Vacuum diode Models & PSpice Simulations

by Stefano Perugini

This article appeared originally in *Glass Audio*, Vol.10, Nr. 4, 1998
and is further edited from the web version.

The renewed interest in the thermionic technology is pushing designers to use the modern tools of circuit simulation via SPICE-oriented simulation for the optimization of components.

This article focuses on the creation of a very simple but accurate vacuum diode model with its associated power-supply components. The use of such devices presents some problems; it is necessary to check by calculations, several parameters such as the *repetitive peak current*, the *hot-switching current*, the *maximum inverse voltage* and so on, otherwise it is possible to lead the device to a premature death. If you begin the process with circuit simulation, on the condition that you have accurate models, you can realize the optimization phase in the virtual world and then build the real object.

At the end of this article I will show the goodness of the mathematical model used by comparing it with a real situation, that is, comparing the results gotten by measurements on a simple power supply physically realized, with the results gotten by the same simulated power supply.

START

In the "sacred texts" the equation derived for describing the operation of the diode is Child-Langmuir's Law:

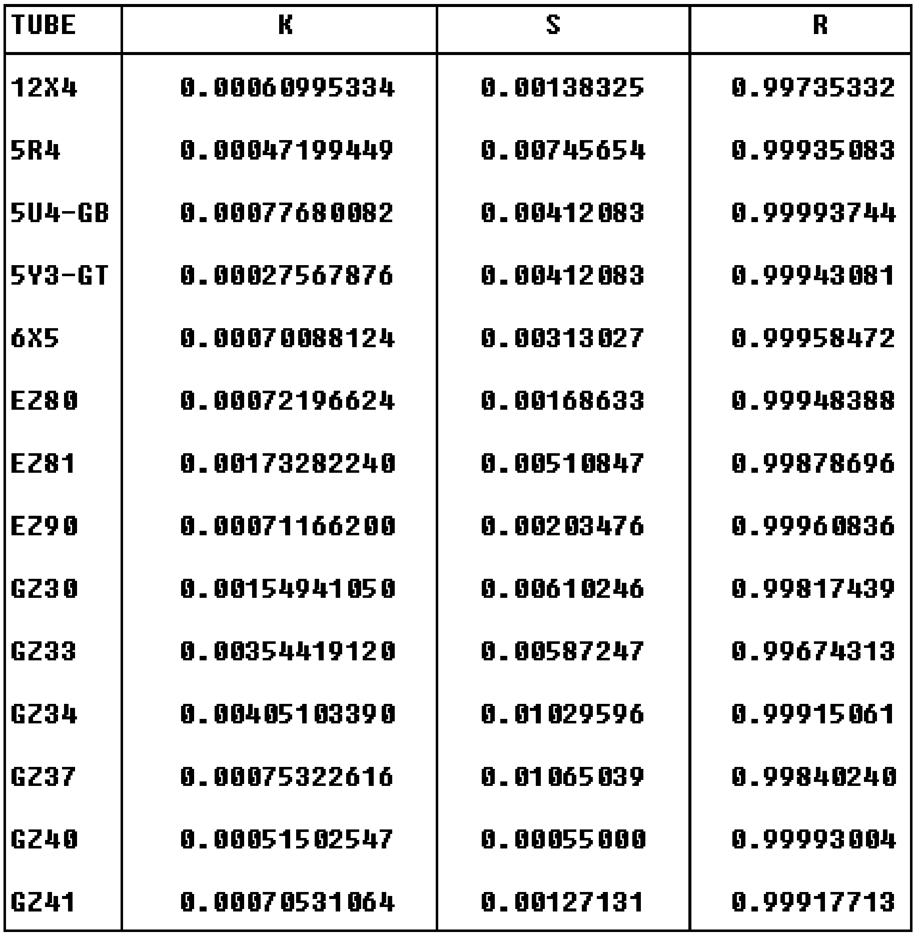
$$Ip = K*Vp^{1.5}$$

where:
Ip is the current that flows in the diode;
Vp is the anode to cathode voltage;
K is a constant, the *perveance*.

The *perveance* is the digital imprint or the genetic code of the diode because in it constitutes the only parameter able to differentiate between the multiplicities of diodes.

The Child-Langmuir's Law is a phenomenological equation that is derived on physical considerations grounds.

In this table, **S** and **R** represent the *Standard Error* and the *Correlation Coefficient* respectively.



TUBE	K	S	R
12X4	0.00060995334	0.00138325	0.99735332
5R4	0.00047199449	0.00745654	0.99935083
5U4-GB	0.00077680082	0.00412083	0.99993744
5Y3-GT	0.00027567876	0.00412083	0.99943081
6X5	0.00070088124	0.00313027	0.99958472
EZ80	0.00072196624	0.00168633	0.99948388
EZ81	0.00173282240	0.00510847	0.99878696
EZ90	0.00071166200	0.00203476	0.99960836
GZ30	0.00154941050	0.00610246	0.99817439
GZ33	0.00354419120	0.00587247	0.99674313
GZ34	0.00405103390	0.01029596	0.99915061
GZ37	0.00075322616	0.01065039	0.99840240
GZ40	0.00051502547	0.00055000	0.99993004
GZ41	0.00070531064	0.00127131	0.99917713

Tab. 1; Results for the $Ip=k*(Vp)^{1.5}$ model

A perfect fitting is gotten when **R** is equal to 1. The values of the Correlation Coefficient **R** in Table 1 must not deceive you. **R** is a parameter of global evaluation and values lower than 0.998 cannot be thought satisfactory as the graphs of the Fig. 1, 2, 3 related to the diodes **12X4**, **GZ30**, **GZ33** show. The model introduces, locally, marked deviations. In these graphs the "small black balls" represent the average data extracted from Data-Sheets.

If we use this equation to apply the *Least Square Method (LSM)* with *linear regression* to the set of experimental data extracted by the Data-Sheets it is difficult to get a good fitting, as you can see by examining Table 1.

In my opinion this model well represents only tubes like the **5U4** and **GZ40**: you can think of them as perfect diodes (although not ideal!). In all the other cases the results can be improved so as to get simulations nearest to reality. The apparent redundancy in Table 1 and those following showing either **12X4** or **EZ90/6X4** will be clarified later.

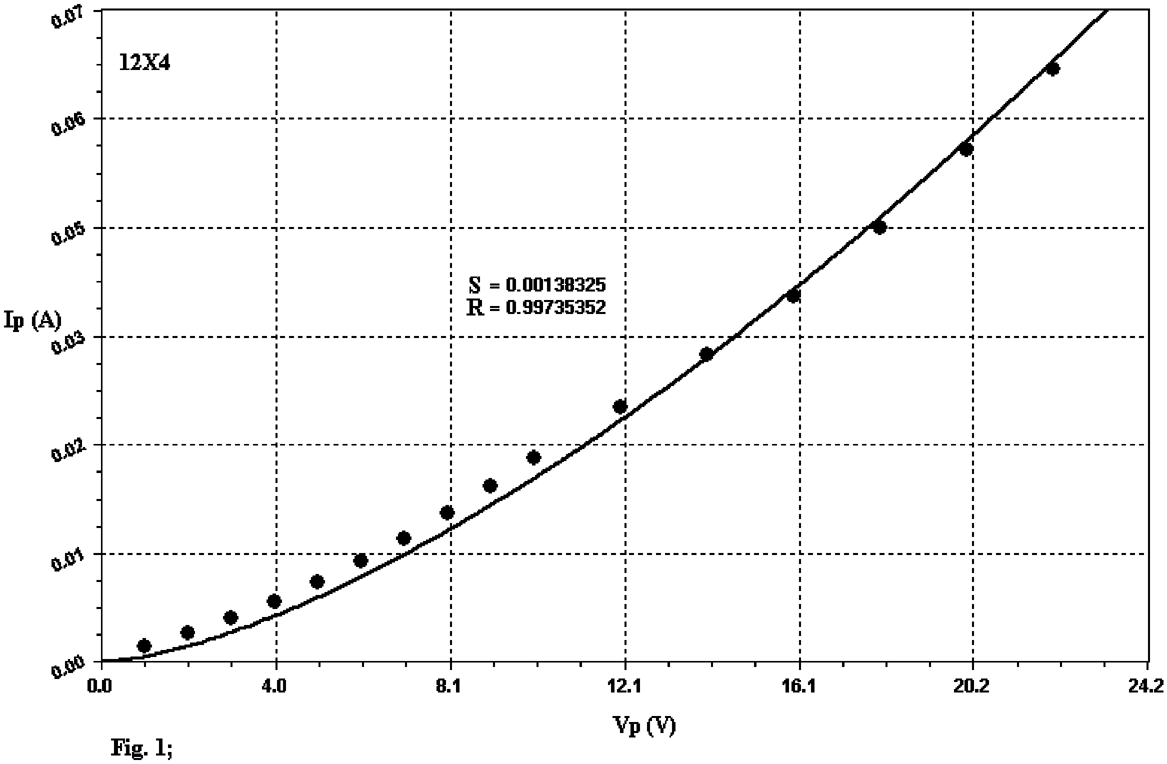


Fig. 1;

