Agilent Technologies
Innovating the HP Way

# Diode Detector Simulation using Agilent Technologies EEsof ADS Software

## Application Note 1156

### Introduction

This application note has been written to demonstrate how the Agilent Technologies EEsof ADS software package can be used to simulate a diode detector circuit reliably against temperature.

For this demonstration a simple two diode configuration has been proposed. This has the benefit of offering temperature compensation of the diode's bias-induced forward voltage. The circuit used in this demonstration is shown in Figure 1 below.

This circuit was designed around the Agilent Technologies Schottky diode, HSMS-2865, which was developed for low cost, high volume, high frequency detector applications.

### Detector Circuit

The input impedance in this example was chosen to be 33 Ohms. L1 and C2 are providing the reactive match, while R1 provides a good broadband resistive match.

The bias is applied to the first diode via R1, and R2 acts as the resistive load for the detector's output voltage. The second diode has an identical DC chain, and therefore the same forward current, assuming

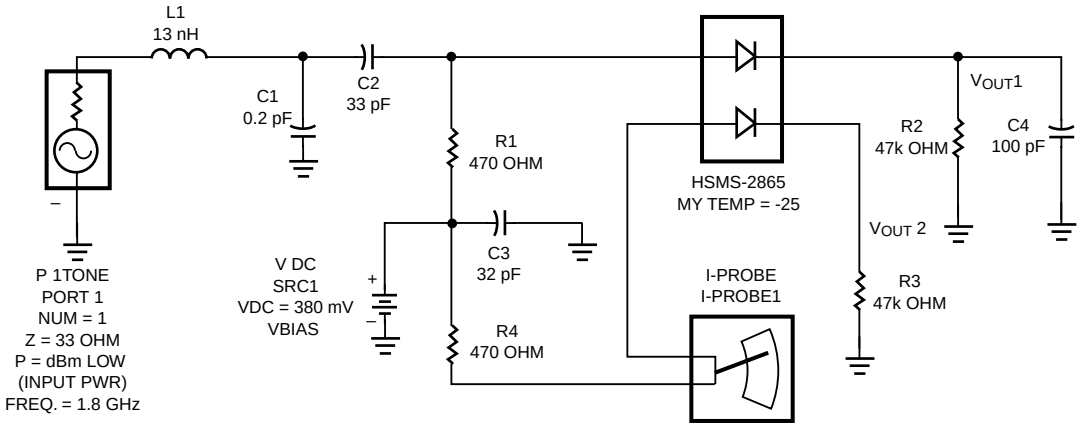



**Figure 1. Diode Detector Circuit**

the diode's dc characteristics are identical. To achieve almost identical dc characteristics, the dice in a SOT-143 are selected from adjacent sites on a wafer, thus ensuring the best possible match between the two diodes. Vbias is selected in this example to give a bias current of 5 μA.

At the node Vout1 there will be a DC voltage equal to the detected voltage Vo, plus the forward voltage, Vf, due to DC bias. At Vout2 there will only be Vf, the forward voltage generated by the dc bias. As we are measuring the voltage across Vout1 and Vout2, we should only be measuring Vo, the detected voltage. Any variation in Vf due to temperature should be cancelled.

As the input power and temperature are varied the S11 of the diode circuit will also vary. This variation is seen in Figure 8 (further detail on Large Signal S-Parameter simulation is given later). The reason for this S11 variation with input power can be easily seen in the junction resistance (Rj) and the saturation current (Is) calculations below.

$$Rj = \frac{nkT}{Is + Io}$$

$$Is = Iso\left(\frac{T}{298}\right)^{\frac{2}{n}} e^{-4060\left(\frac{1}{T} - \frac{1}{298}\right)}$$

Where:

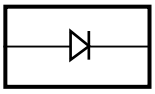
Rj = Junction Resistance
n = Diode Ideality Factor
k = Boltzmann's constant
T = Temperature in ° Kelvin
Is = Saturation Current
Io = Bias Current
Iso = Saturation current at 25°C

The denominator of the junction resistance equation is the diode's own saturation current and the externally applied bias current, Io. Within the circuit, there will also be a third current, the circulating current, Ic = Vo/RL, produced by the rectification in the diode. Under small signal operation Ic is very much less than Is, and can therefore be ignored. However, as the input levels are increased, and the diode is moved into the non-linear region of operation, Ic will increase and cause a corresponding change in Rj and hence change the input impedance of the circuit.

The Output voltage is generated across R2 and R3, with the measurement being taken across nodes Vout1 and Vout2.

