

# Circuit Analyses. Laboration 2 measurements and simulations on DC-circuits ( Thevenin-equivalent )

This booklet, signed by the teacher, serves as a receipt for passing the lab. Each student must have a booklet of his own with solid preparation and completed readings.

Name:

Confirmed (Teacher):

Objective In this lab. You will will perform measurements and simulations of DC networks in order to gain a better understanding of voltage dividers and Thevenin equivalents.

## Equipment

- Digital multimeter (DMM) Fluke45
- Power supply
- RCL meter
- Breadboard and Components

**Literature** Install the program **Orcad Lite** on your own computer (or **LTSpice**). Go through the presentation "PSpice intro". Learn how the program will be used on school computers.

**Report** Write down your measured values, and figures, in this booklet and report to the teacher during the laboration.

## Preparation tasks F1...F3 overview

### F1: Connection of the Volt-Ampere meter

Draw a picture of how the voltage and ammeter should be connected to measure voltage and current with as little error as possible in the measurement tasks M1 and M2.

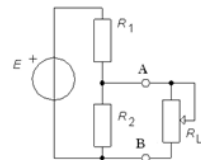
(Note that it is the two-port that is the object of measurement, it is not the  $R_L$  that is to be measured.)

### F2: Voltage divider with load

Set up an equation for  $I(U)$ , containing the constants  $E$ ,  $R_1$  and  $R_2$ , and the variable  $U$ , when the voltage divider in L1 has a load that varies.  $R_L$  shall *not be included* in the expression. Plot the current  $I$  as a function of the voltage  $U$  in Graph 1 when  $R_L$  is varied from 0 to  $\infty$ .

Components:  $E = 2V$ ,  $R_1 = 330 \Omega$ ,  $R_2 = 470 \Omega$

Symboler:  $\text{V}$   $\text{A}$



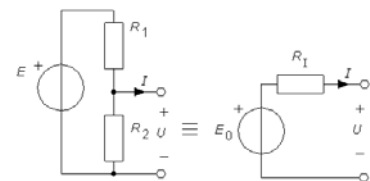
### F3: Thevenin equivalent

Calculate the Thevenin equivalent  $E_0$ ,  $R_1$  to the network in M1.

Set up an equation for  $I(U)$ , containing the constants  $E$ ,  $R_1$  and  $R_2$ , and the variable  $U$ , when the thevenin equivalent has a load that varies.

Compare the equation with that one from F2.

Plot the current  $I$  as a function of the voltage  $U$  in Graph 1 when  $R_L$  is varied from 0 to  $\infty$ .



# Measurements, report of preparation tasks

## M0: Measure the resistors.

Check with an ohmmeter the values of the resistors  $R_1$ ,  $R_2$ ,  $R_1$ , and measure  $R_1 // R_2$ .

You will use the values from this measurement to later explain any discrepancies between the measured and calculated curves in M1 or M2.

M0:  $R_1 =$   [Ω]  $R_2 =$   [Ω]  $R_1 // R_2 =$   [Ω]

## F1: V Connection of the Volt-Ampere meter

### M1: I-U-characteristic for voltage divider with load.

Draw a picture of how the voltage and ammeter should be connected to measure voltage and current with as little error as possible in the measurement tasks M1 and M2.

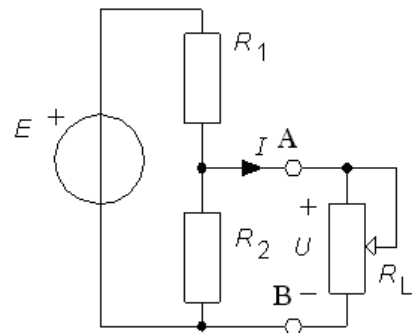
(Note that it is the two-port that is the object of measurement, it is not the  $R_L$  that is to be measured.)

The voltage divider in the figure

The adjacent voltage divider to be set up and tested.

Between the voltage divider outputs A and B, a variable resistance ( $R_L = 1k\Omega$  potentiometer) is to be connected.

