	Diode 03 - Diode at small signal
This project deals with models used to obtain the response of a diode against small signals.	
BOM	Diode: 1N4148 Resistor: 1 kΩ

Index

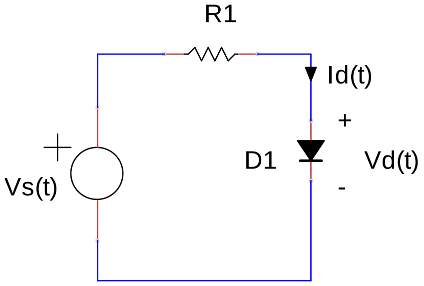
The small signal problem

In the previous diode project we have obtained a **constant voltage diode model** that eases the analysis of circuits that contain diodes. This model is based on making an hypothesis for the diode state, use a simple equation for this hypothesis and finally check that the hypothesis is true. The following table shows this model.

Hypothesis	Diode equation	Hypothesis check
ON	Vd = Vγ	Id ≥ 0
OFF	Id = 0	Vd ≤ Vγ

We call it the constant voltage diode model because, when conducts, we suppose the diode voltage equal to a constant Vγ value that is about 0.7 V for normal silicon diodes.

In general the model works quite well for DC calculations, but there are cases when a small variation of the Vd voltage is important. Here the model fails as it supposes a constant voltage on the diode. We will explain this problem with an example.



Consider the above circuit where Vs(t) is a voltage source whose value changes with time. Let's suppose that Vs(t) is defined so that D1 is always in ON state. Our simple model predicts the following values for Vd and Id:

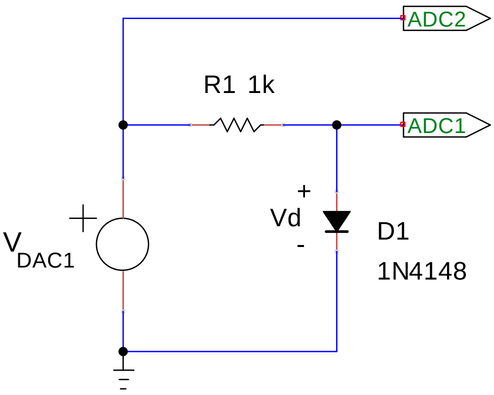
$$V_d = V_\gamma \qquad\qquad I_d = \frac{V_S(t) - V_\gamma}{R1}$$

So the model predicts that Id depends on time and Vd does not. There is a big difference in having a small Vd(t) signal or having no time variation on Vd at all.

In order to test the model against a real circuit lest work with the circuit using an 1N4148 diode, a 1 kΩ resistor and a V_S sine wave with 0.5V amplitude centered at 2.5V. That is, with a minimum at 2V and a maximum at 3V.

	1	Using the constant voltage model with Vγ = 0.7 V, obtain the maximum and minimum values of the current on the diode.

Now we want to measure the circuit to check the calculations. The following schematic shows the circuit we must implement on the breadboard together with the SLab connections.



Open a Python window, import the slab module, connect with the board and set DAC 1 to generate a 100 Hz sinewave with minimum at 2V and maximum at 3V. We will also instruct the board to provide measurements of 5 full waves for two ADCs (ADC1 and ADC2).

```
>>> slab.waveSine(2,3,100)
>>> slab.setWaveFrequency(100)
>>> slab.tranStore(500,2)
```

Now we perform a measurement and store the results.

```
>>> t,vd,vs = slab.wavePlot(returnData=True)
```

Observe that the diode voltage measured on ADC1 is not really constant.
Let's calculate and show the current, in mA:

```
>>> id=vs-vd
>>> slab.plot11(t,id,"","t (s)","id (mA)")
```

Note that the current (in mA) is equal to the voltage difference (in Volt) because the resistor is 1 kΩ.

You can locate the maximum and minimum values on the curve or you can ask SLab to obtain them for you, adding also the peak to peak value to the set:

```
>>> slab.highPeak(id)
>>> slab.lowPeak(id)
>>> slab.peak2peak(id)
```

Now, obtain the maximum, minimum and peak to peak values of the diode voltage:

```
>>> slab.highPeak(vd)
>>> slab.lowPeak(vd)
>>> slab.peak2peak(vd)
```

Take note of those values because you will need to refer to them in the future.

