Polish-Japanese Institute of
Information Technology

ELECTRONIC LABORATORY

WARSZAWA 2003

Table of contents

Michał Ramotowski

ELECTRONIC LABORATORY

Instruction sheet No. 1

MEASUREMENTS of VOLTAGES and CURRENTS

(The text is freely based on the book:
'Laboratorium Podstaw Elektroniki, Materiały Pomocnicze, Część 1',
with the kind permission of the author
Bogusław Kalinowski)

Experiment I

MEASUREMENTS of VOLTAGES and CURRENTS

Goals of the experiment:
- introduction to measuring equipment used in Electronics Laboratory,
- introduction to methods of voltage and current measurement,
- observation of measurement errors caused by connection of measuring device to measured circuit.

Scope of the experiment

1. Introduction to the experiment including presentation of measuring equipment used in the Laboratory.

2. DC voltage measurements
2. 1. Measurement the output voltage of the power supply with a multimeter
2. 2. Measurement the output voltage of the power supply with an oscilloscope
2. 3. Measurement the output voltages of the high and low resistance voltage dividers, problem of loading measured circuit with the measuring device;

3. DC current measurements
3. 1. Measurement of power supply output current with the multimeter
3. 2. Measurement the DC current by reading the voltage drop on known resistor, the ammeter internal resistance problem

4. AC voltage measurement
4. 1. Measurement of the sinusoidal signal
- measurement of frequency
- measurement of peak-to-peak value

4. 2. Measurement of the rectangular signal
- measurement of frequency
- measurement of peak-to-peak value
- measurement of duty cycle.

4. 3. Measurement of the triangular signal
- measurement of frequency
- measurement of peak-to-peak value
- measurement of the voltage change speed

Equipment on the laboratory bench

M-1 - laboratory module for voltage and current dividers with resistors;
MX-620 (or other models) - universal digital meter;
OS-9040 (or HUNG CHANG) - 2 channel oscilloscope;
MX-9000 (or MX-9300) - multifunction device;
Connection cables: BNC-BNC - 2 pcs, BNC-banana - 1 pcs, banana-banana -2 pcs.

1. MEASURING INSTRUMENTS USED IN ELECTRONIC LABORATORY

The instruments listed will also be used in next laboratory experiments. Short descriptions of these devices are given below.

1.1. Multifunction device MX-9000 (MX-9300)

The multifunction device MX-9300 differs from model 9000 in that it has an additional frequency counter range. The front panel of the instrument is shown in Fig.1.1.

On the left side of the instrument are placed three power supplies: 5V/2A; 15V/1A and 0-50V/0-0, 5A. Two from them have their output voltage and the output current limit fixed - in the third unit both values of output voltage and current limit are adjustable. The current limiters have their output characteristics of fold-back type, so in case of overloading they have to be switched off and again on to put them back into operation.

In central, upper part of the instrument the digital frequency meter is placed. Its basic parameters are following:

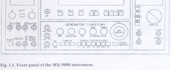


Fig. 1.1. Front panel of the MX-9000 instrument.

- measuring ranges: 10 MHz, 100 MHz
- input impedance: l MΩ, 100 pF
- maximum sensitivity: 15 mV(RMS value, direct input, immediate, from l Hz to 60 MHz)
- maximum input voltage: 15 V(RMS value, direct entry)
- input divider: 1: 10
- time of gate opening: 10 ms, 100 ms, l s, 10 s
- display: LED, 8 digits, indicators: offenses of range, open gate, kHz, MHz
- function 'HOLD'": freezing results on the display
- function 'AUTO'": forcing measurement range 100 MHz and gate opening time l s.

Using digital frequency meter is easy. According to the measured frequency of signal the *frequency range* is selected, and depending on required resolution - the *gate opening time*. For signals greater than 14V RMS the input signal divider should be used. **Direct measurement of power line frequency may damage the frequency meter.**

Below the frequency meter the function generator block is placed. On the left side of the block there are three BNC sockets: the lowest one is the signal output, the middle socket provides the signal in the TTL standard and the upmost socket is an input of modulating signal. The peak-to-peak output signal value w/o load is max. 20 V and falls down to its half value if the nominal load is used (50 or 600Ω, selectable).
The TTL output can be loaded with 1.2 mA in HI state and 24 mA in LO state.
The voltage at the external modulation input may change from about 0 V to +10 V. Such a change produces a hundred-fold change of signal frequency, under condition, that the frequency control knob is left in full CCW position (control marker points number 2 on the scale).

On the right side of the signal socket the output resistance selector is placed. Pressing these switch selects 600Ω, depressing selects 50Ω. Next switch to the right, marked as 'FUNCTION' selects the shape of the output signal - sine, square, or triangle wave. Above the function switch are signal amplitude and its DC offset controls. Amplitude control range equals 20 dB (ten times), pulling the knob adds another -20 dB. Pulling out its knob activates the DC offset control. The DC offset can be set in the range from -10 V to +10 V (pressing knob fixes DC offset at zero).

Next to the function switch is 7-position frequency range switch, whose keys are marked as frequency range multipliers: 1, 10, 100, 1k, 10k, 100k, 1M ($k=10^3$, $M=10^6$). They multiply the basic frequency range which extends from 0.02 to 2 Hz. So the output signal frequency may be set to any value from 0.02 Hz to 2 MHz.

Above the frequency range switch a set of knobs is placed to control the output signal symmetry, the amplitude of automatic frequency modulation and the frequency modulation period. Again, to activate the symmetry control pull out the knob. It works in such a way that voltage change speed rises on one signal edge and decreases on the second one for sinusoidal and triangular signals. For rectangular signal the duty-cycle is changed. The range of regulation: from 10: 1 to 1: 10.

Internal automatic frequency modulator starts by pulling the deviation control. The relation between the maximum and minimum values of the frequency during one period of modulation may reach 100. It corresponds to situation when the deviation control is set fully CW and the frequency control fully CCW. Automatic modulation can be realized in two different ways: with linear or logarithmic frequency change. The choice is done by pulling or pushing the period control knob: push for linear, pull for logarithmic change. Turning the knob causes change of the generator tuning speed that is the period of tuning signal changes over the range from 20 ms to 2 s.

On the right side of the function generator block the continuous frequency control knob is placed and, just under this knob, frequency measurement switch. On the frequency control knob the basic frequency range scale is marked from 0.02 to 2. Using this knob and pressing the suitable range key the output signal frequency is set. Such a setting is not very precise, so the signal frequency can be measured by means of frequency meter placed above. To measure the signal frequency the 10 MHz range and adequate gate time of the frequency meter should be selected and the switch placed just under the frequency control knob subsequently pressed.
The measured frequency is shown on the frequency meter display.

