Intusoft Tackles ASIC Designs

Plessey Semiconductors and **intusoft** have combined forces to create an analog ASIC development system for the PC. Using Plessy's integrated circuit models along with intusoft's integrated ICAPS package, IC designers can now design, develop, analyze, and test their circuit designs before committing them to the foundry. Coupled with an IC layout package from Aptos, Inc., the circuit designer now has a complete IC development system for the PC . An application note utilizing Plessey's models and intusoft's software can be found inside the newsletter. For further information, contact Plessey, Marketing Division, Scotts Valley, CA. Watch for other analog ASIC model libraries and development kits from **intusoft.**

A SPICE Model for a Vacuum Triode

Yes, vacuum tubes are still alive and well! Solid state circuitry has replaced tubes in most low power applications, but the very high power needed for transmitters and certain pulse forming applications still requires tubes. In order to use SPICE to model vacuum tubes, it is necessary to construct a subcircuit model. Both diode and triode current depend on voltage raised to the 3/2 power. Pentode characteristics are close enough to MOSFET's to use the MOS level 3 model. This application will therefore concentrate on the diode and triode problem.

Modeling a 3/2 power function

A 3/2 power function, for example $E_B{}^{3/2}$, can be approximated using a polynomial. Remarkably, using a second order polynomial function gives excellent results. Unfortunately, the same set of polynomial coefficients cannot be used for all applications. Accuracy will depend on the maximum value of the dependent variable. For example; if the dependent variable has a maximum value of 100, then using the best coefficients calculated for a range from 0 to 500 would give greater error than coefficients calculated over the range of 0 to 100. The following coefficients were calculated using the INTUSCOPE polynomial regression feature. The error with respect to the maximum value is 1.8% for each set of coefficients. Additional data points were used for small outputs in order to force the error to be a constant percent of the output value. This technique is described in greater detail in the inlay on the oppposite page.

Making a Polynomial Function with 2 Inputs

The SPICE voltage controlled current source can be made a polynomial function of 2 inputs. Proper selection of the polynomial coefficients yields the triode transfer function. For the case of 2 inputs, the coefficients are:

$$I = P_0 + P_1E_1 + P_2E_2 + P_3E_1{}^2 + P_4E_1E_2 + P_5E_2{}^2$$

The triode equation is:

$$I_K \quad = \quad K_T(E_B/Mu + E_C)^{3/2}$$

Where I_K, E_C and E_B are defined in the following sketch. Both K_T and Mu are constants that describe particular tube characteristics.

Solving for polynomial coefficients

Using polynomial nonlinearities with SPICE is a powerful modeling technique. There is no single "best" set of coefficients because the "best coefficient" criteria depends on the application. Polynomial regression can be used to find the coefficients that give the smallest least squares error fit to a set of data. The data set for modeling is frequently defined by an equation; giving rise to an infinite set of data. Application of polynomial regression to a problem with an infinite set of data requires a means to select a finite set of data points. The method used to select the data will cause the resulting coefficients to differ.

Selecting the data

More data points should be used in regions where high simulation accuracy is required. In digital circuits; for example, data near the one - zero extremes should dominate. In the case of the vacuum tube model, it is useful to logarithmically space the data. Logarithmic spacing can be accomplished during data creation or by editing uniformly spaced data. To edit uniformly spaced data, decide on an interval; for example, use a decade interval. Then discard all data except near the decade boundary (1,10,100, etc.). High order polynomials can have ripple in regions where you would prefer straight lines. Fitting straight lines using polynomial ratios is the main subject of filter synthesis. Borrowing these techniques for SPICE modeling will be the subject of a future application note.

Solving for the coefficients

Polynomial regression is a straight forward mathematical procedure available in many computer programs and text's on programming. For owners of INTUSCOPE, the feature is built in. After getting the data in a suitable array (ASCII SPICE format) it is brought into the INTUSCOPE program and saved as a waveform file. While in SCOPE, the polynomial regression can be performed for various orders with the results seen on the screen. To make the SPICE polynomial, exit the program and run the Poly program; the SPICE coefficients will go to the standard output (your screen or a file if >file was attached to the command line). The coefficients are ready for a SPICE file; just paste them in using a screen editor.