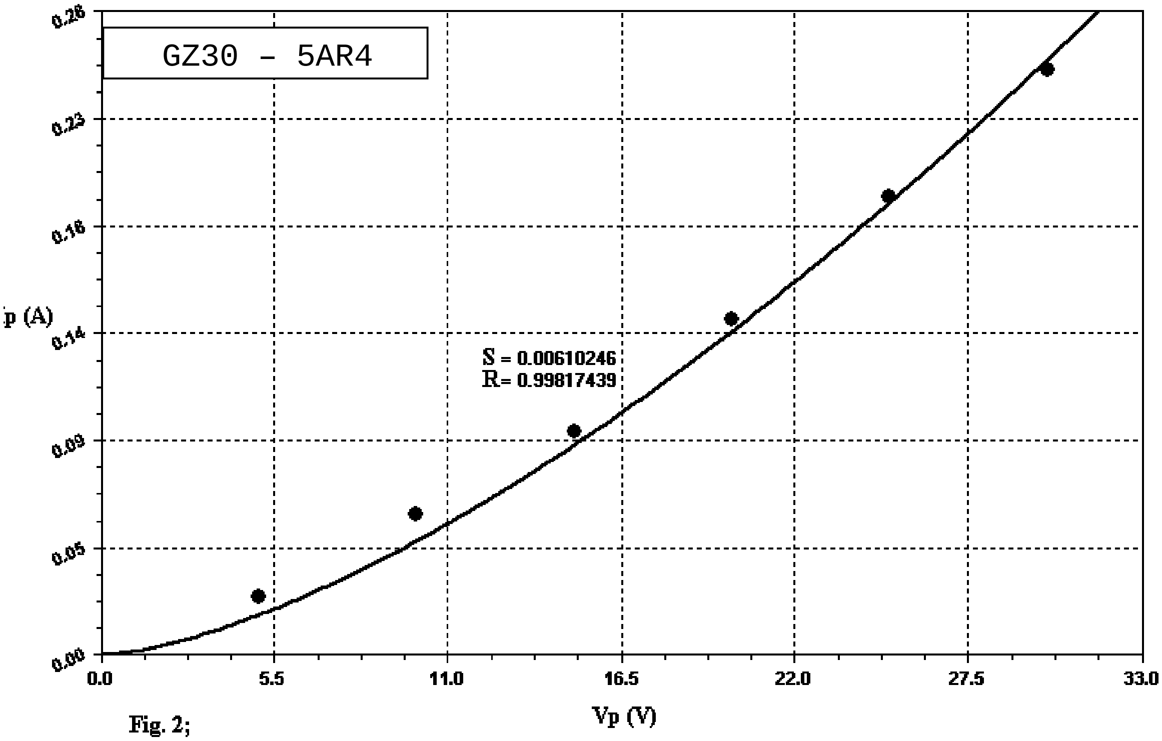

GZ30 – 5AR4
S = 0.00610246
R= 0.99817439
Ip (A)
0.28
0.23
0.18
0.14
0.09
0.05
0.00
0.0
5.5
11.0
16.5
22.0
27.5
33.0
Vp (V)

Fig. 2;

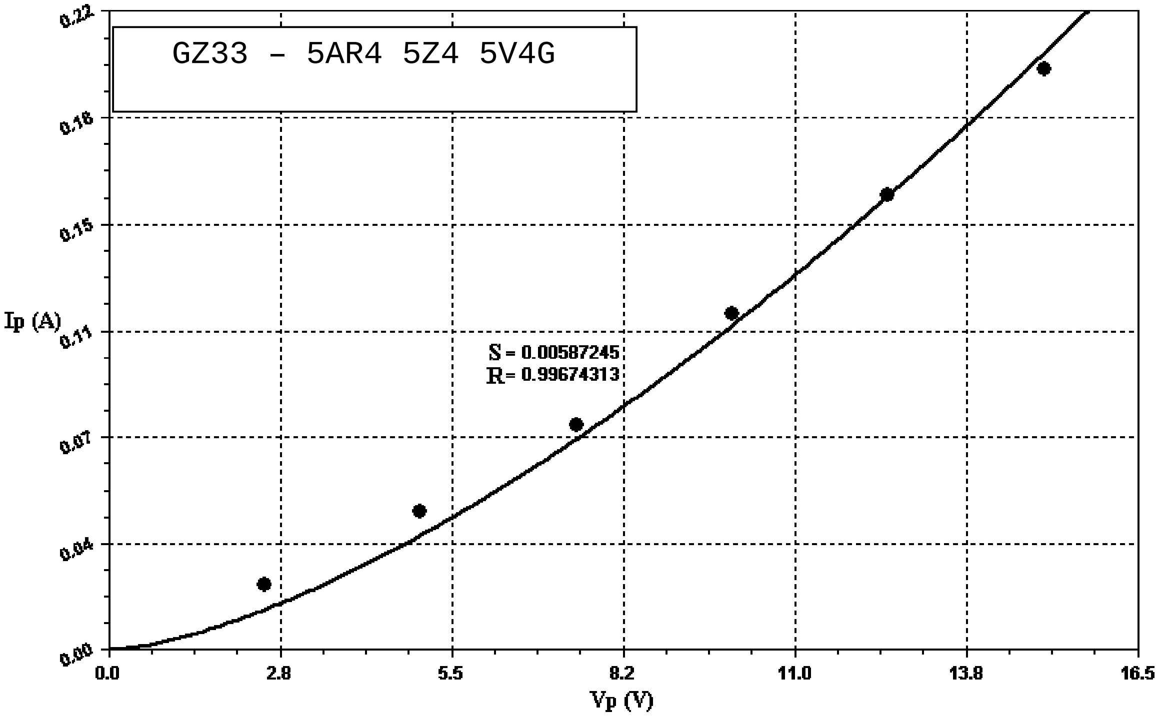

GZ33 – 5AR4 5Z4 5V4G
S = 0.00587245
R= 0.99674313
Ip (A)
0.22
0.18
0.15
0.11
0.07
0.04
0.00
0.0
2.8
5.5
8.2
11.0
13.8
16.5
Vp (V)

Fig. 3;

