ	6.0 kΩ	3.3 kΩ	Push-Pull 300 H

- Calculation:
  $N = \frac{N_p}{N_s} = \frac{4 \cdot 19.2}{3} = 25.6$
  $L_s = \frac{L_P}{N^2} = \frac{300}{25.6^2} = 0{,}46$ Henry
  $K = \sqrt{1 - \left(\frac{L_{pf}}{L_p}\right)} = \sqrt{1 - \left(\frac{0.011}{300}\right)} = 0.99998$

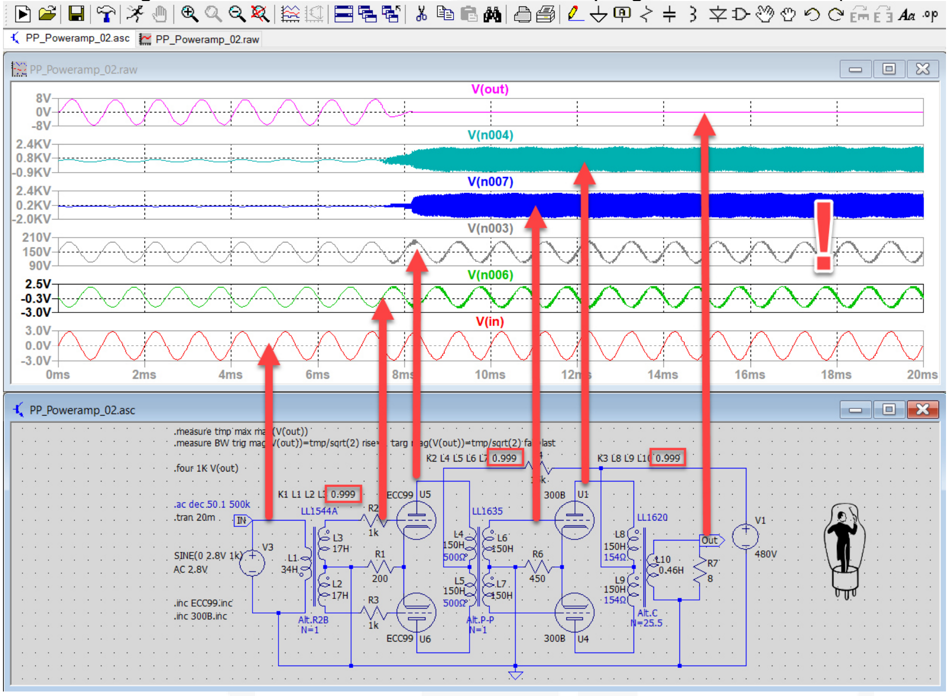
- Connection of the Windings Alt C (6.0kΩ / 8Ω)



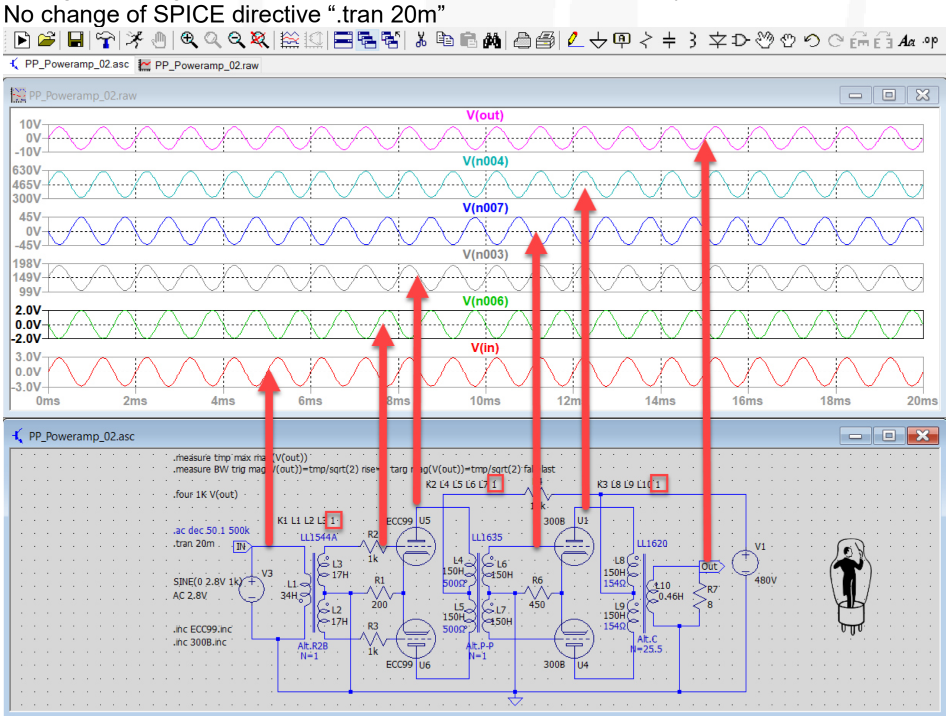
Oh dear LTspice !