Triode Equation *(Continued)*
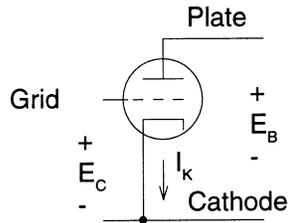
Approximating the 3/2 power function using a second order polynomial yields:

$$I_K = K_T K_1 E_C + K_T K_1/Mu E_B + K_T K_2 E_C^2$$
$$+ 2K_T K_2 E_C E_B/Mu + K_T K_2/Mu^2 E_B^2$$

Where:

$$K_T = I_{K0} / (E_{B0}/Mu + E_{C0})^{3/2} = 1.07E\text{-}3 \text{ for a 6SN7}$$

Table 1 can be used to select the coefficients K_1 and K_2. These coefficients depend on the maximum negative grid voltage and the maximum plate voltage. If the grid goes more negative than the maximum, then the second order term in the polynomial approximation will cause positive current for large negative inputs which is clearly incorrect. The column labeled $-K_1/K_2$ in table 1 sets this limit. For this example, the first set of coefficients satisfy the condition that Vmax < (Vplate / Mu); however the maximum negative grid voltage is 18 Volts which requires the use of the second set of coefficients.

Table 1, Polynomial coefficient for different maximum input voltage			
Max input(E)	K_1	K_2	$-K_1/K_2$
25	1.81	.13	-13.9
50	2.54	.0925	-27.4
100	3.934	.06175	-63
200	5.0	.046	-108
500	7.77	.0299	-267
output = $K_1 E + K_2 E^2$			

Therefore:

<div align="center">6SN7 Coefficients(Mu = 20)</div>

$$
\begin{aligned}
P_0 &= 0 \\
P_1 &= K_T K_1 & &= 2.72\text{E-}3 \\
P_2 &= K_T K_1/Mu & &= 136\text{E-}6 \\
P_3 &= K_T K_2 & &= 99\text{E-}6 \\
P_4 &= 2K_T K_2/Mu & &= 9.9\text{E-}6 \\
P_5 &= K_T K_2/Mu^2 & &= 247\text{E-}9
\end{aligned}
$$

For a given maximum voltage, the coefficients K_1 and K_2 can be derived as in table 1. The polynomial coefficients can then be computed as shown above using data sheet or measured values.

Completing the Triode Model

The polynomial function describes the triode for positive output current. Adding a diode, transistor and interelectrode capacitance as shown in Figure 1 completes the model. The diode and transistor are used to prevent negative current in the plate or cathode.

The curve family shown in figure 2 illustrates the static model characteristics along with the circuit listing. To convert the model to a diode, connect the grid to the plate and remove the grid related components. The polynomial function can be reduced from POLY(2) to POLY(1) when the grid is removed. If the triode is to be used with a forward biased grid and if grid current is an important parameter, then a vacuum diode model can be added across the grid to cathode.
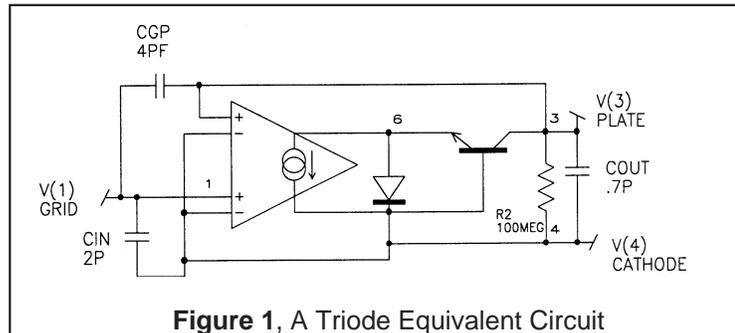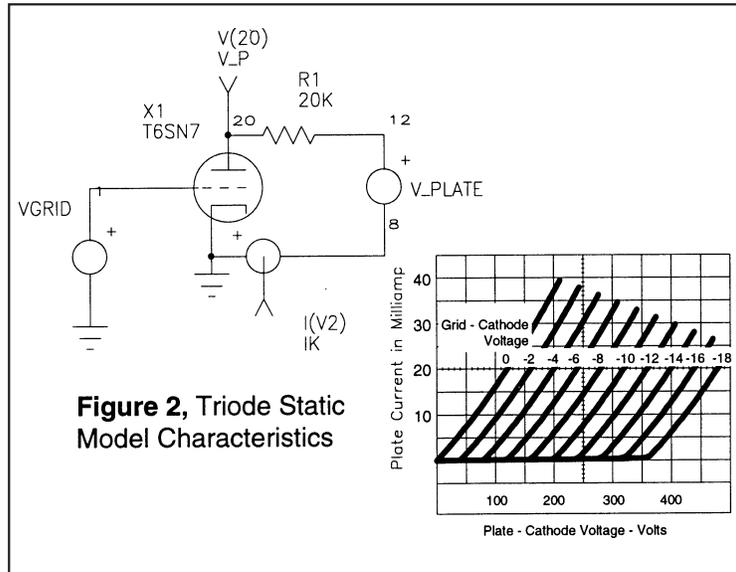