With increasing load (load current from the voltage divider) the output voltage will drop. Component values are



$E = 2V$ ,  $R_1 = 330 \Omega$ ,  $R_2 = 470 \Omega$

## F2: Voltage divider with load

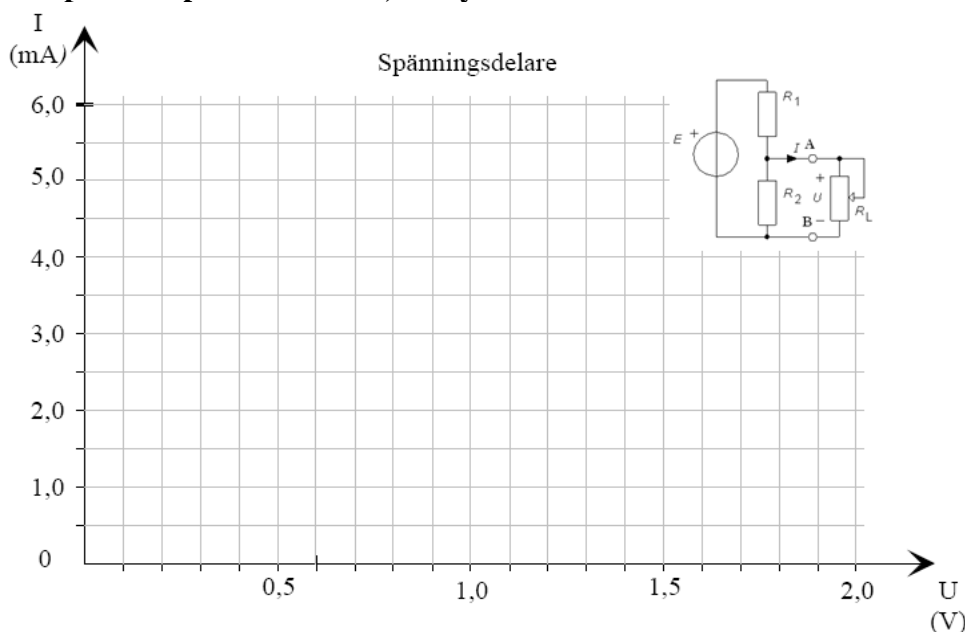
Set up an equation for  $I(U)$ , containing the constants  $E$ ,  $R_1$  and  $R_2$ , and the variable  $U$ , when the voltage divider in L1 has a load that varies.  $R_L$  shall *not be included* in the expression.

$I(U) =$

Preparation Curve: Plot the current  $I$  as a function of the voltage  $U$  when  $R_L$  is varied from 0 to  $\infty$  with the given component values.

## F2, M1:

### Graph 1. Preparation Curve, and your measured values / simulated values



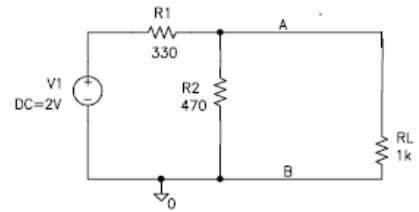
Measured values:

U [V]	I [mA]

Plot values in graph

**Simulation.**

Simulate with Pspice how current  $I$  varies as a function of  $U$  when you change the value of  $R_L$  for the voltage divider circuit. Pspice only need to calculate the Bias Point.



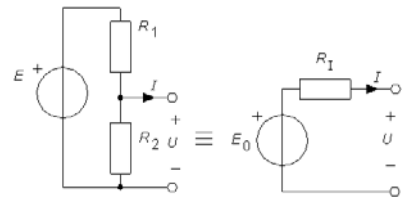
Plot  $I$  as a function of  $U$  in Graph1 with the values you get from the simulation.

**Automate the simulation.**

- A cleverer way to simulate the characteristics of the voltage divider is to put a DC-source instead of  $R_L$  and do a DC sweep on from 0 V to  $E_0$ . (See tutorial)
- Another clever way is to do a parametric sweep with  $R_L$ . (See tutorial)

**F3: Thevenin equivalent**

Calculate the Thevenin equivalent  $E_0, R_1$  to the network in M1. Set up an equation for  $I(U)$ , containing the constants  $E, R_1$  and  $R_2$ , and the variable  $U$ , when the Thevenin equivalent has a load that varies.