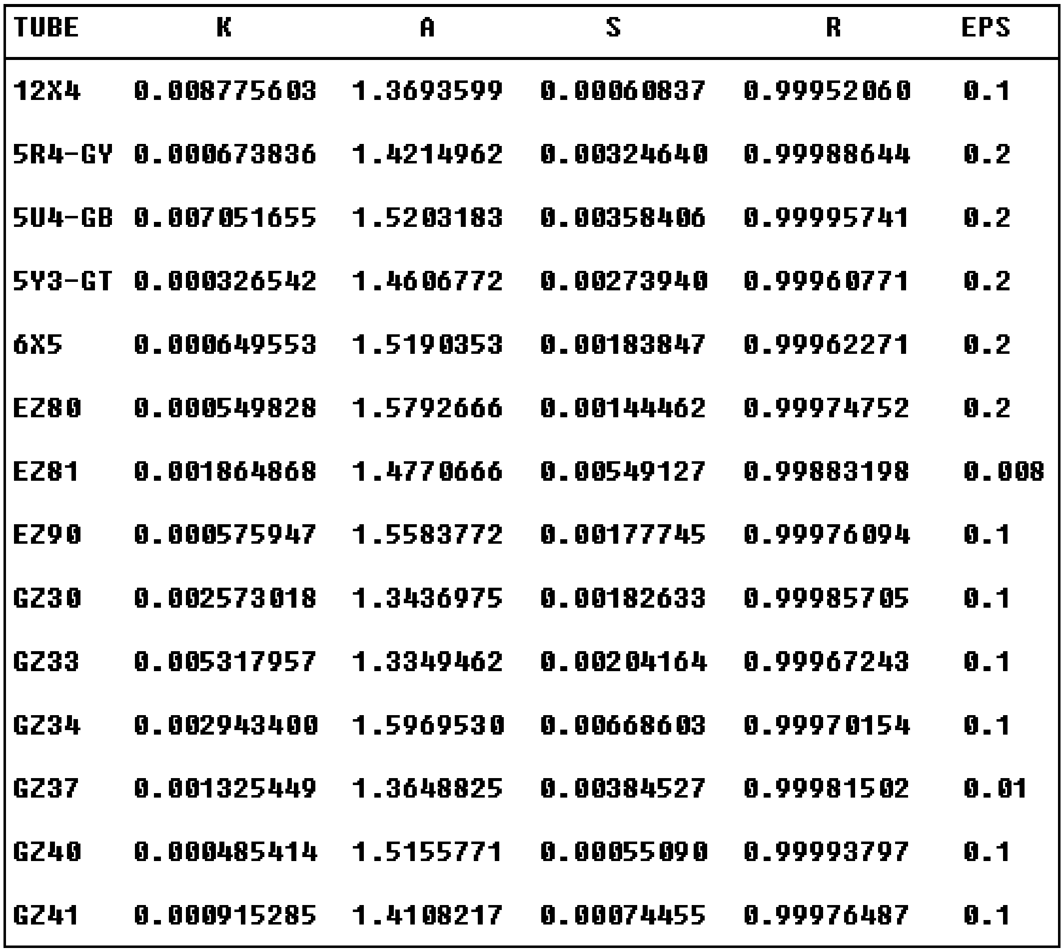
Improvement

You can already get a large improvement by varying the exponent of the equation. This leads to the following expression:

$$(1)\quad Ip= K*Vp^{A}$$

However, it is best to use this equation, modified in the following way:

$$(2)\quad Ip= K*(Vp+EPS)^{A}$$



TUBE	K	A	S	R	EPS
12X4	0.008775603	1.3693599	0.00060837	0.99952060	0.1
5R4-GY	0.000673836	1.4214962	0.00324640	0.99988644	0.2
5U4-GB	0.007051655	1.5203183	0.00358406	0.99995741	0.2
5Y3-GT	0.000326542	1.4606772	0.00273940	0.99960771	0.2
6X5	0.000649553	1.5190353	0.00183847	0.99962271	0.2
EZ80	0.000549828	1.5792666	0.00144462	0.99974752	0.2
EZ81	0.001864868	1.4770666	0.00549127	0.99883198	0.008
EZ90	0.000575947	1.5583772	0.00177745	0.99976094	0.1
GZ30	0.002573018	1.3436975	0.00182633	0.99985705	0.1
GZ33	0.005317957	1.3349462	0.00204164	0.99967243	0.1
GZ34	0.002943400	1.5969530	0.00668603	0.99970154	0.1
GZ37	0.001325449	1.3648825	0.00384527	0.99981502	0.01
GZ40	0.000485414	1.5155771	0.00055090	0.99993797	0.1
GZ41	0.000915285	1.4108217	0.00074455	0.99976487	0.1

Tab. 2; Results for the $Ip= K*(Vp+EPS)^{A}$

Then apply *LSM* to the set of experimental data. The results of this procedure have quoted in Table 2. ***EPS*** represents a parameter that can be manually fixed (typical values are 0.1, 0.2) to guarantee a better convergence of the algorithm. If you compare the Correlation Coefficient of Table 1 and 2, you can note a real amelioration, underlined subsequently by the new graphic representations of the tubes **12X4**, **GZ30**, **GZ33** shown in Fig. 4, 5, and 6. Furthermore the examination of Table 2 reveals that the real diode is dominated by the 3/2-power law in only a few cases.

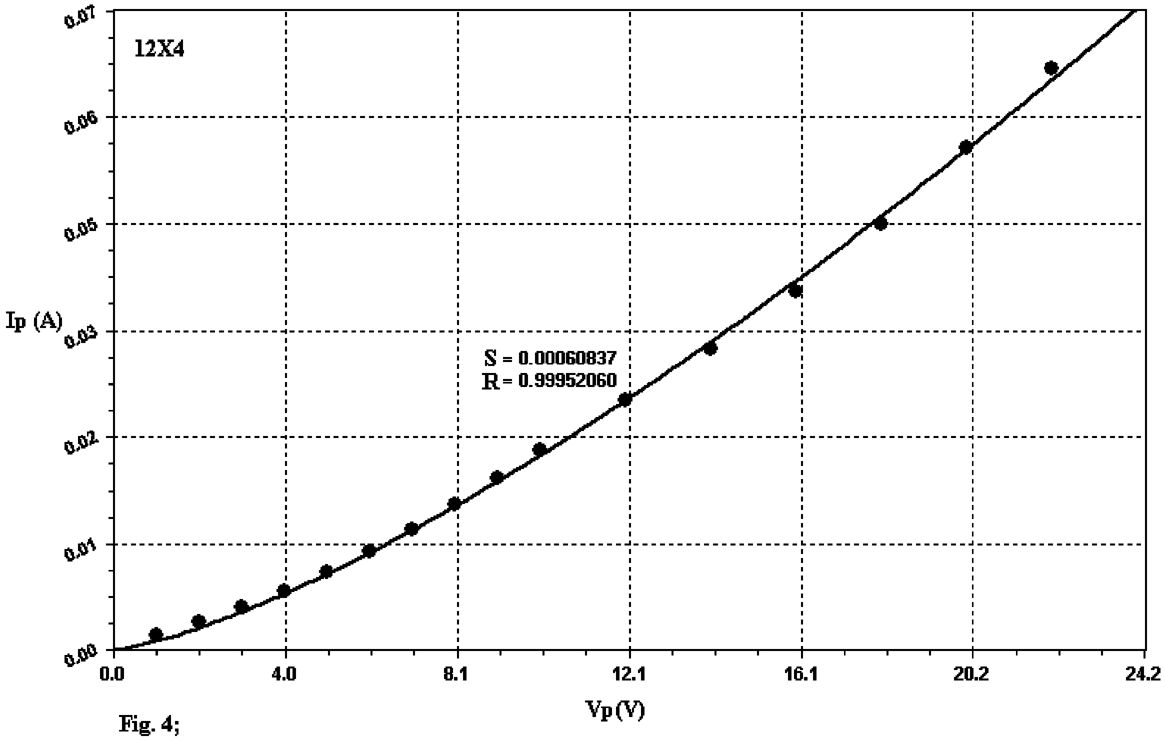


Fig. 4;