Note: If the generator output resistance is set to 600Ω the signal frequency can be changed only over the range from 0, 02 Hz to 100 kHz.

The last right side block of the instrument MX-9000, is the 3 1/2 digit automatic multirange multimeter. Its measuring possibilities are listed below.
- resistance in ranges: 200 Ω, ..., 2 MΩ
- DC voltages in ranges: 200 mV,.., 200 V, 1000 V

- AC voltages in ranges: 200 mV,.., 200 V, 750 V
- DC current in ranges: 200 mA, 10 A
- AC current in ranges: 200 mA, 10 A.

Two sockets placed in the bottom right part of the block are used to connect measured voltages and resistances. Remaining two sockets, being found in bottom left part of the block, serve to: lower one for values up to 200 mA, and upper one for values up to 10 A. The second socket used for measurement of the current is marked 'COMMON'.

Selection of measured value type may be done with the switch 'FUNCTION'. One may choose: resistance, voltages, current up to 200 mA and current up to 10 A.

On the left side of the knob 'FUNCTION' there are four buttons. The uppermost one selects the type of measured voltage or current DC or AC (constant or alternate) or value of the current forced in the external circuit when resistance is to be measured. ('Ω'- small value of the current, 'LO Ω' - large value of the current The second switch from top, marked with symbol ' -MEM', memorizes value shown on least significant position of the result of measurement and subtracts remembered value from results of next measurements. The third switch, marked as 'RANGE', switches off automatic range selection in the multimeter, then the range may be selected manually. Last switch, marked as 'HOLD', freezes measured result (with an exception of current measurements on the range of 10 A).

1.2. Oscilloscope OS-9040

The front panel of the instrument is shown in Fig.1.2.

The instrument is a 2-channel oscilloscope featuring simultaneous observation and measurements of two signals within frequency bandwidth up to 40 MHz. On the front panel of the instrument one may distinguish separate functional areas. The left side area is occupied mainly with the CRT (cathode ray tube) screen. On the inner surface of the screen the special scale, formed with horizontal and vertical lines, is placed. The scale has 10 divisions horizontally and 8 divisions vertically forming squares with side about l cm. The divisions on central lines of the scale are divided in 5 smaller sections, about 2 mm each. The scale is used for measurement various parameters (voltages, currents, frequency, shapes, etc.) of observed electric signals.

To the right of the screen, in a narrow vertical section, there are placed: main line switch, brightness and focus controls, scale graticule control and special square wave signal ($f = 1$ kHz, $V_{p\text{-}p} = 0.5$ V) used for adjustment of oscilloscope probes.

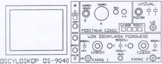


Fig. 1.2. Front panel of the OS-9040 oscilloscope

Following functional section (right bottom side) includes all controls of the vertical deflection block of the oscilloscope. Here are:
- input sockets for both channels,
- slide switches of input modes (DC, AC - , types of coupling the input signal to the input of the vertical amplifier, the third position 'GND' grounds the vertical amplifier input),
- vertical sensitivity (step and variable) and position controls,
- slide switch for vertical mode (selects displaying one signal, both signals, sum of signals), the signal from channel 2 can be inverted, what makes possible to display the difference of both channels.

There is also possible to display the signal in X-Y mode (one signal versus the other one). In such a case signal X is connected to the channel 1, and signal Y to the channel 2.

Here most important parameters of the vertical deflection block:
- input impedance: 1 MΩ, 25 pF
- maximum input voltage: 250 V
- sensitivity ranges:
    - a) standard: from 5 mV/div to 5 V/div in 1, 2, 5 sequence
    - b) increased: from 1 mV/div to 1 V/div in 1, 2, 5 sequence
- bandwidth: 0 - 40 MHz (up to 7 MHz with increased sensitivity)

Above the vertical deflection block are found:
a) four-button time-base switch,
b) horizontal deflection (time-base) block controls,
c) time-base trigger block controls.

In the oscilloscope OS-9040 there are two time-base circuits: normal, named as 'A' and delayed, named as 'B'. Normal time-base covers the beam speed in range from 0.2 μs/div to 0.2 s/div in steps selected in sequence 1, 2, 5. If needed, variable time-base speed control may be used, the speed is then uncalibrated. Possible is tenfold increase of the time-base A speed (typically marked '10x magnifier'). The second time-base speed can also be steeply changed in range from

0.2 µs/div to 20 µs/div, in sequence 1, 2, 5. All display can be moved left or right using the knob marked with horizontal arrows.

In the trigger block area there are few more controls:

- slide switch for selecting a source of trigger signal (it can be internal signal from channel 1, from channel 2, from power line or external signal connected to the trigger socket,
- slide switch of trigger mode (auto, normal, TV-V, TV-H),
- double-knob trigger level and HOLD-OFF control (the last one sometimes improves the trigger action).

The time-base mode buttons give choice of work with single time-base (A or B) or time-base A with delayed B (the part of signal displayed with time-base B is brighter.

Note! An effective use of the oscilloscope looks somehow mysterious for novices and needs some practice. However, it is a very basic measuring device in hands of every electronic engineer or computer hardware expert. Some laboratory benches can be equipped with oscilloscopes of other types than that described above. But they are used for measurements in very similar way.

1.3. Multimeter M-4650B

The front panel of the instrument is shown in Fig.1.3. This is a portable meter designed for measurements specified below:

- DC voltage in ranges:
  200 mV, 2 V, 20 V, 200 V, 1000 V
- AC voltage in ranges:
  200 mV, 2 V, 20 V, 200 V, 750 V
- DC current in ranges:
  200 µA, 2 mA, 200 mA, 20 A
- AC current in ranges:
  2 mA, 200 mA, 20 A
- resistance in ranges:
  200 Ω,..., 2MΩ
- frequency in ranges:
  20 kHz, 200 kHz
- capacitors in ranges:
  2 nF, 200 nF, 20 µF
- h_{FE} (beta$_{DC}$) in range:
  0 ... 1000 A/A

A few more functions are available: testing semiconductor diodes (or bipolar or FET devices), detecting shorts in circuits(signaled with a buzzer), freezing measurement result on the display (button 'H').

During laboratory work some additional information concerning the instrument may be needed.