🛠	**2**	Perform the proposed measurements. Compare the maximum and minimum Id values with the ones previously calculated.

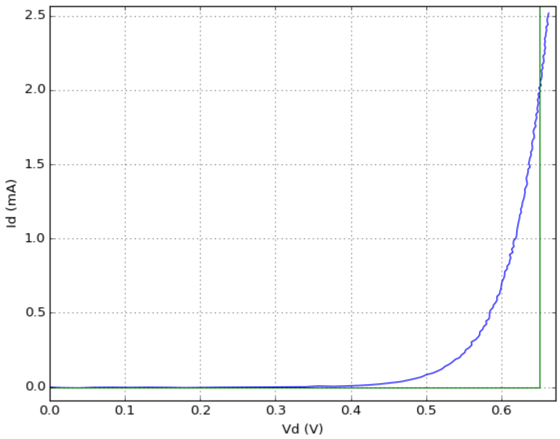
You should see a good agreement in the Id value. The model, however, predicts that Vd is constant so its peak to peak should be zero, and, although the peak to peak value is small, it is not zero.

The model fails miserably because there is a big difference between having a small signal on Vd and not having any signal at all.

We need a better model. A model that can give a rough approximation of the real Vd amplitude without needing to complicate too much the calculations.

Diode small signal model

If we recall from the previous project, the constant voltage diode model suppose that the diode voltage is constant when it conducts. The following figure shows the model for $V\gamma = 0.65V$, in green, against the real diode curve, in blue.



As the diode model is vertical when current flows, it predicts no voltage change at all. But if we see the real diode curve, in blue, we can see that diode voltage, in fact, changes as current changes.

The easy way to change the constant voltage diode model is to substitute the vertical green line with a line that is tangent to the diode curve. But tangent at which point? We can see that the tangent will have a different slope depending on the chosen point.

That is where the "**small**", of the small signal model, comes into play. If the voltage change in the diode is small enough, the slope will have a small negligible change. We will define the circuit quiescent DC operating point as the operating point on the circuit where the slope of the diode is calculated.

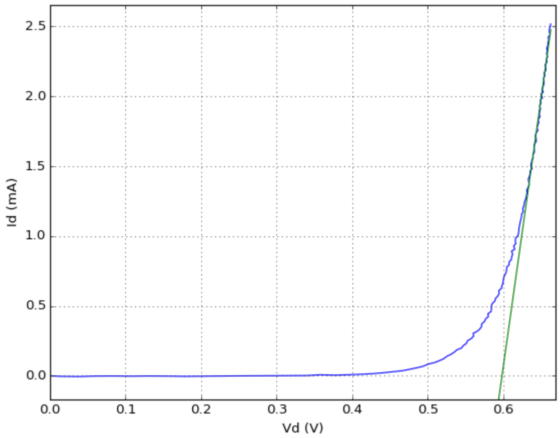
The model is calculated as follows:

① Take the circuit and obtain the rough value of the currents and voltages in its DC operating quiescent point Q. We will get a rough DC value VdQ and IdQ values at this operating point.

② Obtain the slope of the diode curve at Q.

③ Substitute the diode model by a straight line that has the obtained slope and goes through the Q point.

If you do the calculations around a quiescent point VdQ = 0.65 V, IdQ = 2 mA, you get a linear model for the diode operation that can be represented as the green line in the following figure.



You can see that the line represents quite well the diode operation as long as we are close to the quiescent point used to obtain this line. This line defines the small signal model for the diode and requires that the signal in the diode is small enough so that the difference between the line, as predicted by the model, and the real diode curve is small. In our case, calculated for IdQ = 2mA the model is quite good for all shown currents above 1 mA.