## Diode Modeling

To simulate the diode performance in ADS, the non-linear PN junction diode model was used (The PN junction model can be used for a Schottky diode, assuming that Eg is set to 0.69). Agilent Technologies publishes SPICE models for all of its Schottky diodes. These parameters can be entered into the model as seen in Figure 2. Before the diode model can be effectively used the package model[3] must also



DIODE MODEL
b286DIE

| | |
|---|---|
| Is = 5e - 8 | Isr |
| Rs = 5 | Nr |
| N = 1.08 | Ikf |
| Ti = | Nbv |
| Cjo = 0.18e - 12 | Ibvl |
| Vj = 0.65 | Nbvl |
| M = 0.5 | Tnom |
| Eg = 0.69 | Ffe |
| Imax | |
| Xti = 2 | |
| Kf | |
| Af | |
| Fc | |
| Bv = 7 | |
| Ibv = 10 e - 5 | |

**Figure 2. HSMS-286X Die Model**

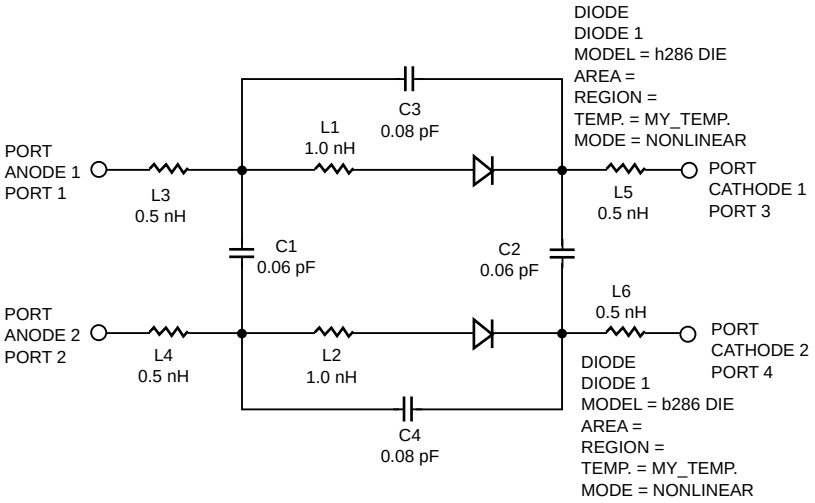



**Figure 3. HSMS-2865 Packaged Diode Model**

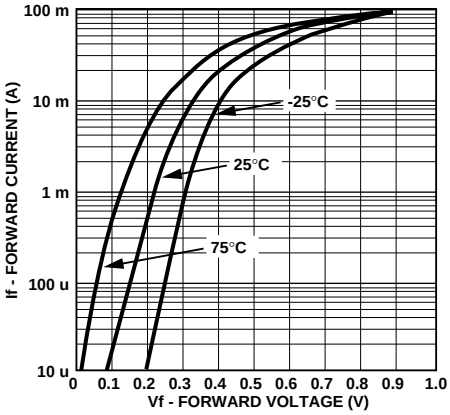



**Figure 4. If vs. Vf over Temperature**

be included (contact Agilent Technologies for further details on package models). This can be achieved by using lumped element components. The finished model for the HSMS-2865 diode can be seen in Figure 3.

The Diode model in Figure 2 specifies Tnom, the nominal temperature at which the SPICE parameters were extracted. By default this parameter is set to 27°C.

The PN junction diode symbol within ADS has the facility to set the physical temperature of operation. This temperature is different than the model Tnom. When a temperature is entered at the symbol level, ADS will scale Eg, Is, Isr, Cjo, and Vj[4].

Modifying the component variable my_temp to the desired value varies the diode temperature. The variable my_temp is seen in Figure 1 and Figure 3.

A confirmation of the temperature scaling can be seen from the simulated diode VI curves shown in Figure 4.

## Non-Linear Circuit Simulation

A simulation of the DC (Video) output voltage of the circuit versus RF input power can be achieved by using the Harmonic Balance simulator. Figure 5 shows the simulator configuration, with associated variables.

The desired DC output voltage from the circuit is measured between node Vout1 and Vout2. To output this voltage from the simulator you can use the equation Vfc as shown in Figure 5. The Vfc function is used to measure a frequency selective voltage between two nodes. [Our_vfc = vfc (vnode1, vnode2, Freq)]. In this example, our _vfc = vfc (Vout1, Vout2, 0 GHz).

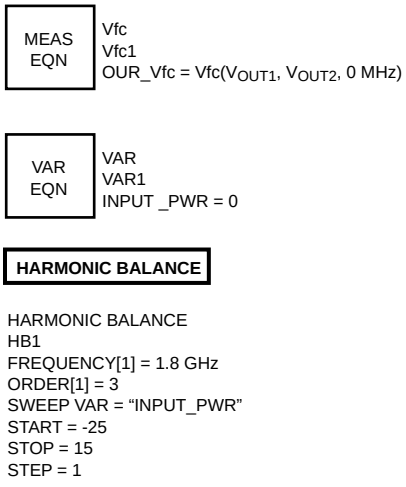



**Figure 5. Harmonic Balance Simulation Configuration**

This simulation was repeated three times, once at -25°C, 25°C, and 75°C. Each time the my_temp variable is set to the appropriate value, and the circuit is simulated using a different dataset (This enables all three temperatures to be displayed on the same graph). Figure 7 shows the results of the simulations. A close-up of the temperature variation can be seen in Figure 9.