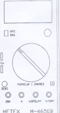


Fig.1.3. Front panel of the M-4650B multimeter

Here it is:

- voltmeter input resistance: 10 MΩ,

- voltage drop on internal resistance of the ammeter:	200 mV,
- frequency range of AC voltage and current measurements:	40-400 Hz,
- measuring voltage of the ohmmeter (EMF):	≤ 1.2 V,
- measuring voltage at diode tests (EMF):	2, 8 V,
- short resistance starting buzzer:	≤ 30 Ω,
- base current at measurement of h_{FE}:	about 10 μA,
- V_{CE} voltage at measurement of h_{FE}:	about 2.8 V.

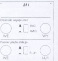


Fig.1.4. Front panel of the M-1 module

1.4. Measuring module M-1

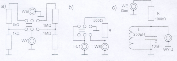


Fig.1.5. Schematic diagrams of circuits in the module M-1

The measuring module M-1 contains two resistor voltage dividers - one with 1kΩ resistors and one with 1MΩ resistors, single 560Ω resistor for current measurement experiment part and an alternating voltage attenuator with a resonant circuit. The layout of sockets and switches in the front panel of the M-1 module is shown in Fig.1.4, and the schematic diagrams of these circuits are shown in Fig.1.5.

According to the diagram in Fig.1.5, the module is designed for assembling and testing voltage attenuators. Connections to the module are possible only with BNC-BNC coaxial cables.

2. MEASURING METHODS USED IN EXPERIMENT I

2.1. Measurement of DC voltage

Measurement methods.

Two methods of DC voltage measurement are practically applicable:

- measurement with the digital voltmeter, of stand alone type, or, what takes place more often, of being a part of digital multimeter,
- measurement with analog or digital oscilloscope.

The measurement procedure by means of a voltmeter being a part of the digital multimeter (do not forget to switch it on), runs as follows:
a) the multimeter function is selected, in this case DC voltage measurement,
b) the measuring range is selected, including or near to the measured value, if measured value is unpredictable, the uppermost voltmeter range is to be selected,
c) the measured voltage is connected to the proper sockets of the meter, typically there are separate sockets for connecting voltage (and resistance) and current,
d) the result of measurement is red on the display.

The meter range used depends on required accuracy of results. For example, suppose that the voltage to be measured with the M-4650B meter is about +10 V. According to the technical specifications of the meter, the error of DC voltage measurement in the range from 200 mV to 200 V is defined by the formula: ±0.05% of measured value ±3 on least significant position of the range. The selected range is 20 V and the display reads 10.020. Then maximum value of error of measurement is equal to:

$$\Delta V = \pm 0.005 \pm 0,\ 003 = \pm 0,\ 008\ \mathrm{V} \qquad (1.1)$$

It means that the real value of measured voltage is somewhere between 10.012 V and 10.028 V. If the measurement is done using with the range 200 V, the display will read 10.02 V with an error

$$\Delta V = \pm 0.005 \pm .\ 03 = \pm 0.035\ \mathrm{V} \qquad (1.2)$$

which is 4 times greater.
An attempt to measure +10.020 V using the range 2 V ends unsuccessful; the display shows range overrun.
Note! There are digital multimeters/voltmeters with automatic change of measuring range. Multimeter this of type is included in the multifunction device MX-9000/9300. Multimeter with an automatic selection of measuring range chooses the range in such a way that minimizes the measurement error.

The measurement procedure of DC voltage by means of an oscilloscope, say as our OS-9040, runs as follows.
a) Switch the instrument on and give it few minutes for warm-up,
b) Select measuring channel; let it be channel l. Set subsequently:
- trigger mode - 'AUTO'
- trigger source - internal from channel l,
- input mode - 'DC',
- reference line position - align with the bottom or top scale line (depending on expected polarity of measured voltage),
- time-base speed - not too low to prevent display from blinking (>2 ms/div).

c) Set vertical sensitivity according to expected value of measured voltage. For example, if measured voltage is 10 V, the sensitivity should be 2 V/div.
Then, considering full measuring area, 8 divisions x 2 V/div = 16 V, so that the picture should not disappear out of the screen.
d) Connect measured signal (DC voltage) to the input of channel l.
e) Count the number of divisions the line has moved and multiply it by selected sensitivity. That's it.

The accuracy of DC voltage measurement by means of the is considerably smaller than the accuracy of the same measurement with the digital voltmeter. For example, measurement of 10

V DC voltage has accuracy less than ±3% , because of calibration error of the sensitivity control, which is typically ±3%, plus and an additional error of reading the line position on the screen. So that, the oscilloscope measurements are not very precise, but fast, and sometimes irreplaceable (as for measurements of voltage differences in signal at different moments of time).

Note! More modern oscilloscopes, both analog and digital, are equipped with built-in digital voltmeters of high precision of measurements. Result of measurement is displayed directly on the screen.

The problem of loading.

To measure a voltage (or other electrical quantity) between two points in a circuit we connect an instrument to them. From these two points the all circuit under test may be considered as an ideal voltage source (EMF) connected in series with some resistance. Any resistance (e.g. internal resistance of the voltmeter) connected to such source starts some current to flow out of the source producing some voltage drop on its internal resistance. Now the output voltage of the source is smaller than it was before. If the internal source resistance is of relatively large value (in respect to the connected external resistance - the loading resistance - the internal resistance of the voltmeter) - the voltage measured across the load will considerably differ from the voltage between connection points when the voltmeter is not connected.

For example, if to the DC source with an EMF of 10 V and internal resistance of l MΩ the oscilloscope with 1 MΩ input resistance is connected then the output voltage of the DC source will change from 10 V to 5 V - half of its EMF value. The error of measurement of the EMF value reached 50% ! If the same measurement is done with a digital multimeter having internal resistance of 100 MΩ, the value of measurement error caused by loading effect decreases to 0,9%.

2.2. DC current measurements

Measurement of current intensity needs to break the circuit where it flows and to connect into the break an ammeter. At present the practical meaning have ammeters designed in the form of small value resistor in parallel connection with the DC voltmeter. And so measurement of a current is replaced by measurement of the voltage drop across the standard resistor and calculation of the current value from the Ohm's formula:

$$I = V/R_{standard} \qquad (1.3)$$

Usually the maximum voltage drop across the standard resistor is fixed to 200 mV(for the end of range). Then for the 2 mA range the standard resistor should be equal to 100 Ω. Resistor of such a value may well disturb the circuit behavior if is inserted in the current path. Let us suppose that the circuit is formed from series connection of voltage source with EMF = 0.2 V and of 100 Ω resistor. Inserting an ammeter with range set to 2 mA in such a circuit decreases the current twice, so it reads 1 mA instead of 2 mA. After switching the ammeter range to 20 ($R_{standard} = 10\ \Omega$) the error coming from deterioration of the circuit is smaller but the meter gives less accurate result.

