

Lab 1: Diode Characterization

1 Task

Electronics Design Automation (EDA) has been dominating in present circuit design practice. It greatly reduces the time for circuit debugging, and component selection. Among them, SPICE is most widely used electronic circuit simulation tool. Its wide acceptance as an industry standard for analog circuit simulation and electronic design verification is mostly due to its sophisticated device models which are based on device physics and yield accurate representation of device terminal behavior. Therefore, if the components at hands do not have ready-made models, fast and accurate measurement of the SPICE parameters is of great importance for the next step in SPICE simulation.

As an electrical engineer, you are going to simulate and build a circuit prototype with the available components in the lab. In the beginning of this project, you are given a task to build a SPICE model for a diode marked as 1N4148 to be used in future simulation.

Generally, you will directly consult the manufacturer's website for a SPICE model. Or alternatively, you can build a SPICE model from those parameters listed on the datasheet. Unfortunately, the markings on the diode do not show the manufacturer's information. You know it is a commonly used diode that quite a number of manufacturers sell it with diversified models. There is no way to figure out which model would best describe the diode to be used.

Therefore, you refer to a third approach, which is to a) extract the SPICE model parameters from direct measurements of an actual device and then b) verify by comparing the test and simulation results.

2 Pre-Lab:

2.1 SPICE model for diode:

.model <modname> <type> (parameter list)

Example:

```
.model 1N4148 D (Is=2.52n Rs=.568 N=1.752 Cj0=4p M=.4 tt=20n Iave=200m  
+Vpk=75 mfg=Motorola type=silicon)
```

The parameters in the list are explained in Table 1.

Table 1: SPICE parameters of Diode in LTspice

Parameters	Name	Descriptions
Is	Saturation current	Key parameters for DC characteristics
Rs	parasitic resistance	
N	ideality factor	
M	Junction grading coefficient	0.33 for linearly graded junction and 0.5 for abrupt junction.
CJO	zero-bias junction capacitance	Key parameters for transient response
TT	transit time	
Iave	average current	
Vpk	break down voltage	
Mfg	manufacturer	
Type	Material type	decide the energy band

Among these parameters, the most important parameters are I_s , R_s and N , which determines the DC characteristics of the diode. If a high speed transient response is needed, such as high frequency switching mode power supplies, TT and CJO parameters need to be found. In this experiment, you will determine the DC parameters only, i.e. I_s , R_s and N .

2.2 Parameter Extraction:

There are different ways to model a diode.

Readings: Google and study “the piecewise linear model” and “the exponential model” of diode. **Question:** how can we find out the forward voltage drop of the diode in datasheet?

In the exponential model, the relationship between I_D and V_D for a diode is given by an exponential equation:

$$I_D = I_s \cdot \left(e^{V_D / (NV_T)} - 1 \right) \quad (1)$$

Where I_s and N is the reverse saturation current and the ideality factor in Table 1. The thermal voltage is given by $V_T = kT / q$ where $k = 1.38 \times 10^{-23} \text{ J/K}$ (Boltzmann’s constant); T is the absolute temperature in Kelvin and $q = 1.602 \times 10^{-19} \text{ C}$ (Electronics charge). Therefore, at room temperature of 27°C , the thermal voltage is 25.8mV .

According to (1), it is assumed that the influence of R_s is ignored ($R_s = 0$). Therefore, the simplest extraction method would be to measure two operating points (I_D, V_D), which would form two equations with two unknown variables I_s and N . Solving these two equations would give you the answer. However, this approach is prone to experimental error.

In practice, there are three types of errors in the measurements. Firstly, experimental error or random error is unavoidable; because, no matter how accurately one performs the measurement, there will always be some uncertainty due to noise. Secondly, due to the limitation of the measurement equipments or methods adopted, there will be some systematic error. Thirdly, personal or human error also exists. Great efforts have been taken in experimental measurements to remove the third error while reducing the first and second errors.