Figure 1, A Triode Equivalent Circuit

```
.SUBCKT  T6SN7  1  3  4
G1  6  4  POLY(2)  1  4  3  4
+ 0  2.72M  136U  99U  9.9U  247N
Q1  3  4  6  NPN
.MODEL  NPN  NPN
D1  6  4  DIODE
```

```
.MODEL  DIODE  D
CGP  1  3  4PF
CIN  1  4  2P
COUT  3  4 .7P
R2  3  4  100MEG
.ENDS
```

Figure 2, Triode Static Model Characteristics

```
TUBES
.OPTIONS LIMPTS=550
R1S  1  0  10K
*SPICE_NET
.DC VPLATE  0  1000  20  VGRID  0  -18  -2
*NCLUDE DEVICE2.LIB
*ALIAS  I(V2)=IK
*ALIAS  V(20)=VP
.PRINT DC  I(V2)  V(20)
VGRID  1  0
V2  0  8
X1  1  20 0 T6SN7
R1  20  12  20K
VPLATE  12  8
.END
```

R1S is used to complete the DC path to ground.

Monte Carlo Simulation of Analog IC's

Monte Carlo analysis is a powerful adjunct to a SPICE simulation that allows yield prediction for IC's. Since yield prediction deals with the central part of a statistical distribution, the simulation accuracy becomes adequate for fairly small samples.

The key to making a good prediction requires a knowledge of the statistics of device model parameters. Moreover, this knowledge must include statistical coupling and circuit parasitics that would skew the mean value. By way of example, the performance of 3 different current mirrors will be evaluated relative to variations in forward current gain; the BF or Beta parameter in SPICE. These examples will be taken in the

mid-region of collector current, where the IKF and ISE terms are relatively unimportant. The sketch below shows the importance of this assumption since operation outside of this region would invalidate the statistical model.
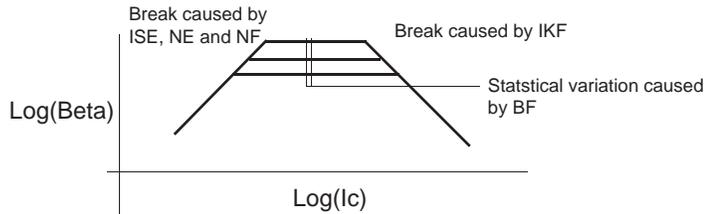

Sketch of Beta vs Current

The 3 current mirror configurations shown in figure 3 are the standard current mirror, the Wilson current mirror and a modified Wilson current mirror. The relative merits are described in the figure captions. The modified Wilson mirror has the potential of achieving accuracies on the order of 0.1% with reasonable IC yield. This configuration can be used for precision analog operation without external trimming components.
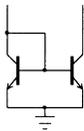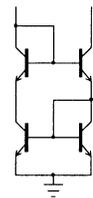

Standard Current Mirror
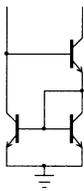
Wilson Current Mirror

Reduced sensitivity to output voltage

Reduced sensitivity to Beta if transistors are matched

Modified Wilson Current Mirror

Reduced sensitivity to output voltage

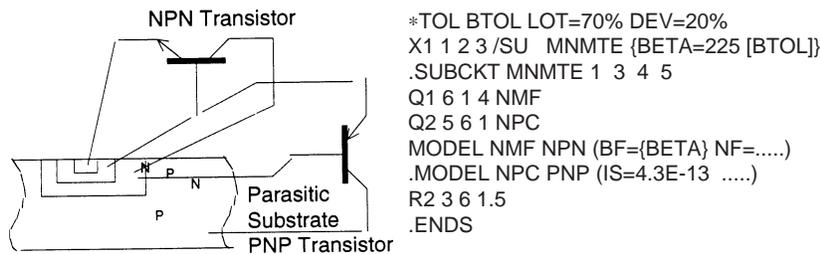Reduced sensitivity to Beta if transistors are matched

Reduced error in gain by matching Vce in lower transistors

Table 2 shows test results obtained using Plessey Semiconductor's component models. The first 3 comparisons were made using single NPN transistors. Paralleling more transistors should improve the statistics by the square root of the number of devices connected in

parallel. It would be more efficient to take advantage of this knowledge and use an area scalar in a manner that would divide the device tolerance by the square root of the device area. To validate this assumption, a simulation was run in this manner and another simulation was run using parallel transistors. The statistical data compare favorably, suggesting that this is a valid method when BF is the only statistical variable.