The general conclusion concerning the voltage and current measurements: for voltage measurements use voltmeters with as high internal resistance as possible, for current measurements use ammeters with as low internal resistance as possible.

2.3. AC voltage parameters measurement

In this experiment some basic parameters of sinusoidal, rectangular and triangle signals will be measured using the oscilloscope.

For every signal its peak-to-peak voltage value and a period of repetition is to be measured. From result of period measurement one can find the frequency of given signal.

For rectangular signals an important parameter to measure is the duty-cycle. It is defined as a relation of part of the period when the signal takes value greater from two possible to full signal period.

For unsymmetrical triangular signals two additional parameters are defined: the speed of the rising edge and the speed of the falling edge.

The accurate measurement of periodic signal frequency can only be done by means of digital frequency meter. The laboratory digital frequency meter, being a part of MX-9000 device, works on the base of counting numbers of signal periods during precisely defined time window - the gate opening time. Resolution of this method is proportional to the gate opening time, so (roughly speaking) the longer the gate-opening time the more accurate result. In general, the problem of the frequency measurement accuracy is not so trivial.

3. EXPERIMENTAL PART OF THE ACTIVITIES

3.1. DC voltage measurement

3.1.1. Measurement of the output voltages of power supplies built-in into the MX-9000 instrument by means of a multimeter

a) Set the output voltage of the adjustable power supply equal 15 V (read value from the power supply display). Remaining two supplies have their output voltages fixed.
b) Set the multimeter function: DC voltage measurement.
c) Set the multimeter range: 20 V.
d) Using multimeter wires placed into suitable sockets (which ones?), connect the multimeter successively to the power supplies outputs. Note results of measurements. Calculate and note values of measuring errors.
e) Change measuring range of the multimeter - first to lower, after that to higher. Note your conclusions under the table with results of measurements.

3.1.2. Measurements of the output voltages of power supplies built-in into the MX-9000 instrument by means of the oscilloscope

a) Do not change of the adjustable power supply setting, it should be the same as in 3.1.1. a)
b) Set the vertical sensitivity in order to obtain maximum move after connection the measured voltage (try to estimate this sensitivity in advance).
c) **Check if the vertical sensitivity has been calibrated.**
d) Set controls: input mode - DC, channel selection - CH1(channel I), triggering - AUTO, CH1, time-base mode - normal (A), time-base A speed - any, without flicking.
e) Connect the channel 1 input to the output of each power supply, read and note number of divisons the line in each case has moved by.
f) Calculate and note in your report the value of the voltage.
g) Comment in the report the accuracy of voltage measurements by means of an oscilloscope.

3. 1. 3. Measurement of the voltage at the output of the low- and high-resistance voltage divider - problem of loading the measured circuit with the measuring device.

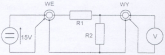


Fig.1.6. Voltage measurement at the resistor divider output

a) Set measuring circuit according to the schematic diagram shown in Fig.1.6. Using a switch on the front panel of the M-1 unit select the voltage divider with resistors R1, R2 equal to 1kΩ
b) Measure the voltage at the output of the divider. Note the result of measurement, making a table is recommended.
c) To the output of the divider connect the oscilloscope (in parallel with the multimeter). Note the result of measurement red from the multimeter display.
d) Repeat measurements as in points b) and c) for the divider composed with 1MΩ resistors. Note results of measurements.
e) Comment observed differences.

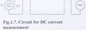


Fig.1.7. Circuit for DC current measurement

3.2. DC current measurement

3.2.1. Measurement of the power supply output current by means of the multimeter

a) Arrange the measuring circuit as in Fig.1.7. Use section 'Pomiar prądu stałego' in the module M-1. Connect a DC source with the EMF of 1V to the socket 'WE'. To the socket 'I-U1' connect a miliammeter. The switch should be depressed, i.e. at position 'I'.
b) Measure the value of the current using multimeter set to 2 mA range and then change the range to 20 mA. Results of measurements place in suitable cells of previously prepared table.
c) Why the results of measurements differ?

3.2.2. Measurement of the current by measuring voltage drop across a resistor of known value

a) Disconnect the miliammeter from the measuring circuit (Fig. 1.7).
b) Press the switch - select position 'R-U1'. Measure and note the resistance of the resistor R (it may differ from the value shown in the schematic).
c) To the socket 'I-U1' connect the DC voltmeter.
d) Basing on Fig.1.7 draw in the report the schematic diagram of actual measuring circuit.
e) Measure the voltage and note it in the report.
f) Calculate the value of the current flowing through the resistor R and note it in the report.
g) Compare the result with preceding measurement of the DC current.

3.3. AC voltage parameters measurement

3.3.1. Measurement of sinusoidal signal parameters

a) Connect measuring circuit as in Fig.1.8.

b) Using the function generator controls set its output voltage as sinusoidal wave of 1 kHz frequency and peak-to-peak value of about 15 V. The generator internal resistance should be set to 50Ω, the DC offset to 0 V.

c) Measure by means of the oscilloscope the peak-to-peak voltage and the period of the signal. Choose the proper time-base speed - the screen should display no more than one or two periods of measured signal (why?).

d) Measure the signal frequency by means of the digital frequency meter.

e) Results of measurements place in the table.

f) Repeat all measurements for about 1 MHz signal of the same amplitude.

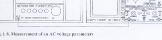


Fig. 1. 8. Measurement of an AC voltage parameters

3.3.2. Measurement of rectangular signal parameters

a) Using the same measuring circuit as before (Fig.1.8) switch the output to rectangular wave of about 1 kHz frequency, p-p value of about 15 V, the DC offset of 0 V and the duty-cycle equal to 0.5.

b) Measure the rectangular wave parameters and note results of measurements in the table.

c) Considerably change the duty-cycle factor of the signal, repeat previous and note results.

d) Change the signal frequency from 1 kHz to 1 MHz. Is the observed signal still of rectangular wave shape?

3.3.3. Measurement of triangular signal parameters

a) Using the same measuring circuit as before (Fig.1.8) switch the output to triangle wave of about 1 kHz frequency, p-p value of about 15 V, the DC offset of 0 V and the same slopes for both edges.

b) Measure parameters of the triangle wave and record results of measurements in table.

c) Change the triangle wave shape, so the leading edge slope is different from the trailing one. Repeat measurements as in previous case. Note results of measurements.

d) Change the signal frequency from 1 kHz to 1 MHz. Describe observed differences in shape of signal.