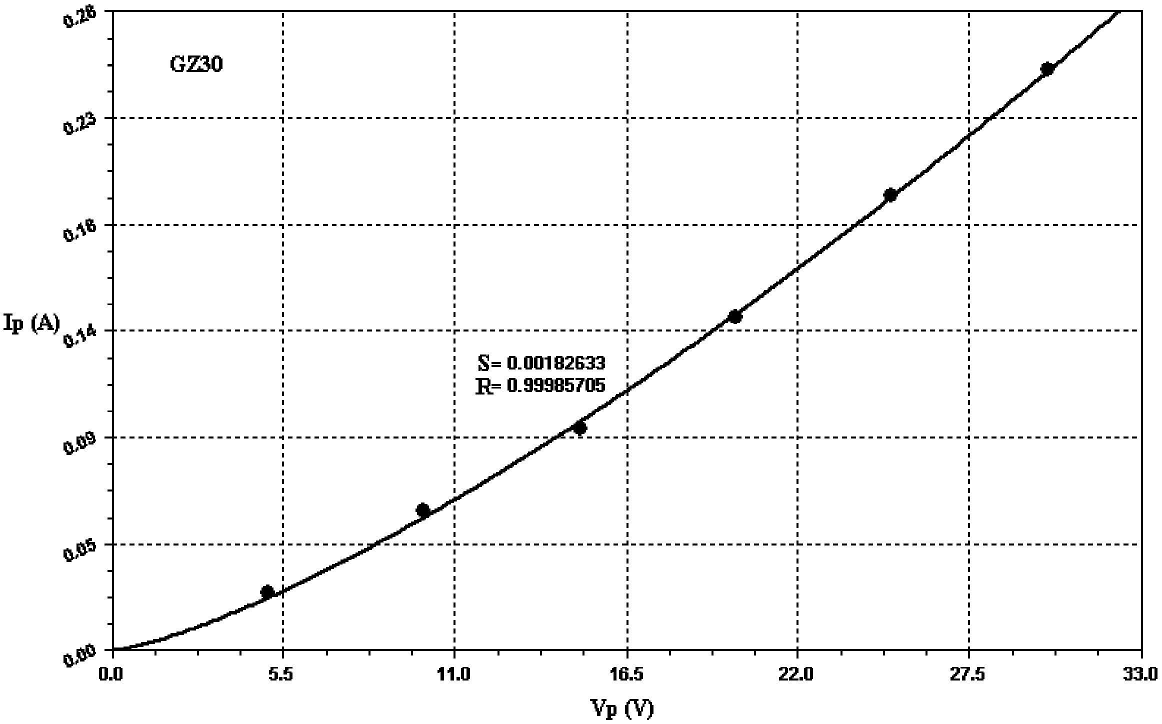


Fig. 5;

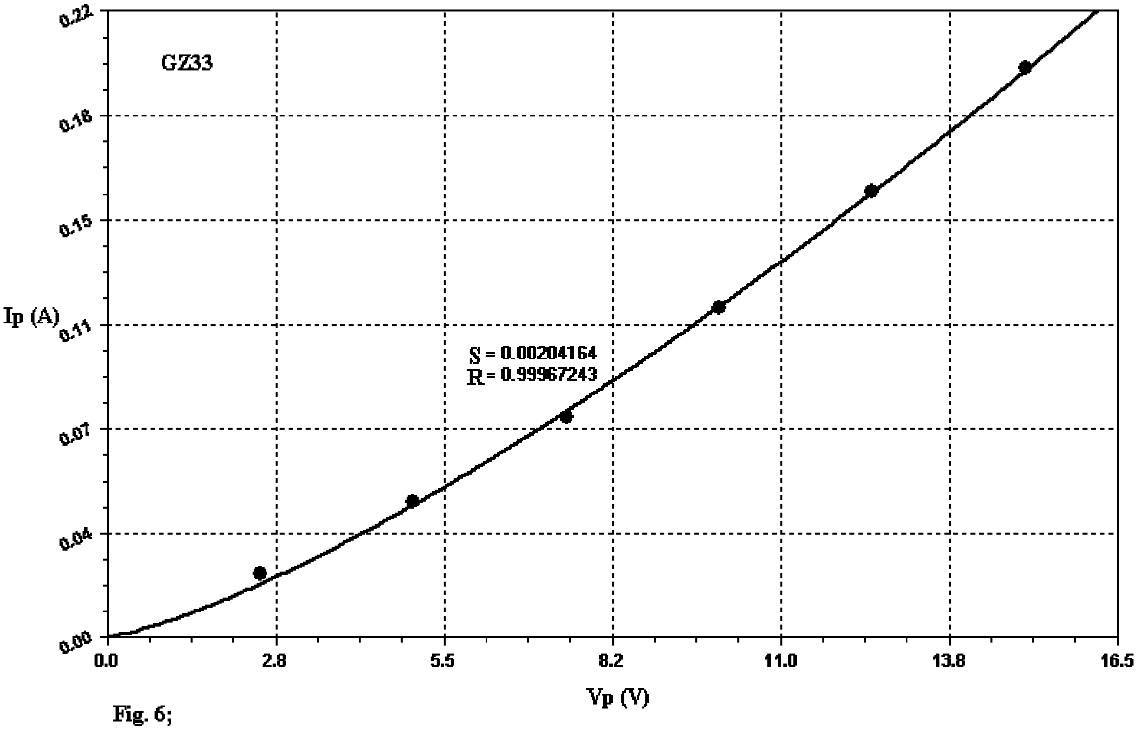
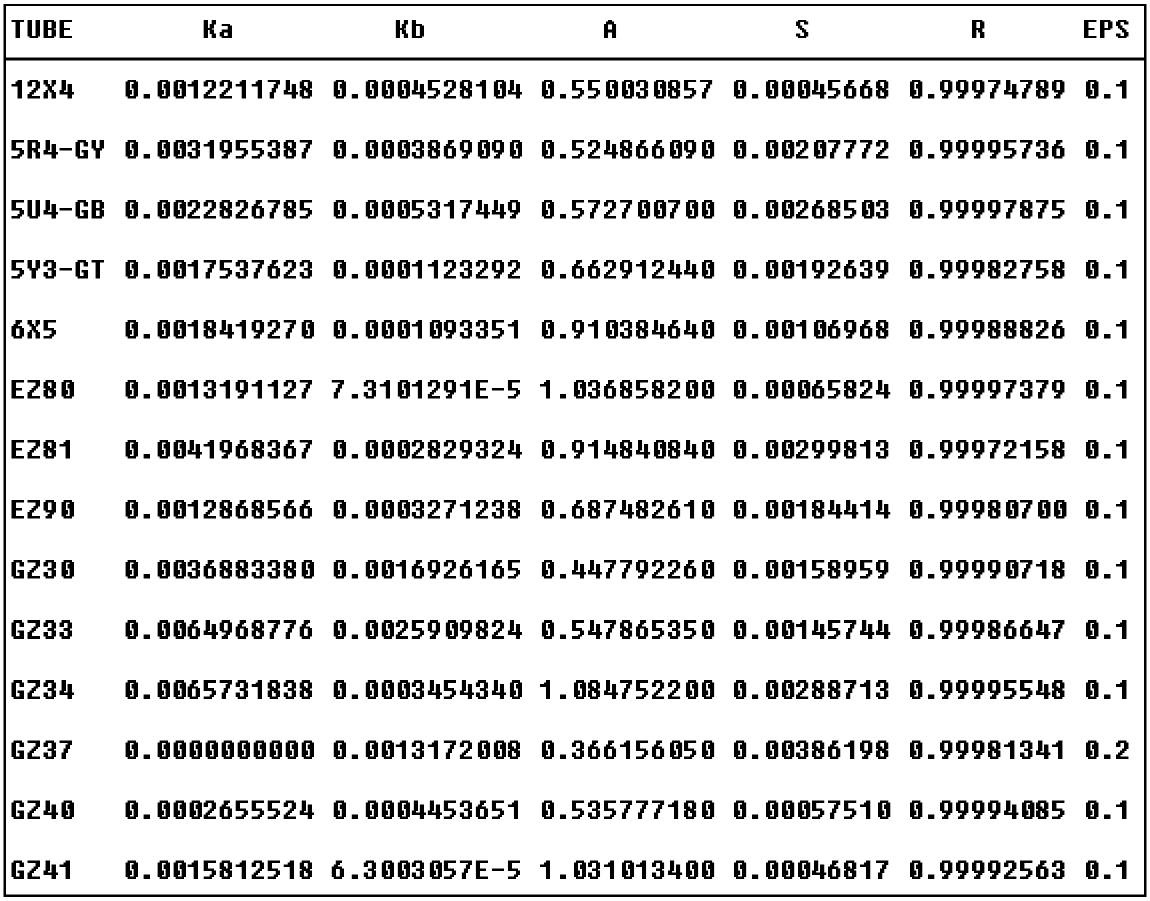


Fig. 6;

Finally you can get an even better fitting curve if you use the following equation:

$$(3)\quad Ip = (Ka+Kb*Vp)*(Vp+EPS)^{A}$$

Equation *(3)* differs from *(2)* because a linear variation to the perveance has been attributed with respect to *Vp*. The application of LSM with linear regression leads to the data in Table 3. You can get further short improvement margins using more sophisticated mathematical models although this injures simplicity and brings larger problems of convergence into the simulations.