The slope can be calculated from the current at the quiescent point by performing some calculations on the diode equation:

$$slope = \frac{\partial I_d}{\partial V_d} = \frac{\partial}{\partial V_d} I_S\left(e^{V_d/\eta \cdot V_T} - 1\right) = I_S \frac{1}{\eta \cdot V_T} e^{V_d/\eta \cdot V_T}$$

If we remember, for currents much bigger than I_S we can simplify the diode equation:

$$I_d(V_d) = I_S\left(e^{V_d/\eta \cdot V_T} - 1\right) \approx I_S \cdot e^{V_d/\eta \cdot V_T}$$

So the slope can be simplified as:

$$slope = \frac{1}{\eta \cdot V_T} I_S e^{V_d/\eta \cdot V_T} \approx \frac{I_d}{\eta \cdot V_T}$$

At the quiescent point, with a IdQ current value, we get:

$$slope\ at\ I_{dQ} = \left.\frac{I_d}{\eta \cdot V_T}\right|_{I_d = I_{dQ}} = \frac{I_{dQ}}{\eta \cdot V_T} = \frac{1}{r_d}$$

The slope is defined as $1/r_d$ as it has dimensions of Ω^{-1}. We call this parameter the dynamic resistance of the diode. It is called dynamic because it depends on the chosen quiescent point.

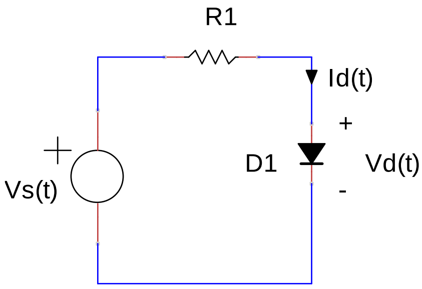
$$r_d = \frac{\eta \cdot V_T}{I_{dQ}}$$

For normal silicon diodes η is about 2 and V_T is about 26 mV at room temperature.

The small signal model has the obtained slope and passes though the Q point so it can be defined as:

$$I_d = I_{dQ} + \frac{V_d - V_{dQ}}{r_d}$$

Let's obtain the small signal diode mode for our circuit:



Remember that R1 was 1 kΩ and Vs(t) had values between 2V and 3V. We can use the mean value of Vs(t) to calculate the quiescent point Q.

$$V_{sQ} = \frac{V_{s\,max} + V_{s\,min}}{2} = 2.5V$$

3

Using the constant voltage model with Vγ = 0.7 V, to obtain the quiescent operation point of the diode defined by VdQ and IdQ.

Obtain the value of the diode dynamic resistance r_d at this point. Consider η to be equal to 2 and the typical 26 mV V_T value.

We can now check the model. First we set the values obtained above in 3 :

```
>>> IdQ =      # Fill value (in mA)
>>> VdQ = 0.7  # From constant voltage model
>>> rd =       # Fill value (in k Ohm)
```

Now we obtain the line representation as predicted by the model. We will need to use the ***arange*** function of the **numpy** package to generate the model diode voltage vdm values for the x axis.

```
>>> import numpy as np
>>> vdm = np.arange(0.6,0.8,0.01)
>>> idm = IdQ+(vdm-VdQ)/rd
```

Recall the 1N4148 real diode curve saved in the previous project to compare:

```
>>> vdr,idr=slab.load("1N4148")
```

And compare it with the obtained model:

```
>>> slab.plotnn([vdr,vdm],[idr,idm],"","Vd (V)","Id (mA)")
```

⚒	**4**	Draw the small signal model straight line against the real diode curve. Compare the slopes at the quiescent point.

You should see that the curves does not agree too well as they are displaced from each other in the horizontal position. They should agree in the slope, however, for currents around the IdQ value.

You can use the small signal model to obtain a prediction of our circuit operation. Remember that we have stored the time dependent measurements in the **t**, **vd**, **id** and **vs** variables.

The response of the circuit when using the small signal model can be calculated from the sine source voltage vs as measured in ⚒2 :

```
>>> id2=(vs-VdQ)/(1+rd)
>>> vd2=VdQ+rd*(id-IdQ)
```

And it can be compared with the measurements both for Vd and Id variables:

```
>>> slab.plot1n(t,[id,id2],"","time (s)","Id (mA)")
>>> slab.plot1n(t,[vd,vd2],"","time (s)","Vd (V)")
```