By changing the circuit simulator from Harmonic Balance to Large Signal S-Parameters (LSSP) you can measure the S11 of the circuit against RF input power and temperature. Figure 6 shows the simulator configuration for LSSP analysis.

Figure 8 shows the results of this simulation, indicating that the match remains very good over a broad range of input power and temperature.

## Summary

This application note has demonstrated a useful technique for accurately simulating diode detector circuit performance against RF power and temperature using Agilent Technologies EEsof ADS software.

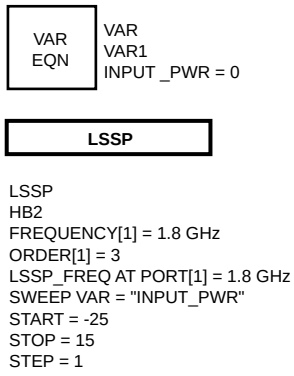



**Figure 6. LSSP Simulator Configuration**

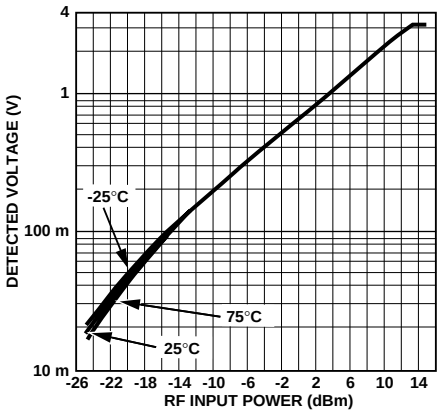



**Figure 7. Detector Output Voltage vs. RF Input Power Over Temperature**

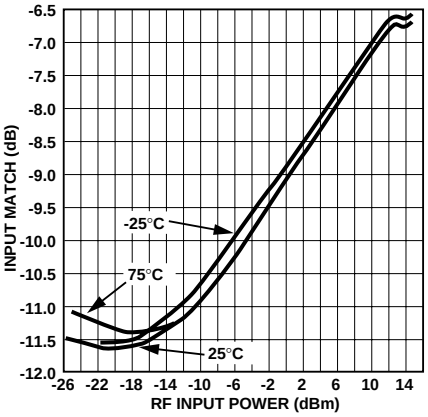



**Figure 8. Input Match (S11) vs. RF Input Power**

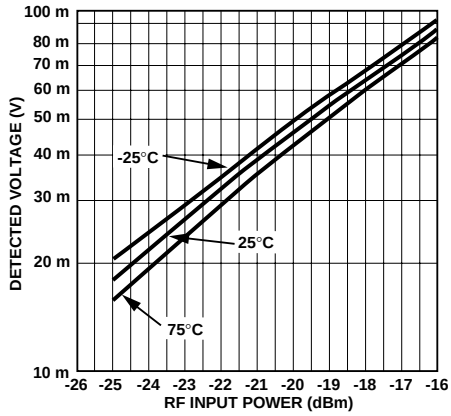



**Figure 9. Close-up of Temperature Variation**

**References**

2. Agilent Technologies Application Note 956-4, *Schottky Diode Voltage Doubler*
3. Agilent Technologies Application Note 1124, *Linear Models for Diode Surface Mount Packages*
4. Agilent Technologies EEsof *Circuit Components Manual for ADS*



5968-1885E (11/99)

# 射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

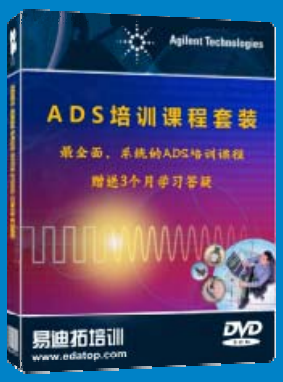
易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html


## 射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

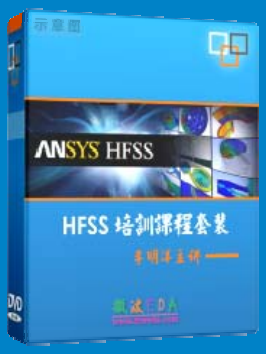
课程网址：http://www.edatop.com/peixun/rfe/110.html

## ADS 学习培训课程套装




该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...

课程网址： http://www.edatop.com/peixun/ads/13.html




## HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

## CST 学习培训课程套装


该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

**课程网址**：http://www.edatop.com/peixun/cst/24.html


## HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

**课程网址**：http://www.edatop.com/peixun/hfss/122.html

## 13.56MHz NFC/RFID 线圈天线设计培训课程套装


套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

**详情浏览**：http://www.edatop.com/peixun/antenna/116.html

## 我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

## 联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com