1. Oscillation !

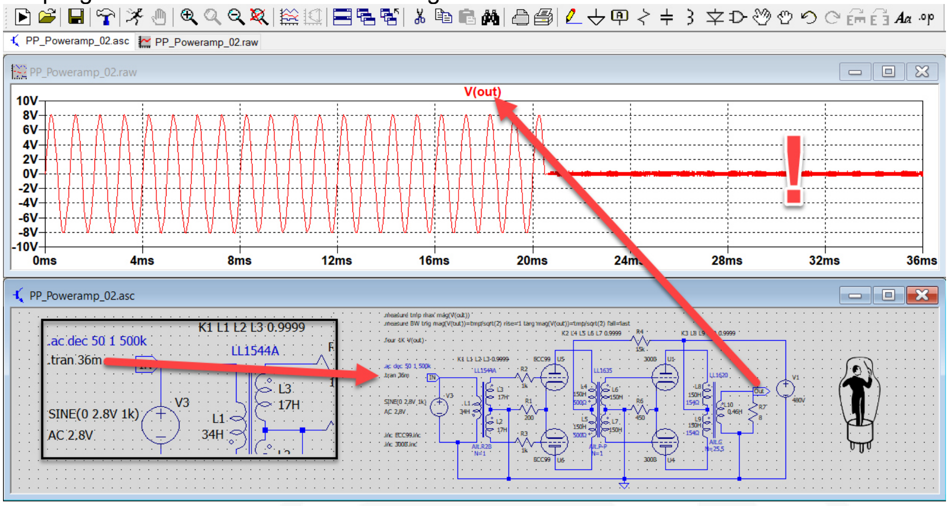
- LTspice don't like this PP Amplifier with three transformers in the row
- It is oscillating with SPICE directive ".tran 20m" and coupling factor "K = 0.999 (also with 0.9998)" !



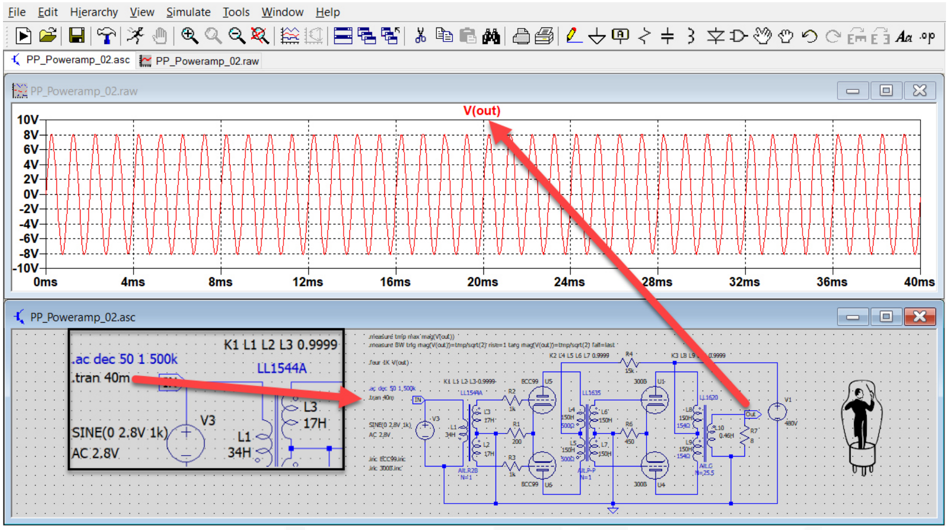
Change coupling factor "K to 1.0" and there is not oscillation anymore !
No change of SPICE directive ".tran 20m"