Michał Ramotowski

ELECTRONIC LABORATORY

Instruction sheet No. 2

INVESTIGATION OF PASSIVE ELECTRONIC COMPONENTS

(The text is freely based on the book:
'Laboratorium Podstaw Elektroniki, Materiały Pomocnicze, Część 1',
with the kind permission of the author
Bogusław Kalinowski)

Experiment 2

INVESTIGATION OF PASSIVE ELECTRONIC COMPONENTS

Goals of the experiment:
- familiarize with passive elements used in electronics
- measurement of basic parameters of passive elements

Scope of the experiment:

1. Introduction to experiment (preview of passive components used in electronics).

2. Investigation of resistor proprieties:
2.1. Measurement of the resistance of the resistor.
2.2. Observation of phase relations between current and voltage.

3. Investigation of capacitor proprieties:
3.1. Measurement of the capacitance of the capacitor.
3.2. Observation of the phase relation between current and voltage,
3.3. Measurement of the amplitude characteristic of the low-pass filter.

4. Investigation of an inductor proprieties:
4.1. Observation of phase relation between current and voltage.
4.2. Estimation of the inductance of the inductor.
4.3. Measurement of the amplitude characteristic of the low-pass filter

5. Investigation of semi-conductor diode proprieties:
5.1. Measurement of a static characteristic of the rectifying semiconductor diode, deriving parameters of the piecewise-linear semiconductor diode model.
5.2. Observation of a rectifying circuit work.

Equipment on the laboratory bench:

M-2 - laboratory module for voltage and current dividers with resistors;
MX-620 - universal digital meter;
OS-9040 - 2-channel oscilloscope;
MX-9000 - multifunction device;
Connection cables: BNC-BNC - 2 pcs, BNC-banana - 1 pcs, banana-banana -2 pcs.

1. DESCRIPTIONS OF PASSIVE COMPONENTS

1.1. Resistor

The resistor is a twoport component used for control the current value in the electric circuit. Forcing a voltage at the resistor terminals results in a current flow having value proportional to the forced voltage. The flow of the current produces heat in the resistor what warms-up the component and may end sometimes in burning it. So the maximum dissipated power is an important parameter of the resistor. The most important parameter of the resistor is of course the resistance. A few more parameters of the resistor are listed below:

- tolerance of a resistor,
- maximum allowable terminal voltage,
- temperature coefficient of resistance.

In an electric circuit containing a linear resistor R the relation between the voltage V at the resistor terminals and the flowing current I comes from the well-known Ohm's law:

$$V = R \cdot I \qquad (2.\ 1)$$

This relation is valid for both direct current and for momentary values of variable current of about any shape as well. The more general notation of the formula (2.1) is as follows:

$$v(t) = R \cdot i(t) \qquad (2.2)$$

If the voltage $v(t)$ is a sinusoidal wave then the current $i(t)$ is also of the same shape. There is no phase shift between the voltage and current in a circuit consisting of only resistors.

1.2. Capacitor

Capacitors can store an electric charge, so they can store energy. This stored energy is the energy of the electric field existing inside the charged capacitor. The basic parameter of the capacitor is its electrical capacitance marked with symbol C. The capacitance C relates the stored charge Q in the linear capacitor charged to voltage V. For the DC voltage the relation is as follows:

$$Q = C \cdot V \qquad (2.\ 3)$$

For variable voltages and charges there is similar dependence:

$$q(t) = C \cdot v(t) \qquad (2.\ 4)$$

Differentiating the above and taking into account that the time derivative of charge is simply a current, we have:

$$d/dt\ q(t) = C \cdot d/dt\ v(t) = i(t) \qquad (2.\ 5)$$

If the voltage $v(t)$ is a sinusoidal wave $v(t) = |V|sin(\omega t)$, then the current flowing through the capacitor C also is a sinusoidal wave $i(t) = \omega C|V|cos(\omega t) = |I|sin(\omega t + \pi/2)$ leading the voltage for about 1/4 of period. Or speaking other way, the voltage lags the current for about $\pi/2$ radians (is shifted in phase about -90°, i.e. about -1/4 of period).

All the capacitors are formed from two parallel metal layers, very close each other, separated with an insulating material called dielectric. Most often as dielectrics are used: air, ceramics, paper, mica, plastic (polystyrene, polyethylene, policarbons, teflon) and metal oxides. Capacitors usually take their type names from a kind of used dielectric. Capacitors used for constructions are of two basic types: not polarized and polarized (electrolytic).

Except of nominal capacitance there are few more parameters defined for capacitors:
- tolerance of a capacitor (a maximum allowable deviation of its value from the nominal one),
- maximum allowable terminal voltage,
- temperature coefficient of capacitance,
- coefficient of dielectric losses (tg δ),
- leakage current or intrinsic time constant of the capacitor.
- leakage or resistance of isolation (or intrinsic time constant of a capacitor).

1.3. Inductor

A current flowing through an inductor produces a magnetic field. Inductors can store the energy of a magnetic field. The basic parameter of the inductor is its electrical inductance marked with symbol L. The inductance L relates the induced magnetic stream Ψ in the linear inductor to the current I flowing through it. The relations are as follows:

$$\Psi = L \cdot I, \ \text{for the DC current} \qquad (2.6)$$

$$\psi(t) = L \cdot i(t), \quad \text{for the variable current} \tag{2.7}$$

Differentiating (2.7) and taking into account that the time derivative of magnetic stream is simply a voltage (precisely - EMF of self induction), we have:

$$d/dt\ \psi(t) = -e_L(t) = L \cdot d/dt\ i(t) = v(t) \tag{2.8}$$

If the current $i(t)$ is a sinusoidal wave $i(t) = |I|sin(\omega t)$, then the voltage on inductor L also is a sinusoidal wave $v(t) = \omega L|I|cos(\omega t) = |V|sin(\omega t + \pi/2)$ leading the current for about 1/4 of period. Or speaking other way, the current lags the voltage for about π/2 radians (is shifted in phase about -90°, i.e. about -1/4 of period).

Every wire conducting a current behaves as an inductor. Its inductance grows considerably larger, when the wire is forming a coil, or many coils, and, if inside of the coils is placed a piece of a ferromagnetic material called as a core of the inductor. Examples of core materials are steel alloys and ferrites.

Designers of electronic equipment always try to minimalize number of inductors used in designed device, trying to replace them where possible with capacitors or sophisticated active circuits. Problem is especially essential in design of integrated circuits. However, it is not always possible to get rid of inductors, e.g. if a magnetic coupling is to be applied.