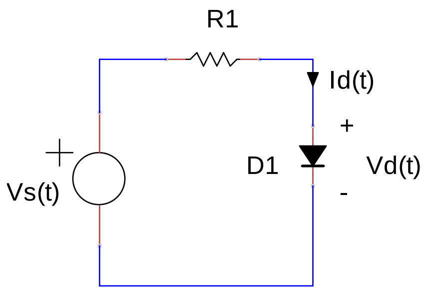
You should obtain a good agreement in the currents and a not so good agreement on the voltages. The voltages, however, should show a constant displacement. If you compute the amplitudes they should match quite well:

```
>>> slab.peak2peak(vd)
>>> slab.peak2peak(vd2)
```

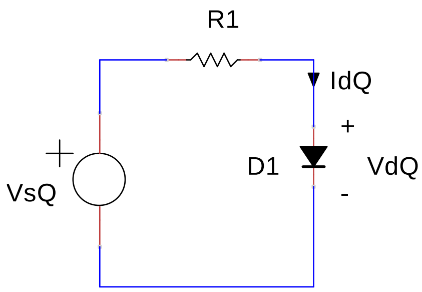
	5	Obtain the circuit response when using the small signal model. Compare the response, as predicted by the model, with the circuit measurements.

Incremental circuit

To resume the previous operations, we had a circuit we wanted to solve:



We obtained the quiescent estate for the circuit using the simple **constant voltage diode model**:



We obtained a **small signal model** for the diode that agrees quite well with the diode curve for points in the diode curve that are not far from the quiescent point:

$$I_d = I_{dQ} + \frac{V_d - V_{dQ}}{r_d}$$

Using this model we calculated the response of the original circuit.

We can benefit from the fact that the small signal model is linear to apply the superposition concept. We can define incremental circuit variables, like Δx, as the deviation of the circuit variables, like x, from the quiescent point, like X_Q.

$$\Delta x = x - X_Q$$

By definition, the quiescent circuit state is defined in DC and is not time dependent, so if we have time dependent variables on the circuit they will necessary have time dependent incremental values:

$$\Delta x(t) = x(t) - X_Q$$

In particular, in the diode, we can define the following incremental variables:

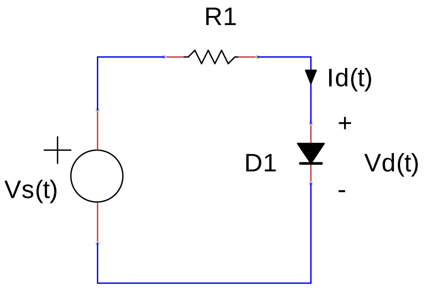
$$\Delta v_d = v_d - V_{dQ}$$
$$\Delta i_d = i_d - I_{dQ}$$

Another way to see the incremental behavior of the circuit is to understand that any circuit variable can be computed by adding its incremental value over the quiescent value:

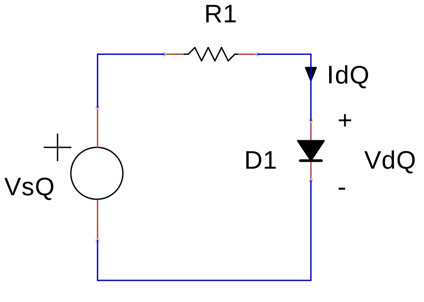
$$v_d = V_{dQ} + \Delta v_d$$
$$i_d = I_{dQ} + \Delta i_d$$

If the circuit is linear, we can solve it by superposition of two circuit solutions. One solution is the circuit quiescent state and the other one is its incremental operation. As the small signal model for the diode is linear, the linear superposition property can be applied.

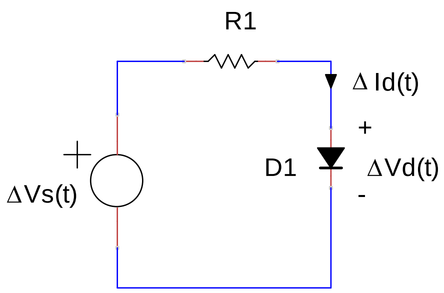
In order to obtain the incremental variables we can use an **incremental circuit**. This is a circuit where all variables are incremental. For instance, given the circuit:



We already know that we can obtain its quiescent state using, for instance, a constant voltage diode model:



We can also define an incremental circuit as the circuit where all variables are incremental:



In the case of the source voltage, the incremental voltage is:

$$\Delta V_S(t) = V_S(t) - V_{SQ}$$

In the case of the resistor, as it is a linear component, it provides the same relationship between quiescent and incremental values. To demonstrate that statement, the relationship established by the resistor is:

$$V_R = R \cdot I_R$$

The relationship in quiescent state is:

$$V_{RQ} = R \cdot I_{RQ}$$

The incremental variable Δv_R can be obtained then as:

$$\Delta v_R = V_R - V_{RQ} = R \cdot I_R - R \cdot I_{RQ} = R\left(I_R - I_{RQ}\right) = R \cdot \Delta i_R$$

So, the ohm's law applies both to quiescent and incremental variables.

In the case of the diode, the small signal model is:

$$I_d = I_{dQ} + \frac{V_d - V_{dQ}}{r_d}$$

So, the incremental relation is:

$$\Delta i_d = I_d - I_{dQ} = I_{dQ} + \frac{V_d - V_{dQ}}{r_d} - I_{dQ} = \frac{V_d - V_{dQ}}{r_d}$$

But remember that:

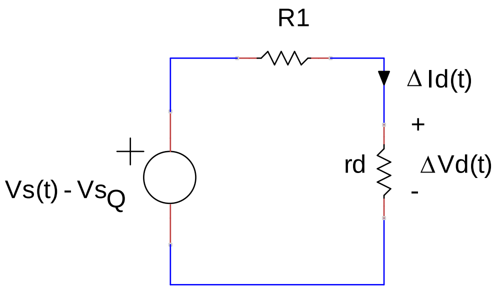
$$\Delta v_d = v_d - V_{dQ}$$

So:

$$\Delta i_d = \frac{V_d - V_{dQ}}{r_d} = \frac{\Delta v_d}{r_d}$$

The relationship that the diode establishes between its incremental variables is the same as the one defined by an r_d resistor.

Joining all the above relations, our incremental circuit is:



Let's try that with the circuit we have measured. Remember that R1 = 1kΩ, Vs(t) is a sine wave with minimum at 2V and maximum at 3V, and VsQ is 2.5V.

6	Solve the above circuit. In particular, obtain the maximum and minimum values of Δid and Δvd. Use the r_d value calculated at 3

Remember that any time dependence is incremental by definition so the amplitude of the Δid and Δvd shall be the same of the variables id and vd.

7	Compare the amplitudes calculated on 6 with the ones measured in 2.

Once we have the incremental variables we can obtain the full circuit variables superposing them on the quiescent circuit state:

$$v_d = V_{dQ} + \Delta v_d$$
$$i_d = I_{dQ} + \Delta i_d$$

Small signal method

To summarize the above explanations this section indicates the needed steps to analyze a circuit that includes one or several diodes using the small signal model.

① Obtain the quiescent state of the circuit. In particular it is vital to obtain the quiescent current on all diodes. In order to solve the circuit we will use the **constant voltage model** for the diodes. Remember that we need to make hypothesis on the combination of states of all diodes.
In the quiescent circuit all constant voltage and current sources will have its own value and time dependent sources will have its mean value.

② Calculate the dynamic resistance for all the diodes. Diodes in OFF state have an infinite dynamic resistance (open circuit). Diodes in ON state have a dynamic resistance defined by:

$$r_d = \frac{\eta \cdot V_T}{I_{dQ}}$$

③ Obtain the incremental circuit. This circuit is obtained substituting all components by their incremental equivalent:

- For voltage or current sources they have their value with the quiescent value subtracted. That means that constant sources are zero in the incremental circuit.