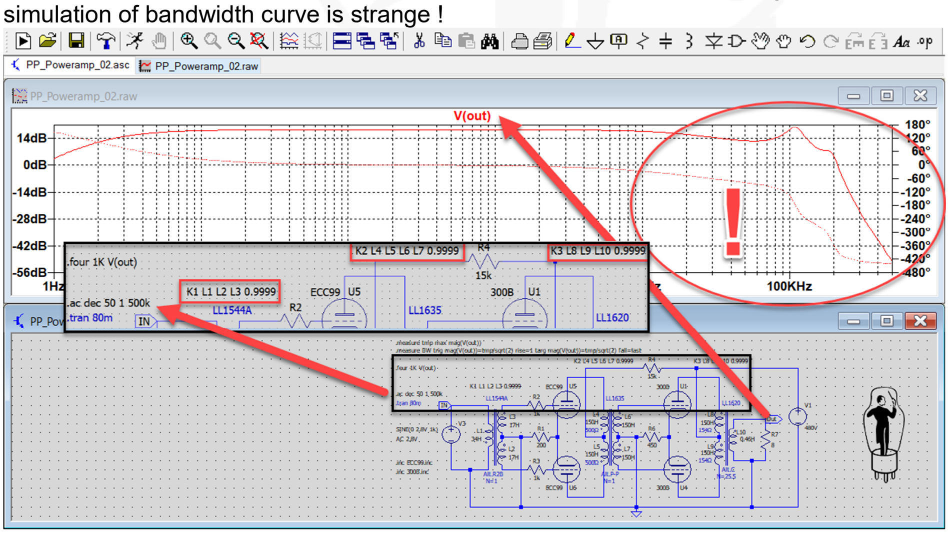
- Same PP Amplifier with SPICE directive from “.tran 5m until .tran 36m” and coupling factor “K = 0.999” it is oscillating !



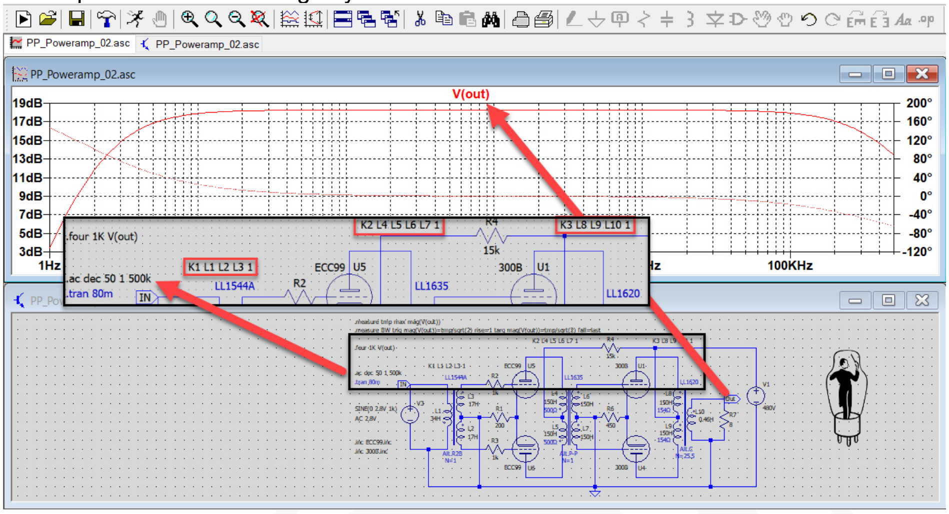
Change SPICE directive to “.tran 40m” and higher and same coupling factor “K = 0.999”, the amplifier is not oscillating anymore !