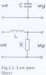


Fig.2.1. Low-pass filters

1.4. Low- and high-pass filters

Resistors, capacitors and inductors are basic components of simple useful circuits - twoports - which are used for analog filtration of electric signals. They are called electric filters.

The circuits shown in Fig. 2.1 are low-pass filters. They let pass without suppression or small suppression, not exceeding 3 dB (that is, $Vout/Vin > 0.707$) signals with frequencies from 0 to f_{3dB}, defined as below:

$$f_{3dB} = 1/(2\pi RC), \text{ or } f_{3dB} = R/(2\pi L) \tag{2.9}$$

:

Transmittances, that are to say relations of: output to input voltages of not loaded filters, in real and complex form are as below:

$$\frac{V_{wy}}{V_{we}} = \frac{1}{j\omega RC + 1}; \quad \left|\frac{V_{wy}}{V_{we}}\right| = \frac{1}{\sqrt{\omega^2 C^2 R^2 + 1}} \tag{2.10}$$

$$\frac{V_{wy}}{V_{we}} = \frac{R}{j\omega L + R}; \quad \left|\frac{V_{wy}}{V_{we}}\right| = \frac{1}{\sqrt{\omega^2 L^2 + R^2}} \tag{2.11}$$

The frequency dependence of the transmittance magnitude is called an amplitude characteristic of the filter. The damping of the filter rises 20 dB (i.e. ten times) for each decade of frequency rise for frequencies greater than f_{3dB}.

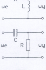


Fig.2.2. High-pass filters

The circuits shown in Fig.2.2 are high-pass filters. They let pass without suppression or small suppression, not exceeding 3 dB

(that is, Vout/Vin > 0.707) signals with frequencies greater than f_{3dB}. The damping of the filter falls down 20 dB (i.e. ten times) for each decade of frequency decrease for frequencies smaller than f_{3dB}.
Transmittances, i.e. relations of output to input voltages of not loaded high-pass filters, in real and complex form are as below:

$$\frac{V_{wy}}{V_{we}} = \frac{j\omega RC}{j\omega RC+1}; \quad \left|\frac{V_{wy}}{V_{we}}\right| = \frac{\omega RC}{\sqrt{\omega^2 C^2 R^2+1}} \tag{2.12}$$

$$\frac{V_{wy}}{V_{we}} = \frac{j\omega L}{j\omega L+R}; \quad \left|\frac{V_{wy}}{V_{we}}\right| = \frac{\omega L}{\sqrt{\omega^2 L^2+R^2}} \tag{2.13}$$

The low-pass filter is called sometimes as integrating circuit, and high-pass filter as differentiating circuit. It comes from observations that $\omega > 1/(CR)$ or $\omega > R/L$ for low-pass filters or for $\omega < 1/(CR)$ or $\omega < R/L$ for high-pass filters their output voltages can be considered as an integrals or derivatives of their input voltages respectively. The integrating or differentiating effect of signal is best visible for an input voltage in form of rectangular wave with properly chosen frequency.

1.5. Semiconductor diodes

The semiconductor diode is a two terminal element conducting a current only in one direction. For the purpose of our experiments the simple diode model in a form of a nonlinear, noninertional resistive element will be sufficient. The model characteristic is given with an exponential formula:

$$I = I_S\,[\exp(V/V_T) - 1] \tag{2. 14}$$

Where: $V_T = kT/q = 25.8$ mV for T = 20 C.

The reverse polarized diode may be sometimes treated as an opening. For fast calculation the diode model in conducting region may even more simplified - its exponential characteristic is approximated with two sections of a straight line. It leads to the model consisting only from two elements in series connection: the voltage source modeling so called firing voltage of the junction and the resistor modeling an average value of the conducting diode resistance.

There are many types of semiconductor diodes. Most important of them are: rectifying, pulse, LED (Light Emitting Diodes), laser, varicap (*vari*able *cap*acitance), and voltage stabilizers (Zener diodes). The basic parameters of the diodes given in catalogs are: the forward voltage drop V_F at specified forward current I_F and reverse breakdown voltage V_B. For special diodes some additional parameters may be defined. For example, for Zener diodes the stabilized voltage, differential output resistance and temperature coefficient of stabilized voltage at given working current are specified.

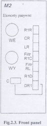


Fig.2.3. Front panel of the M-2 module

1.6. Measurement module M-2

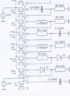


Fig. 2.4. Circuits in the module M-2

Measurement module M-2 is used for investigation of the passive component's properties. The components under test are: the resistor R1 (1kΩ), the capacitor C (10nF), the inductor L (250μH) and the semiconductor diode D - all of them are shown in Fig..2.4. To allow for simultaneous observations of the voltage across investigated components and their current, a small resistor is connected in series with components R1, L, C. The voltage across this small resistor is negligible in relation to voltage across the investigated component but is proportional to the component current, reflecting its shape.

The front panel of the module M-2 is shown in Fig.2.3 but schematic diagrams of circuits in this module are shown in Fig.2.4. For selection of the circuit a push-button switch is used. For example, if the key "R1R" is pressed, almost all the voltage connected to the socket "WE" arrives across the resistor R1 (the voltage across R is negligibly small). The voltage across R, being proportional to the current through R1, is connected to the socket "WY".
Similarly, same situation takes place when keys "CR" and "LR" are used.

Having the key "Filtr R1C" pressed, a low-pass filter is inserted between sockets "WE" and "WY". A second low-pass filter, consisting of inductor L and resistor R, is selected after pressing the key "RL". Two least keys - "R1D" and "DR1" -connect front panel sockets with semiconductor diodes' circuits. With "R1D" key the static diode characteristic is measured, using the key "DR1" one may observe the rectifier circuit operation. An additional switch "C" connects a capacitor in parallel to the resistor R being the rectifier load. Connections to the M-2 module are only possible with the coaxial cables of the BNC-BNC type.

2. EXPERIMENTAL PART

2. 1. Investigation of the resistor proprieties

2.1.1. Measurement of the resistor resistance

a) You are given resistors with values marked on their bodies: 10 Ω 100 Ω, 1.5 kΩ and 1 MΩ. Measure their real values of resistances by means of ohmmeter being a part of the multimeter from the instrument MX-9000 or portable multimeter (M-4650B or MX-620). Results of measurements write into prepared table.
b) Estimate an error of every measurement. Are the resistances of connecting wires of any meaning?