Compare the equation with that one from F2.

$I(U) =$

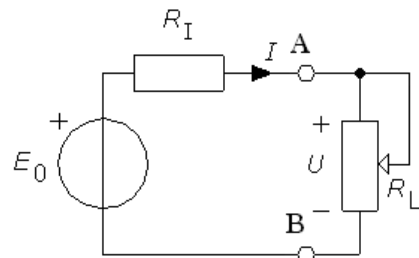
Preparation Curve: Plot the current  $I$  as a function of the voltage  $U$  in Graph 2 when  $R_L$  is varied from 0 to  $\infty$ .

**M2: Measurements with the Thevenin equivalent**

Switch the network to the Thevenin equivalent to the circuit in M1. Set the voltage source to the value of  $E_0$ .

How to "arrange" the value  $R_1$ ?

Draw how to connect voltmeter and ammeter in the figure.



Repeat the corresponding simulations and measurements of M1 for the Thevenin equivalent. Plot values in Graph 2.

Also draw a curve of the power in the load  $P_L (U \cdot I)$  as a function of  $R_L (U/I)$  in Graph 3.

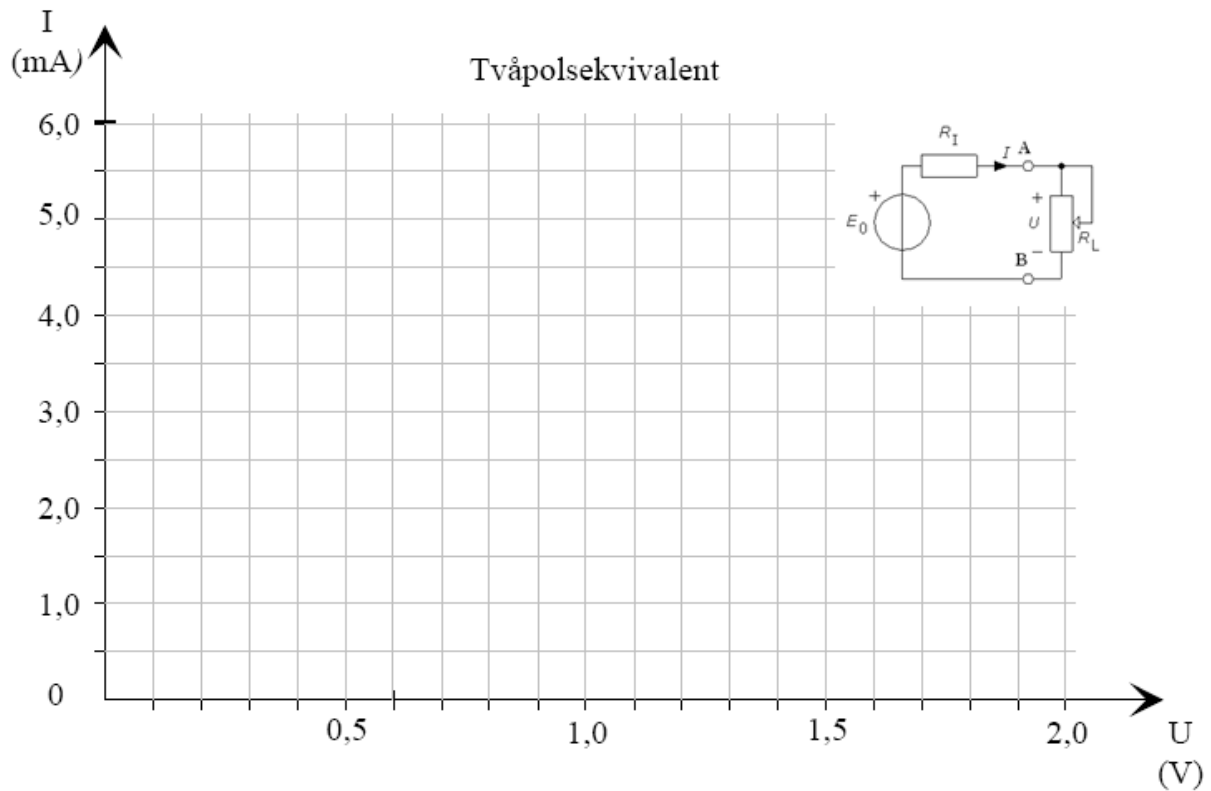
- At which  $R_L$  has the load power its maximum, and how much power is this max value?
- Compare with the simulation curve.