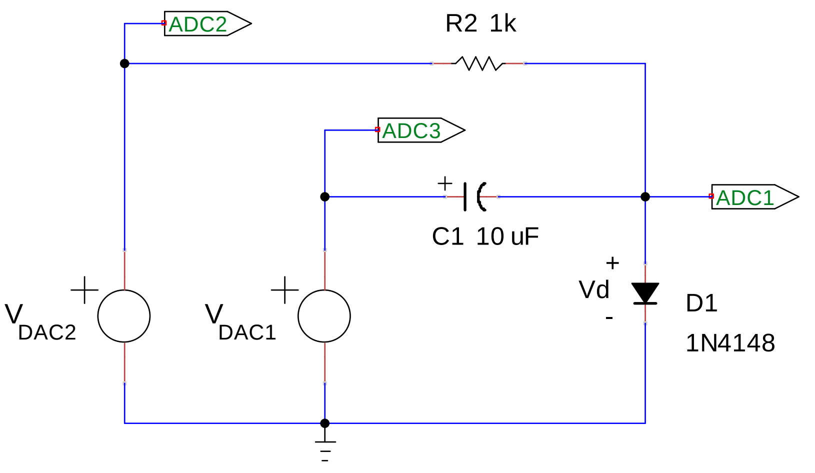
- For linear components, like resistors, they do not change.

- For diodes, they are substituted by their dynamic resistances.

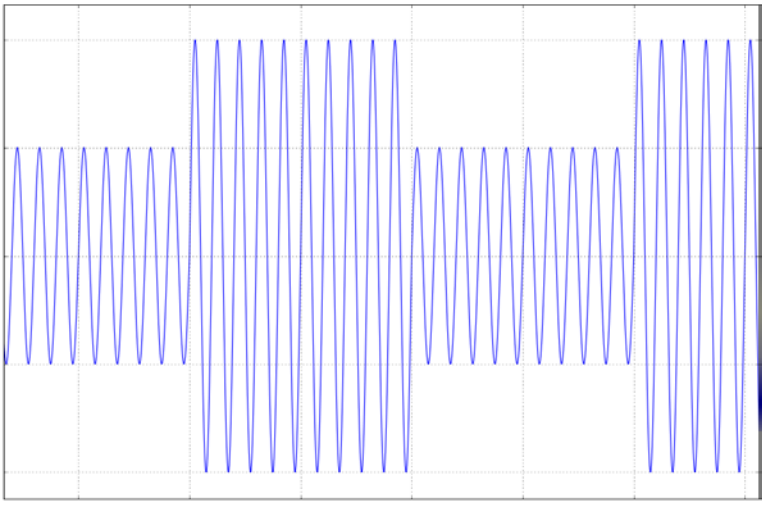
④ Solve the incremental circuit to obtain the incremental variables.
The amplitude of any time dependent signal can always be obtained from the incremental variables.
If you need to obtain a global circuit variable, you can obtain it using superposition:

$$x = X_Q + \Delta x$$

To end this document, we will analyze the diode modulator shown on the following figure.



A modulator is a circuit that modulates a carrier signal, usually a constant frequency sine wave, with another lower frequency time variant signal that carries some information. In our case the circuit performs amplitude modulation. It will modulate a carrier 200 Hz sine wave with a 2 Hz square wave. The following example shows one example of a sine carrier modulated in amplitude with a square wave although the frequency ratio is 1 to 10 not 1 to 100 like in our case. If it were represented with a 1 to 100 ratio it wouldn't be possible to scale the image so that both the carrier and the modulating signals are easy recognizable.



The circuit uses the non linearity of the diode to modulate the carrier generated on DAC 1 with the square signal generated on DAC 2.

The DAC 1 output generates a 200 Hz sine wave with minimum at 2 V and maximum at 2.5 V so the peak to peak voltage is 0.5 V and the amplitude 0.25 V.

The DAC 2 output generates a square wave. That means that it alternates two output values that in our case will be 2V and 3V. For each of those two values, we have a different quiescent point for the diode.

The capacitor impedance is:

$$Z_{C1} = \frac{1}{j \cdot 2\pi f \cdot C1}$$

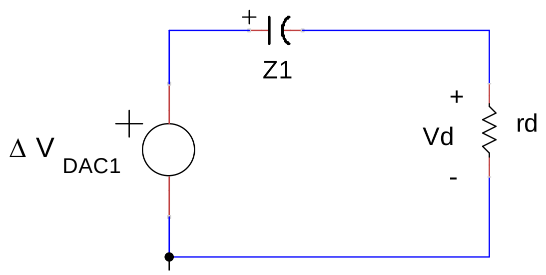
At the carrier 200 Hz frequency it is quite low, but at the 2 Hz modulating signal it is high enough for the capacitor to be modeled as an open circuit.