Table 2, Current Mirror Statistics Summary .					
Mirror Type	Standard	Wilson	Modified Wilson		
Area	1	1	1	9	1*9
Gain Error Mean	3.52	-.723	.015	.0284	.0371
Gain Error Std. Dev.	.262	.117	.136	.050	.0395

Notes:
1. Current density is the same for all cases.
2. When area is increased, device tolerance is divided by sqrt(area)
3. 1*9 means same statistics as area == 1 with 9 parallel devices
4. Gain error and deviation are in percent where a gain of 1 is taken to be 100%.
5. Data is based on 30 samples with Gaussian statistics.

In order to run the Monte Carlo simulation using ICAP/2 or ICAP/3, it is first necessary to include all device models that have statistical parameters within a subcircuit used to represent the transistor. Then the statistical parameter can be defined by equation; in this case the trivial statement BF={BETA} is used in the model. The following sketch shows the NPN model, its schematic and salient model parameters. For the complete model listing you must contact Plessey Semiconductor. It is important to evaluate the parameters <u>after</u> the statistics are assigned. This will cause each transistor to have a different BF. If BF was given a statistical parameter within the model, such as BF=225 TOL=20%, then the same transistor model would be used for each device, making the statistical analysis worthless. Turning on the ICAPS switch "Param after Monte" in the simulation menu achieves this result.


```
*TOL BTOL LOT=70% DEV=20%
X1 1 2 3 /SU   MNMTE {BETA=225 [BTOL]}
.SUBCKT MNMTE 1  3  4  5
Q1 6 1 4 NMF
Q2 5 6 1 NPC
MODEL NMF NPN (BF={BETA} NF=.....)
.MODEL NPC PNP (IS=4.3E-13  .....)
R2 3 6 1.5
.ENDS
```

Intusoft Multiple Winding Transformers

There are a number of SPICENET symbols which exist merely as artwork. That is, they do not have a SPICE subcircuit that they represent. They can be used as templates for your own symbols or just as artwork to represent a design concept. One such symbol is called "XFMR-TAP", which represents a multiple winding transformer. For this symbol to be functional, a SPICE netlist must be developed for it. To generate a SPICE netlist for this symbol we can utilize an ideal transformer. The symbol and subcircuit for an ideal transformer is shown below.
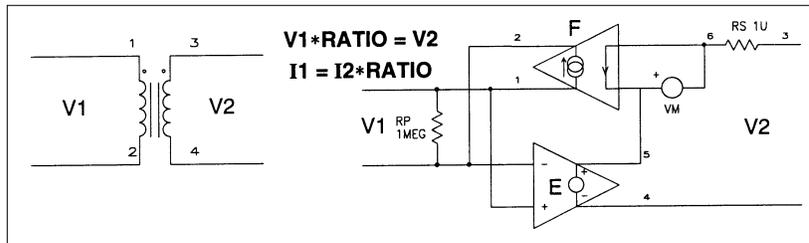
Although coupled inductors could be used to build a transformer, the controlled sources based model will have an ideal behavior; suitable for use with a saturable core. The ***intusoft*** model is more sophisticated than some other vendors models because of the parameter passing feature built into the model. This one transformer may take on any primary to secondary ratio simply by specifying the RATIO to the model - allowing one model to represent many transformers. This basic model can be extended to build more complicated topologies.

The schematic symbol, transformer topology, and corresponding subcircuit netlist is shown below. By using the ideal transformer, a myriad of combinations may be generated.

```
X1  1  2  3  4  5   XFMR {RATIO=xxx }

.SUBCKT  XFMR-TAP  1 2 3 4 5
E1   7 8 1 2 {RATIO}
F1   1 2 VM1 {RATIO}
RP  1 2 1MEG
RS  6 3 1U
VM1  7 6
F2   1 2 VM2 {RATIO}
E2   9 5 1 2 {RATIO}
R5  8 4 1U
VM2  9 8
.ENDS
```

XMFR-TAP Netlist   **SPICENET Symbol**   **Actual Topology**

Multiple winding transformers may be built out of combinations of the ideal transformer. RATIO equals V2/V1. The transformer is center tapped and V2 will equal V3.