2.1.2. Observation of phase dependence between current and voltage

a) Select measuring circuit as in Fig. 2.5, with R1 = 10 kΩ and R = 10 Ω (the resistor R is used here as a sensor of the current flowing through Rl). The output signal of the generator GEN should be sinusoidal wave with a the peak-to-peak value of about 15 V and frequency of 10 kHz. The oscilloscope should be set to two-channel mode with internal triggering from channel l.

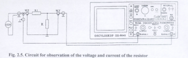


Fig. 2.5. Circuit for observation of the voltage and current of the resistor

Remember to set the input modes to 'DC' for all phase shift observations (ask your supervisor for comment).
b) Sketch waveforms observed on screen in your report. Note: sensitivities of both channels and time-base speed.
c) Write your conclusions, if there exists any phase shift between current flowing through the resistor and the voltage across its terminals. What happens after change of shape of signal and its frequency?

2.2. Investigation of the capacitor proprieties

2.2.1. Observation of phase dependence between current and voltage
a) Connect measuring circuit as in Fig. 2.6, with C = 10 nF and R = 10 Ω (the resistor R is used here as a sensor of the current flowing through C). The output signal of the generator GEN should be sinusoidal wave with a the peak-to-peak value of about 15 V and frequency of 100 kHz. The oscilloscope should be set to two-channel mode with internal triggering from channel l.
b) Sketch waveforms observed on screen in your report. Note: sensitivities of both channels and time-base speed.
c) Write your conclusions, if there exists any phase shift between current flowing through the resistor and the voltage across its terminals. What happens after change of shape of signal and its frequency?

Fig.2.6. Circuit for observation voltage and current of the capacitor

2.2.2. Measurement of the low-pass filter amplitude characteristic

a) Connect measuring circuit as in Fig. 2.7, selecting C = 10 nF and R1 = 1 kΩ. The output signal of the generator GEN should be sinusoidal wave with the peak-to-peak value of about 15 V and frequency varied from 100 Hz to 2 MHz. The oscilloscope should be set to two-channel mode with internal triggering from channel I.
b) Varying the GEN frequency measure the relation of the filter output p-p voltage to its input p-p voltage.. Results of measurements and calculations write into prepared table.
c) Results of calculations of filter attenuation factor *k* [dB] present in form of a graph.
d) Estimate from the graph the upper f_{3dB} frequency of the filter, and then check it experimentally (write in the report how you did it).

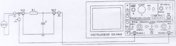


Fig.2.7. Circuit for measurement of the low-pass filter amplitude characteristic

Fig.2.8. Circuit for observation voltage and current of the inductor

2.3. Investigation of the inductor proprieties

2.3.1. Observation of phase dependence between current and voltage

a) Connect measuring circuit as in Fig. 2.8, selecting L = 250 μH and R = 10 Ω (the resistor R is used here as a sensor of the current flowing through L). The output signal of the generator GEN should be sinusoidal wave with the peak-to-peak value of about 15 V and frequency of 100 kHz. The oscilloscope should be set to two-channel mode with internal triggering from channel I.
b) Sketch waveforms observed on screen in your report. Note: sensitivities of both channels and time-base speed.
c) Write your conclusions, if there exists any phase shift between current flowing through the resistor and the voltage across its terminals. What happens after change of shape of signal and its *frequency?*

2.3.2. Measurement of the low-pass filter amplitude characteristic

a) In the circuit assembled as in Fig.2.8 replace resistor R = 10 Ω with resistor Rl = 100 Ω. For the output pp voltage to the input pp voltage ratio measure its dependence on frequency. Note results of measurements and of calculations.
b) Varying the GEN frequency measure the relation of the filter output p-p voltage to its input p-p voltage.3 Results of measurements and calculations write into prepared table.
c) Results of calculations of filter damping *k* [dB] present in form of a graph.
d) Estimate from the graph the upper f_{3dB} frequency of the filter, and then check it experimentally (write in the report how you did it).

2.3.3. Estimation of the inductance value of the inductor

a) Having measured the upper f_{3dB} frequency of the low-pass LR filter calculate the inductance of filter inductor.

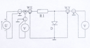


Fig.2.9. Circuit for measurement of the static characteristic of the diode

2. 4. Investigation of the semiconductor diode proprieties

2.4.1. Measurement of the static characteristic of the rectifying semiconductor diode; calculation the parameters of the diode model.

a) Assembly the measuring circuit as in Fig. 2.9. As the resistor R use the resistor 1 kΩ (the value marked on its body). Input voltage change from 0 to about 20 V. Results of measurements place in prepared table.
b) Measure the real value of R1 and note it ina heading of the table.
c) For every value of the input voltage calculate the diode current:

$$I = (V - V_{in}) / R1$$

and place it in the proper place in the table.
d) Draw the diode current versus the diode voltage.
e) Approximate the diode characteristics using two sections of a straight line. Find the diode model elements: the voltage source and the series resistance.

2.4.2. Observation of the waveforms in the rectifying circuit
a) Connect the measuring circuit as in Fig. 2.10, with Rl = 1 kΩ. The output signal of the generator

Fig.2.10. Circuit for investigation of the rectifier waveforms

GEN should be sinusoidal wave with the peak-to-peak value of about 15 V and relatively low

frequency (say, below 500 Hz). The oscilloscope should be set to two-channel mode with internal triggering from channel I.
b) Sketch waveforms observed on screen in your report. Note: sensitivities of both channels and time-base speed.
c) Change frequency of the input signal. Explain observed changes of the output signal.
d) To the resistor Rl connect in parallel given capacitor C. Has the output signal shape changed? How does the circuit behave now if the input signal frequency changes?

Michał Ramotowski

ELECTRONIC LABORATORY

Instruction sheet No. 3

BASIC APPLICATION OF PCB DESIGN SOFTWARE

(The text is freely based on the book:
'Laboratorium Podstaw Elektroniki, Materiały Pomocnicze, Część II',
with the kind permission of the author
Marek Pawłowski)

Experiment 3

BASIC APPLICATION OF PCB DESIGN SOFTWARE

Goal of the experiment:

- familiarize with placing of elements on printed boards and connecting them with metalization paths.

3.1. Scope of the experiment:

1. Demonstration of preparation of connections list and its application for PCB design. Placing and connection of some components selected from previously prepared schematic diagram.
2. Completing PCB design and printing of silkscreen and signal layers.

Equipment on the laboratory bench:

- terminal of school LAN with access to a printer,
- software: Integra Station of Mentor Graphics.