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

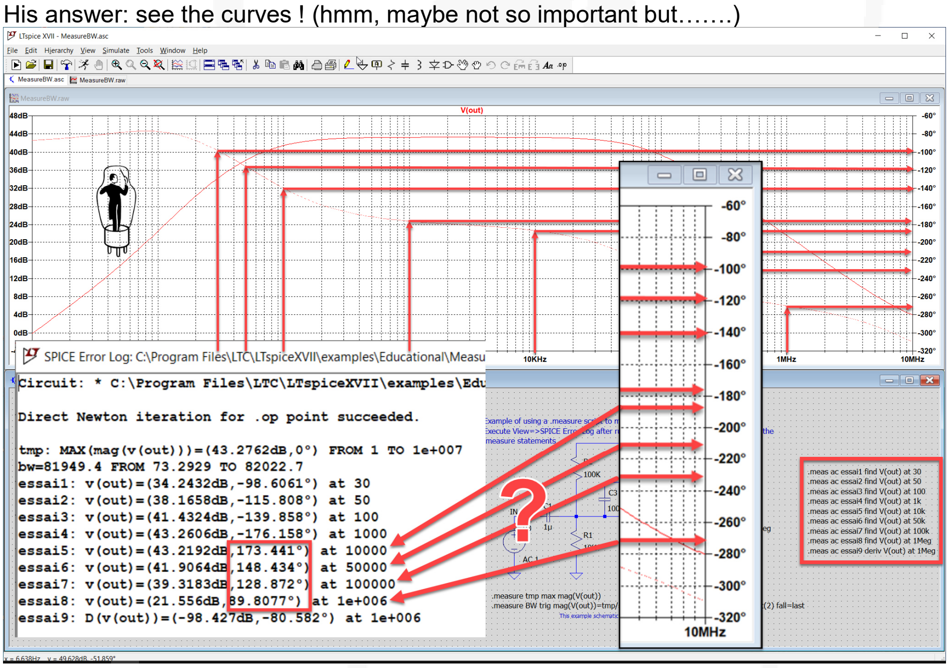


Change coupling factor “K to 1.0”, keep SPICE directive the same and the amplifier is not oscillating anymore !



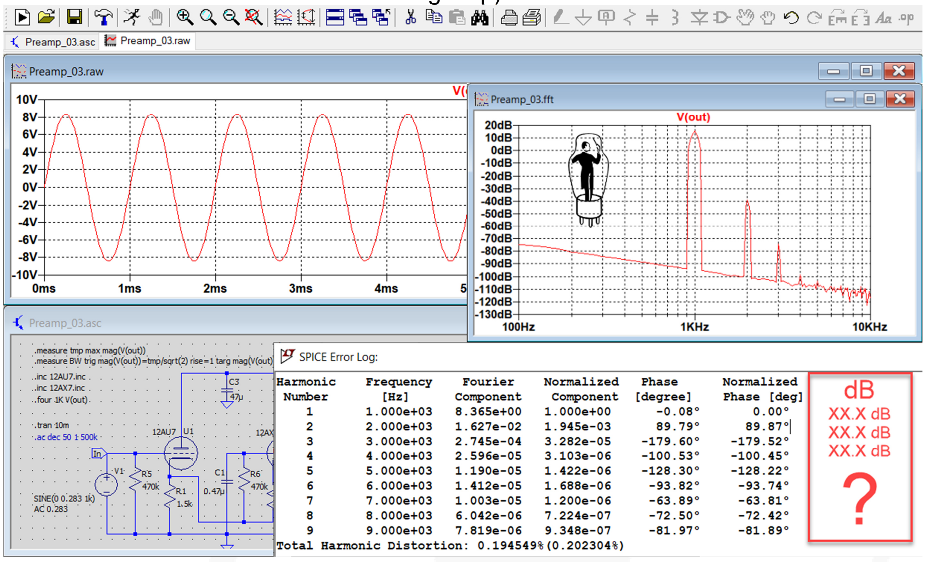
2. Incorrect Information in the SPICE Error Log Table when calculating minus 3dB Bandwidth !