TUBE	Ka	Kb	A	S	R	EPS
12X4	0.0012211748	0.0004528104	0.550030857	0.00045668	0.99974789	0.1
5R4-GY	0.0031955387	0.0003869090	0.524866090	0.00207772	0.99995736	0.1
5U4-GB	0.0022826785	0.0005317449	0.572700700	0.00268503	0.99997875	0.1
5Y3-GT	0.0017537623	0.0001123292	0.662912440	0.00192639	0.99982758	0.1
6X5	0.0018419270	0.0001093351	0.910384640	0.00106968	0.99988826	0.1
EZ80	0.0013191127	7.3101291E-5	1.036858200	0.00065824	0.99997379	0.1
EZ81	0.0041968367	0.0002829324	0.914840840	0.00299813	0.99972158	0.1
EZ90	0.0012868566	0.0003271238	0.687482610	0.00184414	0.99980700	0.1
GZ30	0.0036883380	0.0016926165	0.447792260	0.00158959	0.99990718	0.1
GZ33	0.0064968776	0.0025909824	0.547865350	0.00145744	0.99986647	0.1
GZ34	0.0065731838	0.0003454340	1.084752200	0.00288713	0.99995548	0.1
GZ37	0.0000000000	0.0013172008	0.366156050	0.00386198	0.99981341	0.2
GZ40	0.0002655524	0.0004453651	0.535777180	0.00057510	0.99994085	0.1
GZ41	0.0015812518	6.3003057E-5	1.031013400	0.00046817	0.99992563	0.1

Tab. 3; Results for $Ip= (Ka+Kb*Vp)*(Vp+EPS)^{A}$ model

SPICE

Now you can use the data in **Table 3** to build the SPICE model of the diode. After the excellent articles by Raynolds, Marshall and Koren, I don't think you will find any difficulty understanding the code in Table 4. The only required operation is to use the information in Table 3 and complete the code in Table 4 based upon the values of the selected diode.

```
.SUBCKT tubename P K
+ PARAMS Ka=      Kb=
+ A=       Eps=
E1       1   0    VALUE = {Ka + Kb * V(P,K)}
RE1    1   0     1G
E2       2    0    VALUE = {V(P,K) + EPS}
RE2    2    0     1G
G1       P   K    VALUE = {V(1)/2 * (PWR(V(2), A) + PWRS(V(2), A))}
RPK      P   K     1MEG
* CPK   P   K      .5n
.ENDS
```

(*) CPK can replace RPK in case of serious convergence problems.

Tab. 4

How use this model

With an accurate model of vacuum diode you can determine, when the device is used as a rectifier, a series of parameters for the *repetitive peak current*, the *hot-switching current*, the *output impedance* etc. Besides the SPICE model, you will also need further inputs to place the device in within safety areas. This information is graphically available in *Data-Sheets Rating Charts*. In my opinion the *Radiotron Designer's Handbook* fully deals with the subject as it faces the whole problem list merely in engineering terms and therefore with calculations and formulae finalized to the solution of an actual situation. In the *Radiotron's* presentation of formulated procedures it is possible to plan power supplies with both condenser input filters and choke input filters.

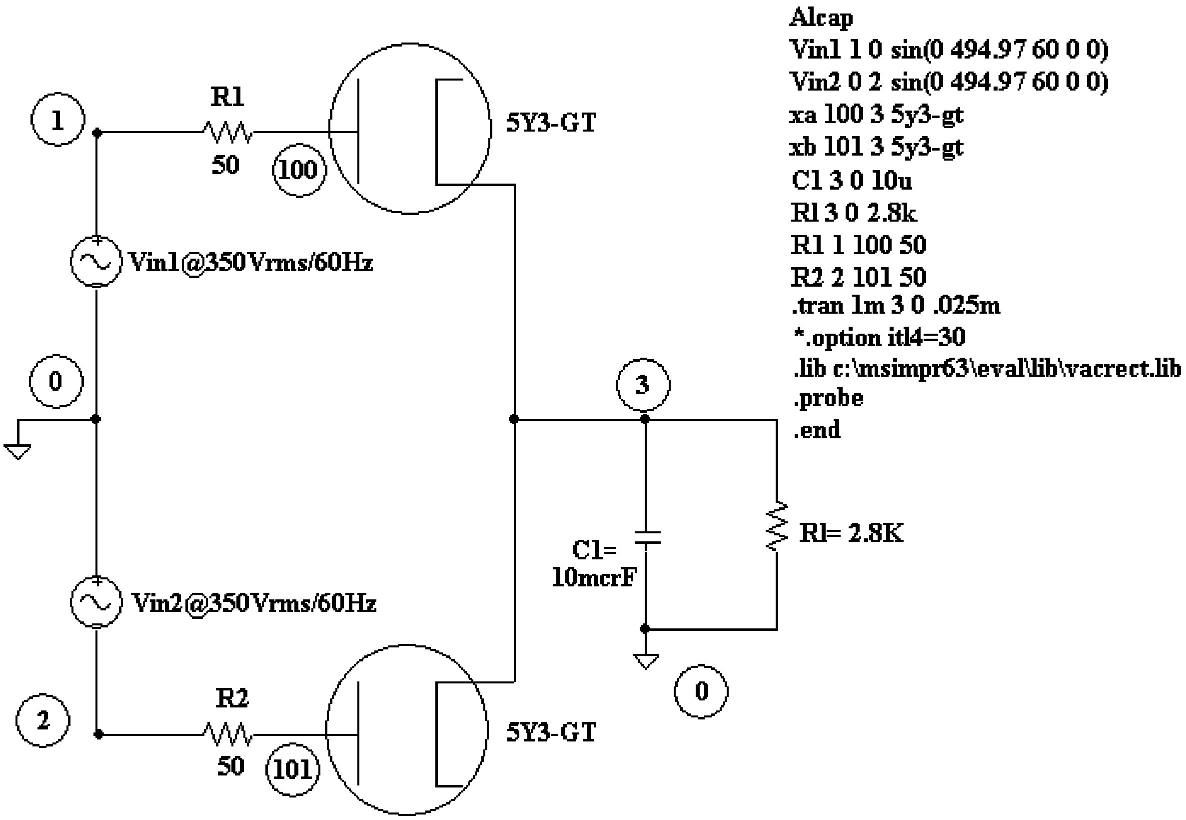
Unfortunately the accuracy, especially in the evaluation of the impulsive currents, is not high but this is due to the use of purposely-approximate formulae having rather simplistic calculations that at the time were made by the hand. The book further introduces two levels of approximation if you have complete data.