Keep all connections for M3.

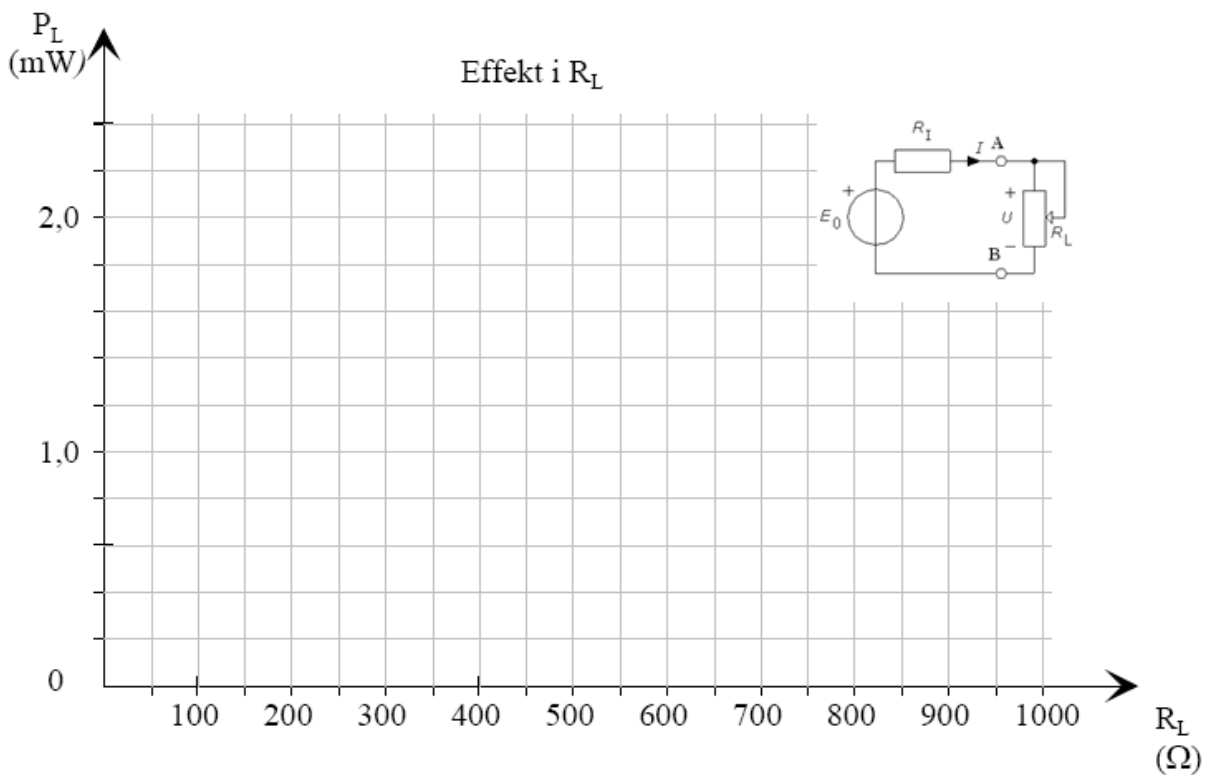
With Fluke45 both  $U$  and  $I$  are measured and displayed simultaneously because it has two displays. You should also calculate  $R$  and  $P$  from the readings. The table is prepared for nine measurements.