- 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, but for the "normal user" it would be a big help)



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

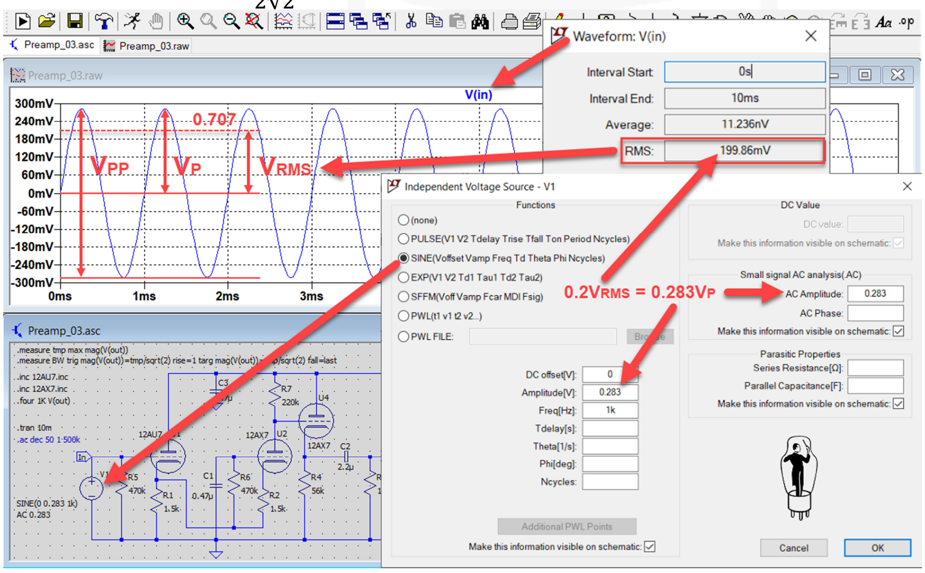
- Lots of internet-examples are not correct interpreted, because of the misunderstanding of the voltage terms: V_{PP} or V_P or V_{RMS} and the calculation out of it.
- It's easy to oversee the voltage terms at the LTspice input screen of the signal source is always V_P, but at the output signal is V_{PP} !
- 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:

"$V_{RMS} \rightarrow V_P$" $= V_{RMS} * \sqrt{2} = 0.2V_{RMS} * 1.414 = 0.2828V_P$

"$V_{RMS} \rightarrow V_{PP}$" $= V_{RMS} * 2\sqrt{2} = 0.2V_{RMS} * 2.828 = 0.566V_{PP}$

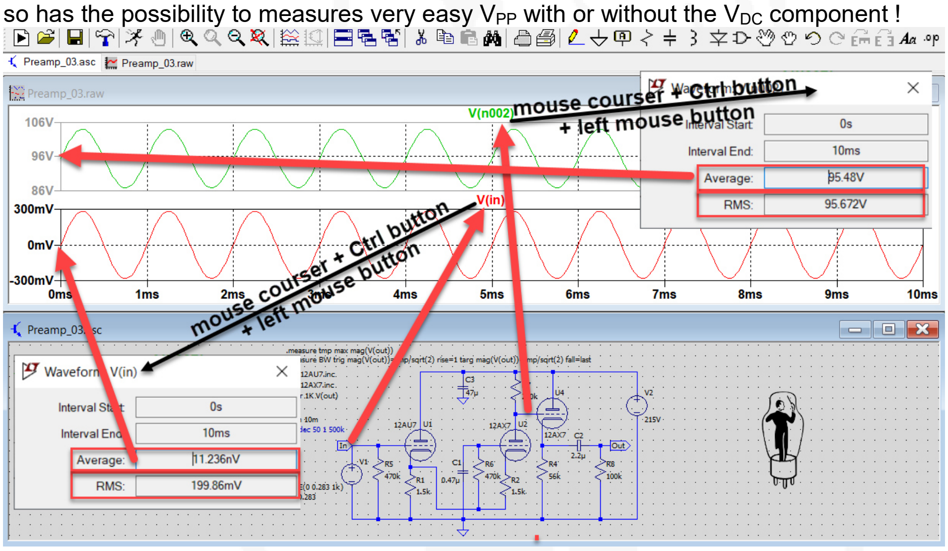
"$V_P \rightarrow V_{RMS}$" $= V_P * \frac{1}{\sqrt{2}} = 0.2828V_P * 0.707 = 0.199V_{RMS}$

"$V_{PP} \rightarrow V_{RMS}$" $= V_{PP} * \frac{1}{2\sqrt{2}} = 0.566V_{PP} * 0.354 = 0.200V_{RMS}$



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} 199.86mV_{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 !



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