In Fig. 7 and 8 I have brought two hypotheses of power supply with condenser and choke input filter respectively together with their SPICE codes related to present examples beginning from the pages 1174 and 1183 of the *Radiotron Designer's Handbook*. The resistances *R1* and *R2* are the *total effective plate supply impedance per plate* and they represent the impedances brought to every secondary winding according to the formula:

$$\mathbf{Rs= Rsec + N^2 * Rpri};$$

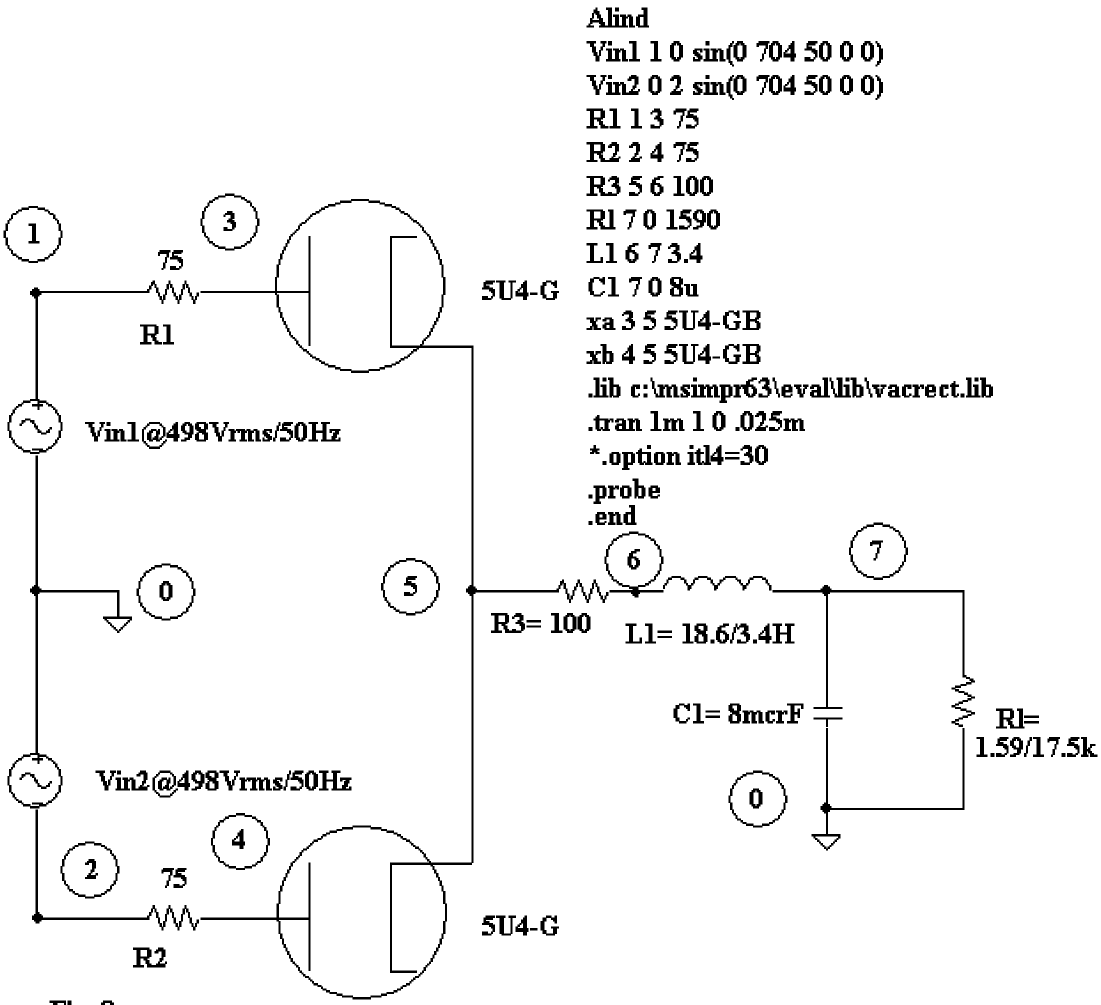
where:

N = Voltage ratio of transformer at no load:
(primary to half secondary in case of full-wave rectification)
Rpri = Resistance of primary winding in ohms;
Rsec = Resistance of secondary winding in ohms
(or half secondary in case of full-wave rectification;
Rs = total effective plate supply impedance per plate.



```
Alcap
Vin1 1 0 sin(0 494.97 60 0 0)
Vin2 0 2 sin(0 494.97 60 0 0)
xa 100 3 5y3-gt
xb 101 3 5y3-gt
C1 3 0 10u
Rl 3 0 2.8k
R1 1 100 50
R2 2 101 50
.tran 1m 3 0 .025m
*.option itl4=30
.lib c:\msimpr63\eval\lib\vacrect.lib
.probe
.end
```

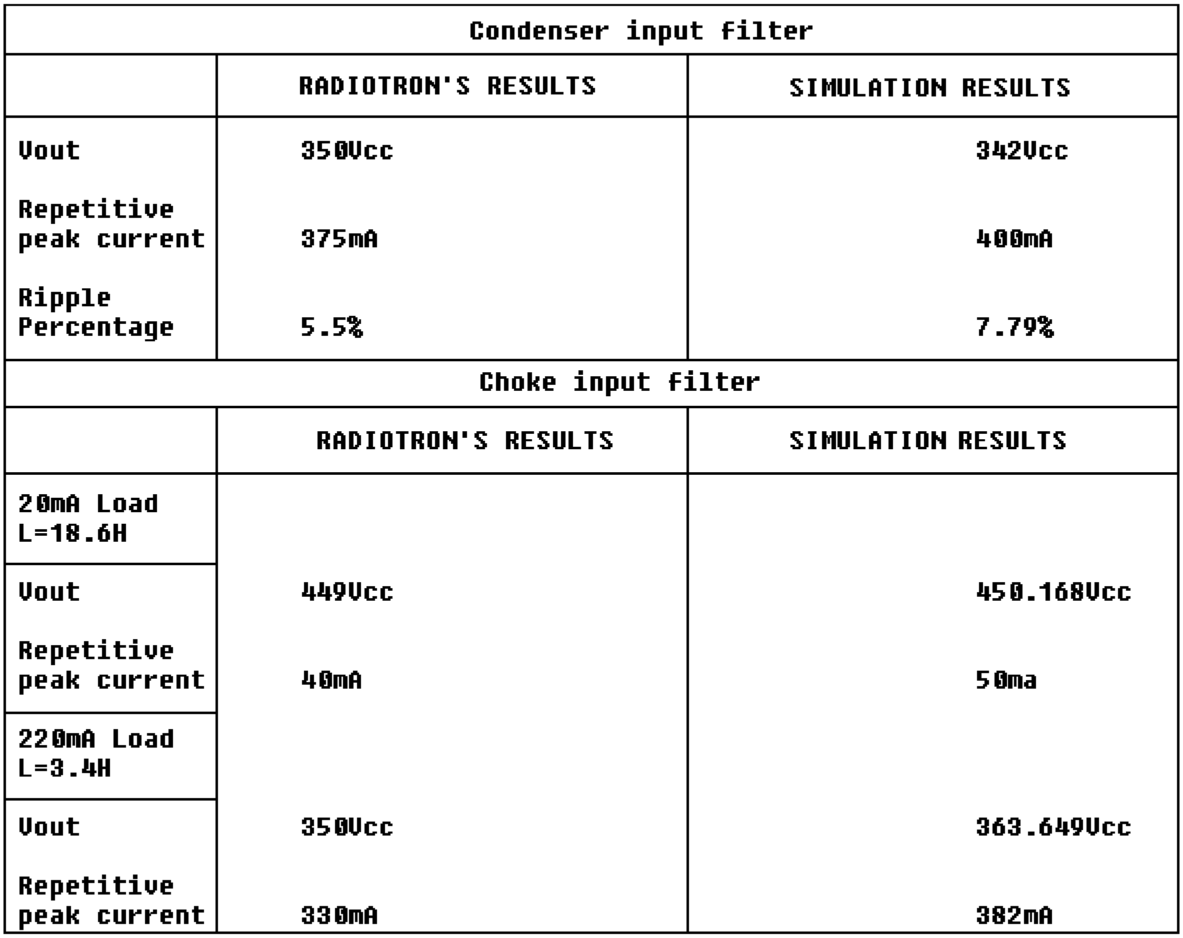
Fig. 7;



```
Alind
Vin1 1 0 sin(0 704 50 0 0)
Vin2 0 2 sin(0 704 50 0 0)
R1 1 3 75
R2 2 4 75
R3 5 6 100
Rl 7 0 1590
L1 6 7 3.4
C1 7 0 8u
xa 3 5 5U4-GB
xb 4 5 5U4-GB
.lib c:\msimpr63\eval\lib\vacrect.lib
.tran 1m 1 0 .025m
*.option itl4=30
.probe
.end
```