U[V]								
I [mA]								
R[Ω]=U/I								
P[mW]=U·I								

**F3, M2: Graph2. Preparation Curve , and your measured values / simulated values**



**M2: Graph3. Calculated values from measurements, (compare with the simulation)**

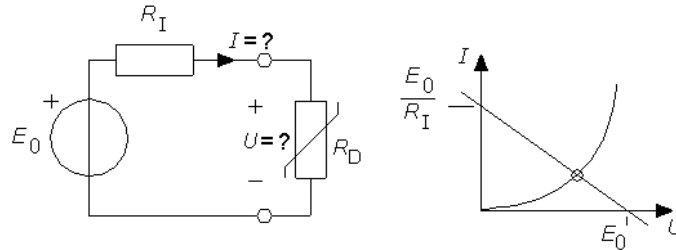
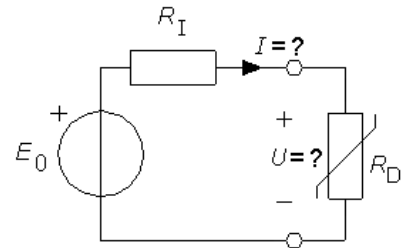


Max power  $P_{Lmax} =$   [mW]      At  $R_L =$   [Ω]

Your comment:

**M3: Theory. Nonlinear component (diode) connected to the two-port.**

The laws of circuit analyses only apply to circuits consisting of linear components. If we connect an ordinary linear resistor  $R_I$  with a nonlinear component  $R_D$ , we can not simply calculate the substitution resistor as  $R_{ERS} = R_I + R_D$ . The nonlinear resistor is influenced by the current through it, which in turn is determined by the entire circuits substitution resistance – and that is what we tried to calculate!

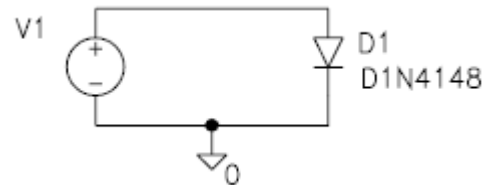


The resultant resistance can however be determined graphically. The I-U-characteristic of the two-port is plotted together with the I-U-characteristic of the non-linear component in the same graph. The two components have only one point, intersection, in common. Since the same current flows in the entire circuit, the  $I$  (and therefore  $U$ ) have those values that correspond to the intersection of the curves. This is the circuit's **operating point**.

**Simulate the diode.**

As an example of a nonlinear component shall we connect a diode, 1N4148, to the network in M2. A diode is a component which, in principle only can conduct current in one direction (the forward direction) and blocks current in the other direction (the reverse direction). Depending on how you connect it, it can conduct or not conduct electricity. In the forward direction is the I-U-characteristic is an exponential function.

Draw this diagram in Schematics (D1N4148 from library eval). Make a DC sweep from -5V to +1 V and you'll see that the current is close to zero in the reverse direction and exponential in the forward direction.



Then make a DC-sweep from 0V to 0.75V with step 0:01 V. Plot with Probe the curve of the diode current as a function of the diode voltage.

**The Diode together with the equation for the two-port in Probe.**

Insert the equation that you derived in F3 together with the diode characteristic in Probe. Use Trace/Add and insert the numerical values for E, R1 and R2. The equation should contain the swept voltage ( $V_{V1}$ ) as the x-variable .

Note the voltage and current at the intersection. (use the cursor to read exact values).

**Measurement. Connect the diode to the two-port circuit.**

Connect the diode with the two-port circuit from M2 with the diode on the breadboard and measure voltage  $U$  and current  $I$  for the diode.

Operating point of diode	U [V]	I [mA]
Simulation		
Measure		

**( Simulation. Two-port with diode. )**

Pspice can of course solve the problem of "two-port and diode" directly. Simulate the two-port from M2 connected to the diode (instead of  $R_L$ ), run operational point, and read the values of current  $I$  and voltage  $U$  . Compare the simulation with your measurement.

**You could use this page for your calculations.**