# Assignment 1

EEE2045F
Analog Electronics
Due Date: 23th April 2018

March 29, 2018

---

*This Assignment relates closely to the covered topics from the Electronic Devices by Thomas Floyd as well as the Introductory Lecture on LTSpice. Compile your submission in the form of a technical report. The layout of this report should explain the problem you are intending to investigate, the experiments you have used to explore the problem as well as the screen-shots pertaining to the results.*

**Please note this is an individual assignment which would be MARKED and your reports will be checked for plagiarism using Turnitin.**

---

## 1. VI Characteristics

(a) Connect the 1N914 diode in forward-bias with a voltage source and resistor and plot its V-I characteristics.

*Hint: Set the simulation spice directive to a linear DC sweep between -10V and 10V with increments of 0.1V*

(b) Change the 1N914 diode model by adding a reverse breakdown voltage at 5V and plot its V-I characteristics.

*Hints:* ***BV*** *is the breakdown attribute, which can be edited by by going to:*
`c:\Program Files\LTC\LTspiceIV\lib\cmp\standard.dio`
*or by adding the SPICE model directive that follows the form:*
`.model 1N914 D(bv=x)`
*Where x is the reverse break down voltage value that need you need to set.*

## 2. Clamping Circuits

Simulate the clamping circuit shown in Fig. 1 with the following parameters:

- The voltage source should be sinusoidal with amplitude of 10V and frequency of 100Hz.
- There should be a current limiting resistor with a resistance of $1M\Omega$.
- The diodes should use `.model 1N750 D(type=Zener bv=4.7)` as their model.
- The simulation should be run using transient analysis for 0.1 seconds.

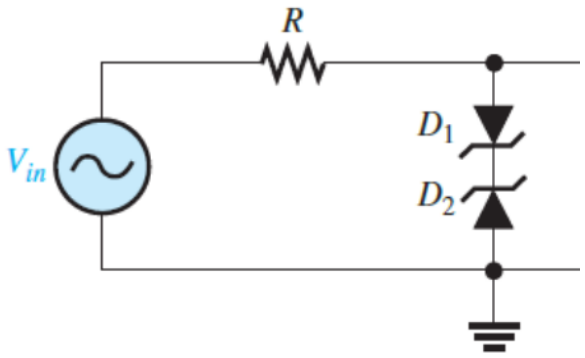


Figure 1: Zener Diode Clamping Circuit

(a) Plot the voltage output of this clamping circuit.

(b) Explain why the results shown in **(a)** were obtained.

(c) Vary the reverse breakdown voltage and explain the results.

## 3. Diode SPICE Parameters

Open `Diode Model.asc` that can be found on Vula and you should see the same circuit diagram as shown in Fig. 2. You should be able to immediately understand what this

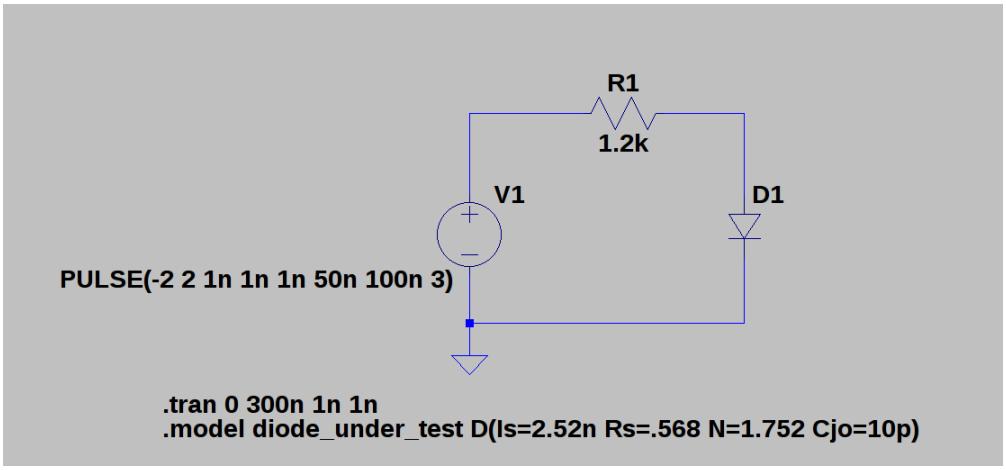


Figure 2: Diode Model LTSpice circuit

circuit should be doing, however when you run it, you will find that the output voltage is not characteristic of an ideal diode.

(a) Which model parameter of the diode is causing this uncharacteristic waveform?

(b) Explain how to rectify this problem and prove this with the 'fixed' output waveform.

*HINT: Look at the diode model parameters in Table. 1*

Table 1: SPICE Diode Model Parameters

| **Name** | **Parameter** | **Unit** | **Default** |
|---|---|---|---|
| IS | Saturation Current | A | 1.0e-14 |
| RS | Ohmic Resitance | Ohm | 0 |
| N | Emission Coefficient | - | 1 |
| TT | Transit Time | sec | 0.1ns |
| CJO | Junction Capacitance | F | 0 |
| VJ | Junction Potential | V | 1 |
| M | Grading Coefficient | - | 0.5 |
| EG | Band Gap Energy | eV | 1.11 |
| XTI | Saturation Current Temperature | - | 3.0 |
| KF | Flicker Noise Coefficient | - | 0 |
| AF | Flicker Noise Exponent | - | 1 |
| FC | Forward Bias Coefficient | - | 0.5 |
| BV | Reverse Breakdown Voltage | V | infinite |
| IBV | Current at Breakdown Voltage | V | 1.0e-3 |
| TNOM | Parameter Measurement Temperature | deg C | 27 |

## 4. LTSpice Recap

(a) From your understanding, when would you use each of the following simulation commands in LTspice?

i. DC Operating Point

ii. Transient

iii. AC Analysis

iv. DC Sweep

## 5. Common Emitter Amplifier

(a) Design and implement a Common-Emitter Amplifier for a gain greater than 3 to drive a load of 100K. Show calculations and output plots for the voltage amplification. In addition to this design answer the following questions:

i. How can you increase the gain without affecting the DC bias?

ii. How far can you drive the output before the voltage starts to distort?
*HINT: Vary the input sine wave to a higher value to observe output variations. Its normally very small in mV.*

iii. Run AC simulations to observe the frequency response What is the frequency range of this amplifier you designed? What are the upper and lower cutoff frequencies? How can you change the lower cutoff frequency of your amplifier?

*NOTE: This assignment will count toward your course grade.*