Fig. 8;

I



Condenser input filter

	RADIOTRON'S RESULTS	SIMULATION RESULTS
Vout	350Vcc	342Vcc
Repetitive peak current	375mA	400mA
Ripple Percentage	5.5%	7.79%

Choke input filter

	RADIOTRON'S RESULTS	SIMULATION RESULTS
20mA Load L=18.6H		
Vout	449Vcc	450.168Vcc
Repetitive peak current	40mA	50ma
220mA Load L=3.4H		
Vout	350Vcc	363.649Vcc
Repetitive peak current	330mA	382mA

Tab. 5;

Table 5 shows a comparison between manual calculations (as per the *Radiotron Handbook*) and SPICE simulations. The more meaningful graphic representations of such simulations are demonstrated in Fig. 9, 10, 11.

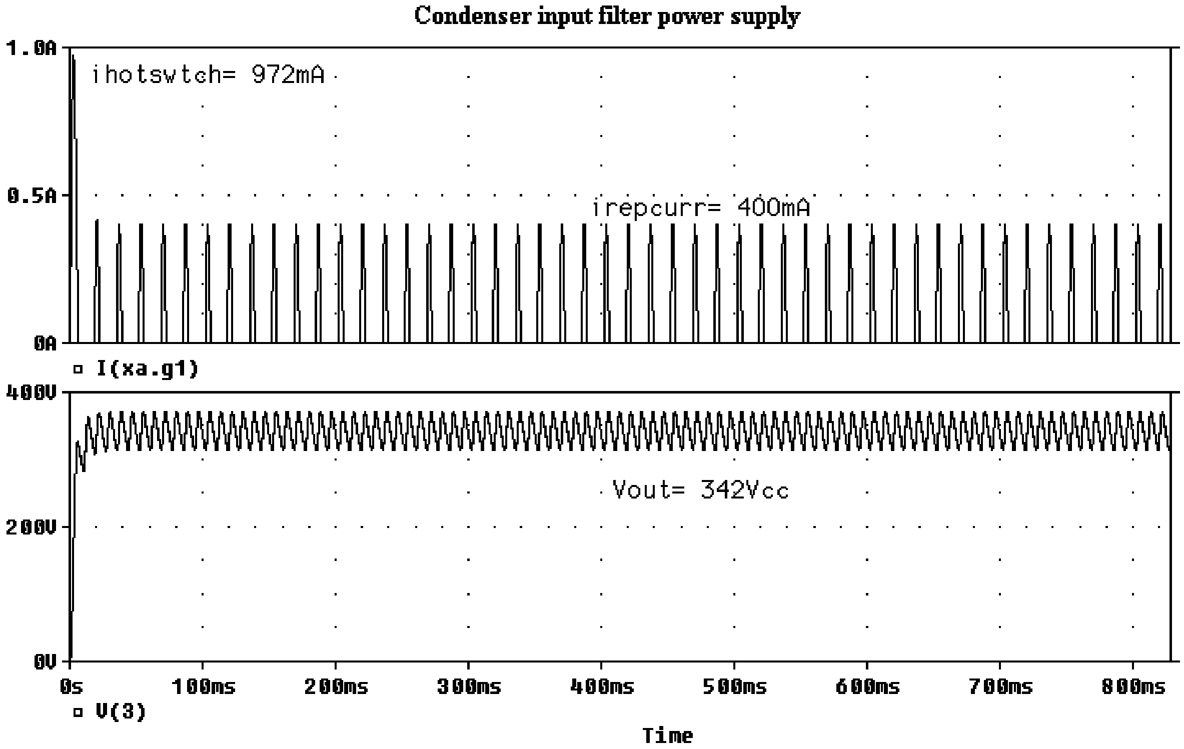

Condenser input filter power supply

Fig. 9;

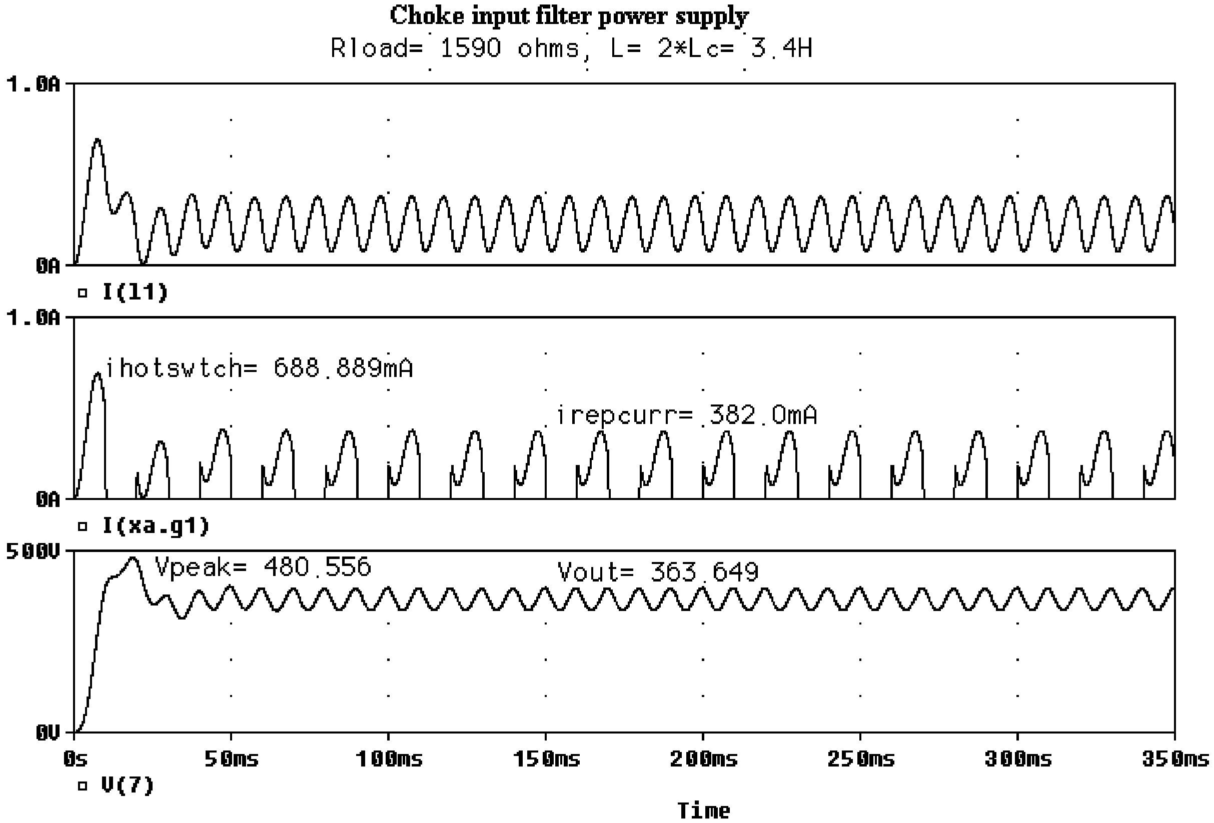

Choke input filter power supply
Rload= 1590 ohms, L= 2*Lc= 3.4H

Fig. 10;