In order to reduce the random error in measurements, one common practice is to use more data sets to do curve fitting. In the case of diode characteristics, a simple linear curve fitting based on (1) is possible to extract I_s and N .

3 Resources:

Table 1: List of hardware and software

Name	Qty	Description
PC	1	For simulation, test and mathematical analysis
NI ELVIS II	1	Test platform including software installed on PC
LTSpice	1	Circuit Simulation Software
Excel / Matlab or others	1	Mathematical analysis tool

Table 2: Component list

Name	Qty	Description
Diode 1N4148	1	DUT (Device Under Test) with datasheet attached

● NI ELVIS II

The National Instruments Educational Laboratory Virtual Instrumentation Suite (NI ELVIS II), integrates 12 laboratory instruments in one compact platform and hence is ideal for hands-on learning for students. In this lab, we will use one of the 12 instruments: “Two-Wire Current-Voltage Analyzer” to sweep the IV curve of the diode. For more information about the usage of NI ELVIS II platform, including both hardware and software, please refer to “A brief

guide of NI ELVIS II system”.

- **LTSpice**

LTSpice is a SPICE circuit simulation software, which includes thousands of models of commercially available products. You may download a copy of LTSpice IV from the website below for free. Also available on the same page is LTSpice Users Guide and LTSpice Getting Started Guide from Linear Technology. A short introduction of LTSpice can be found in “A brief guide of LTSpice”.

<http://www.linear.com/designtools/software/#LTSpice>

- **Excel**

Knowledge of mathematical analysis such as curve fitting is prerequisite for this module. Please do a self study on how to do it using any math software you like. Microsoft Excel is one of the options.

4 Lab procedure:

As shown in Figure 1, “Two-Wire Current-Voltage Analyzer” of NI ELVIS II will sweep the voltage V_{sweep} across the DUT (Device Under Test). The DUT voltage and current (obtained by an internal sensing resistor R_{sense}) will be measured and displayed on the SFP (Soft Front Panel).

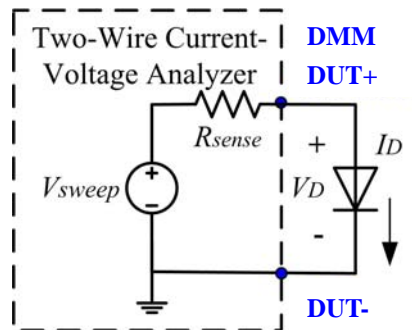


Figure 1: Test setup for measurement of diode I–V characteristics

Recommended settings for Two-Wire Current-Voltage Analyzer:

- **Voltage Sweep:** Start 0 V; Stop: 0.8 V (maximum values: ± 10 V); Increment: 0.01 V;
- **Current Limits:** Negative: -10mA; Positive: +10mA. (Maximum values: ± 10 mA)

- 1 Connect the diode under test between DMM DUT+ and DUT- terminals as shown in Fig.1.
Set up the test settings properly. Measure the current-voltage characteristics of the given diode using “Two-Wire Current-Voltage Analyzer” in NI ELVIS II. The signals can be measured is limited to within ± 10 V and ± 10 mA.
- 2 Extract the steady state parameters in diode’s PSPICE model (N , I_s) from the measured I-V curve.
- 3 Find out the current-voltage characteristics of the given diode in simulation using calculated PSPICE model parameters as Figure 2. Compare the simulation result with the test result.

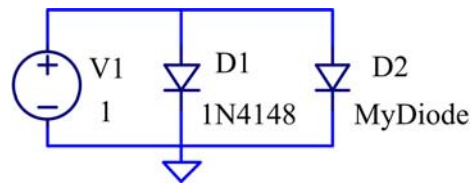


Figure 2: LTSpice simulation circuit for diode I-V sweep
(D1 uses internal model “1N4148”; “MyDiode” of D2 refers to your own model name)

As shown in Figure 2, by connecting the diode under study to a voltage source, the current through each diode vs. the voltage can be drawn using DC sweep function in LTSpice.