	8	Obtain the diode dynamic resistance for the two considered DAC 2 voltage cases cases: $r_{d\,2V}$ and $r_{d\,3V}$. Consider a constant voltage silicon diode model with $V\gamma$ = 0.7V and suppose that C1 current is zero for those calculations.

Once we have the small signal model for the diode for the two cases, we can obtain the incremental circuit. In the incremental circuit, D1 is substituted by its dynamic resistance r_d. the DAC 2 source voltage is zero. That means that R2 is parallel with r_d. As r_d is always much lower, R2 can be neglected. DAC 1 source is substituted by its incremental value:

$$\Delta V_{DAC\,1} = V_{DAC\,1} - \overline{V_{DAC\,1}} = 0{,}25 \cdot sin(2\pi f \cdot t)$$

Capacitor C1 behaves as its impedance at 200 Hz.



We can now obtain the Vd amplitude for the two rd values that depend on the modulating signal value.

	9	Determine Z_{C1} for the 200 Hz carrier frequency. Obtain the amplitude of Vd for the two cases $r_{d\,2V}$ and $r_{d\,3V}$. Remember that Z1 impedance is imaginary.

We can now check the calculations against the measurements.

Mount the circuit and generate the two DAC outputs. Observe that C1 has polarity; don't connect it in the wrong way. We will only use 20 points for the carrier in order to have enough storage space for two periods of the modulating 2Hz signal.

```
>>> slab.waveSine(2,2.5,20)
>>> slab.setWaveFrequency(200)
>>> slab.waveSquare(2,3,20000,second=True)
```

Now, define a measurement for two full modulating square waves storing information for ADCs 1, 2 and 3 and perform the measurement:

```
>>> slab.tranStore(4000,3)
>>> slab.wavePlot(dual=True)
```

🛠	**10**	Perform the measurements and compare the Vd amplitudes, measured at ADC 1 with the calculations performed on 9. Do they agree?

You can expect some discrepancies between the calculations and the measurements. Absolute amplitudes can be off up to about a factor of two although relative amplitudes relationship should agree better.

The small signal method uses a rough model of the diode around a quiescent point and we have added also some simplifications to the calculations like neglecting R2 in the incremental circuit and considering C1 open circuit in the rd calculations. Moreover, the components, especially capacitor C1 have some tolerance on its real value respect to its nominal value. So, we cannot expect exact results from the calculations.

The main importance of the calculations is that they give us a good sense of how the circuit operates. This is something that is quite difficult to grasp using a simulator or only performing measurements.

Finding the error sources

If you are curious, you can perform other measurements, like obtaining the real current on R2 or measuring the real C1 reactance at 200 Hz. That way you can detect the main sources of error on the calculations.
Performing enough measurements, all error sources can be located. It's up to you.

Last comments

This project has shown the small signal calculation method for the diode. This method is crucial to understand the operation of other non linear devices like the bipolar and field effect transistors.

In the last exercise we have calculated the response of a diode modulator that takes profit of the non linear diode behavior to modulate a carrier signal with a square wave. Building modulators is, in fact, a typical RF application for diodes. Although the measurements would not perfectly agree with the calculations, being able to roughly predict the operation of the circuit gives us a great insight on how it is working.

The **constant voltage** diode model and the **small signal** diode models are the two main models we will use later for hand calculation of most diode circuits. There will be, however, some special cases where both models are not enough and we will need to use the full exponential model. That should serve to remind us that an important part in engineering is being able to select the model best suited to a problem. This is the model that most eases the calculations while also adds a reasonable error to the values it predict.

References

SLab Python References

Those are the reference documents for the SLab Python modules. They describe the commands that can be carried out after importing each module.
They should be available in the **SLab/Doc** folder.

TinyCad

Circuit images on this document have been drawn using the free software TinyCad
https://sourceforge.net/projects/tinycad/

SciPy

All the functions plots have been generated using the Matplotlib SciPy package.
https://www.scipy.org/

LTSpice

Circuit simulator provided for free from Linear Technology. You can use the simulator to obtain numerical solutions of complex circuits.
http://www.linear.com/designtools/software/