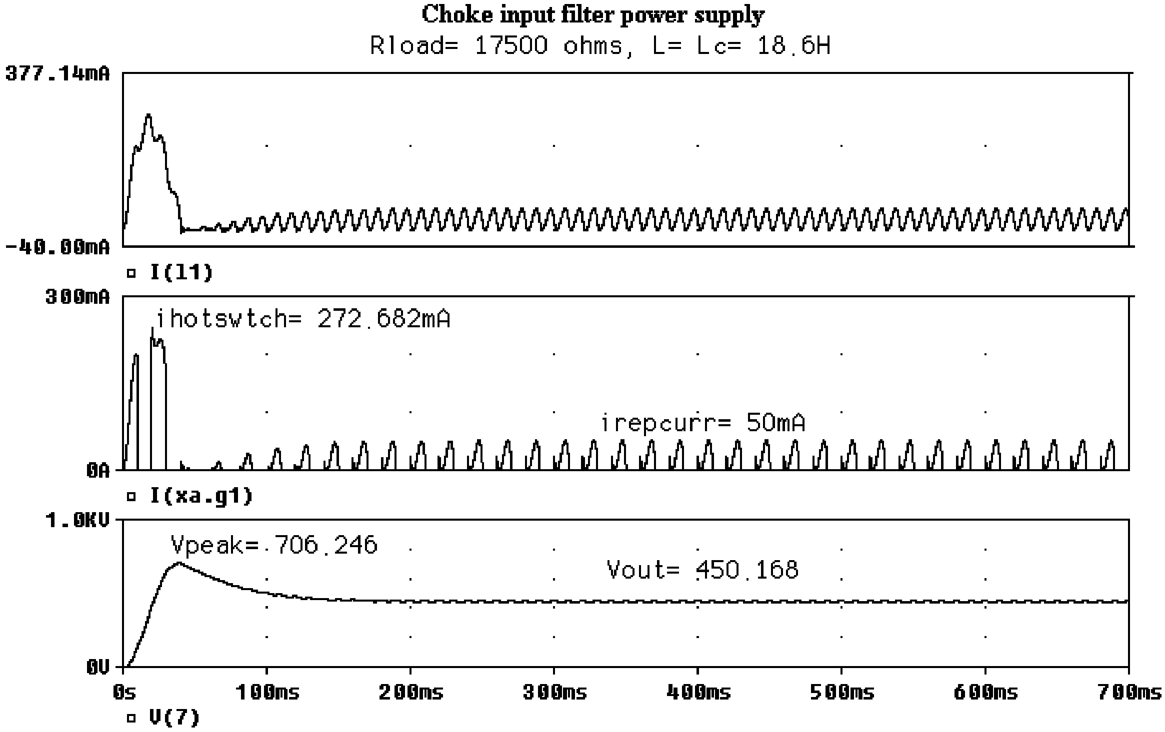


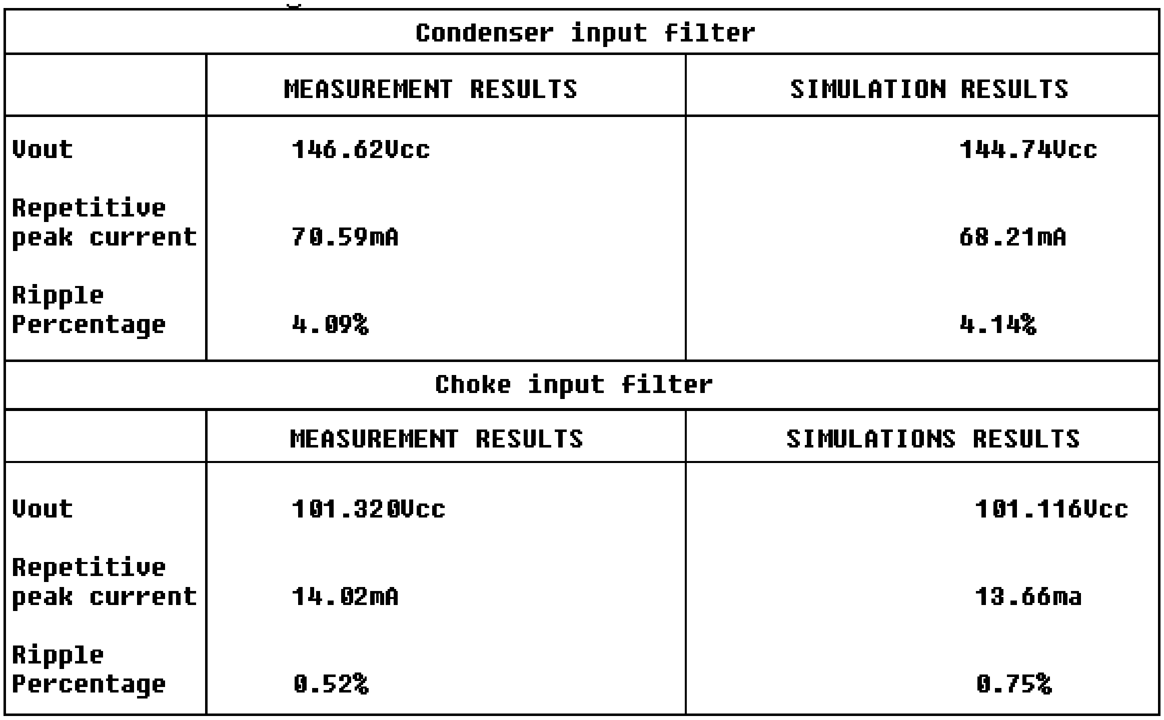
Fig. 11;

You will observe in Fig. 9 that it is rather easy to overcome the safety limits related to the impulsive currents with small input condensers. Therefore their capacitive value has to be valued very attentively. Fig. 11 further reveals that the swinging choke calculus is not very accurate.

An actual comparison

In Table 6 I present the results of two simple power supplies with condenser and choke input filter. Schematics are the same of Figs. 9 and 10 but with the following differences:

Vin1, Vin2@120Vrms;
R1= R2= 6.8 ohm;
L1= 30H in choke input filter power supply;
C1= 8.10 mF;
Rl= 10.16K;
12X4 vacuum-diode.



Condenser input filter		
	MEASUREMENT RESULTS	SIMULATION RESULTS
Vout	146.62Vcc	144.74Vcc
Repetitive peak current	70.59mA	68.21mA
Ripple Percentage	4.09%	4.14%
Choke input filter		
	MEASUREMENT RESULTS	SIMULATIONS RESULTS
Vout	101.320Vcc	101.116Vcc
Repetitive peak current	14.02mA	13.66ma
Ripple Percentage	0.52%	0.75%

Tab. 6;

The **12X4** SPICE model whose parameters are listed in line 1 of *Table 3* have been drawn just from the real diode used for the construction of the two power supplies explaining the reason for the apparent redundancy in Tables 1, 2, 3. Here the comparison is made with reference to **Sylvania's 12X4** sample. The utmost closeness of the numerical results in Table 6 makes me rather optimistic about the validity of the implemented model; the small differences that you can recognize are partly attributable to the errors of the measurement process in real world.

Conclusion

I hope this article can add a small wedge to the big mosaic of circuit simulation as applied to vacuum tube amplifiers. I believe the thermionic technology simulation related to audio amplification devices has not yet totally reached its real potential. The evidence is given by modern designs that incorporate concepts going back to the dawning of electronics (as well as harmonic cancellation, feed-forward, transformer coupling, choke input filter etc.), producing sonic results better than the crowd of Williamson-like amplifiers.

References:

Charles Rydel	SIMULATION OF ELECTRON TUBES WITH SPICE", AES preprint 3887 (G-2), 98th AES Convention, Paris 1995;
Scott Reynolds	VACUUM TUBE MODELS FOR PSPICE SIMULATIONS, GLASS AUDIO n. 4, 1993
W. Marshall Leach, Jr	SPICE MODELS FOR VACUUM TUBE AMPLIFIERS, JAES March 1995
Norman Koren	IMPROVED VT MODELS FOR SPICE SIMULATION, GLASS AUDIO, N. 5 1996
F. Langford-Smith	RADIOTRON DESIGNER'S HANDBOOK, 4th Ed. 1953
Piotr Mikolajczyk Bohdan Paszkowski	UNIVERSAL ELECTRONIC VADEMECUM