3.2. Printed Circuit Boards - short characterization

In electronic devices printed boards are used for two reasons. First from them is to fasten of elements in device, and second - to electrically connect them. The isolated wires are replaced with metallic stripes connecting soldering points, to which are fitted elements' pins. Soldering points can contain metalized hole, which holds soldered component pin or they can create small metalized areas for soldering surface mounted devices (SMD's).

Depending on number of necessary connections among elements placed on the board there may be used one, two or more signal layers. Signal layers are separated with an isolating layers that mechanically supports all the construction. The interconnections of signal layers are realized in pin soldering points or with special smaller metalized holes, called **via**.

The automated process of drilling and placing of components needs all the holes in the board to be placed in a specific raster. The raster is a virtual mesh of equally spaced perpendicular vertical and horizontal lines. The distance between adjacent lines is usually given in inches and equals 0.1" in basic raster and 0.05" in a smaller one. The components are placed at intersections of raster lines, whereas the traces are placed along the raster lines. In case of two-signal layer board the vertical traces are located on one side and the horizontal ones on the second side of the board.

Soldering points.

The soldering point is formed from soldering areas connected with a metalized hole in the isolation substrate. In case of one-sided board there is only one soldering area and a hole without metalization. The soldering point parameters are:

- d_w - hole diameter before metalization (size of the drill)
- d_o - inner diameter of metalized hole (it results from maximum diameter of element d_{emax}, enlarged about 0.3 mm for mounting clearance),
- D - diameter of soldering area.

The soldering point parameters for different element pins' diameters for 0.06" laminate thickness are give in the table below. (All dimensions in inches, on the basis of the table 8.7 [1]).

d_{wmax}	Components and packages	d_o	d_m	D	
				min	preferred
0.020	Metal resistors up to 0.25W Electr. caps up to 22mF/100V Diodes up to I_F = 100mA (DO13) Low power transistors (TO18) Logic gates (DILxxx)	0.025	0.03	0.05	0.055
0.028	Wire resistors Electr. caps up to 2200mF/63V Rectifier diodes (DO41)	0.03	0.04	0.055	0.070
0.034	Power transistors (TO3, TO220)	0.04	0.05	0.070	0.090
0.044	Rectifier diodes (DO27)	0.05	0.06	0.090	0.100

Traces

Trace width depends on the current in it, maximum voltage drop on it, what does have to lead, admissible parasitic inductance and capacitance and precision of etching.

The minimum trace width depends on the maximum allowable voltage drop ΔV along the trace. For the copper foil of 0.0014" thickness the minimum width W of the trace of length L may be found from the expression below:

$$W = (I_{max} \cdot L \cdot 0.5 \cdot 10^{-3}) / \Delta V$$

Units: I - amperes, ΔV - volts, W and L - millimeters.

The copper foil has to be etched from those areas of the board that are no solder points, traces or vias. The etching process removes some copper from edges of masked traces, so it should be considered when selecting trace width at design time. For the copper foil of 0.0014" thickness these losses are of order of 0.002" (2 mils) each.

Recommended widths of traces for two-layer boards with large number of connections:
0.010" - 0.015" - for signal traces placed between IC pins,
0.020" - 0.030" - for other signal traces,
0.060" - 0.080" - for power supply and ground traces.

Another important factor are distances between traces. They depends on: voltage difference between paths, admissible crosstalks, soldering technology (manual, on wave), working environment of printed board (humidity, pressure and air pollution).
Minimum clearances between traces are: 0.015" for voltages 0-50V, 0.025" for voltages 51-150V, 0.050" for 151-300V. Basic rule of leadership of paths.

Some basic rules of routing traces

1. Traces should be routed through raster intersections.
2. For multi-layered boards traces should be of vertical or horizontal direction (this rule forces more vias, but makes possible larger density of connections), leaving this rule is allowed in close distance to soldering point, to which the trace is connected.
3. For low noise signals number of vias should not exceed 3.

4. Traces should connect soldering points on solder layer.
5. Traces should approach soldering point radially and end in its center.
6. Necessary clearances to other traces, soldering points and vias should be preserved.

3.3. Integra Station 3.1.

In the experiment the INTEGRA Station PCB design system is used. The system consists of schematic editor, layout editor, component editor, package editor, and more. The user may start from drawing the schematic diagram of the device, next may place components on the piece of a virtual laminated board, manually or using autoplace facility, next may route traces, again manually or using an autorouter tool. A photoplotter control file may be generated and a drill file as well. But what we are going to do in this experiment is only a small fragment of the full PCB design process. We will use earlier prepared schematic diagram of two-transistor RTL inverter to learn only a small piece of the layout design process. Full INTEGRA Station possibilities are far beyond the scope of this exercise.

To start the system click the proper icon on the Win95 desktop, or select the 'INTEGRA Station' from the Start>Programs menu. After a window "Integra Station for Windows" is open select **File>Open Project** option, which will open window "Open". Find the directory **Integra\projects\User\mils** and choose project name **lpe_cw7**. The project file full path will be displayed in the Project Manager window. Click the path, it became grayed.

3.3.1. Introduction to the PCB editor of the INTEGRA Station system

To begin edition of the board click either the right mouse button inside the Project Manager window and select **Layout editor** or click an icon marked with a IC package image (second icon from left in line of tools). The editor window pops-up containing the project at some phase of design, or empty if we begin new project.

The user interface is 'window sensitive'. the menu and tools lines and columns contents depend on what window is actually active.

Having the editor window active the desktop contains: line of menus' names, line of tools' icons, two columns of tools with icons, project desktop and a status line. The most important for laboratory work desktop components are described below:.

Menu line contains:
- **File** : open an existing project, store the entire project, manage libraries for the project, import/export data in DXF format (e.g. from/to Autocad), manage circuit modules (selected schematic blocks),
- **Edit** : undo/redo last operation, functions for altering, copying, and deleting objects.
- **View** : change the view of the layout, or the appearance of objects on the layout (zooming), mark areas for autoplace/autoroute tools.
- **Insert** : create or copy from libraries all the objects you need on a layout - packages, traces, vias, etc.
- **Edit Logic** : edit and handle the connections and components of a circuit without using the Schematic Editor.
- **Setup** : specify settings for a variety of parameters which affect the Layout Editor esp. grids, layers, etc.
- **Tools** : tools for autoplacing components or autorouting in the layout, run test routines that check the validity of the layout

- **Reports** : output varying types of information about the layout and the project:
- **Help** : invokes on-line help files, glossary, tutorial, etc. .

Tool icon line contains:

- **Esc** : cancel the current function.

Six object filter icons - **Component, Trace, Pad/Via, Area, Text, Testpoint** - selection/deselection of objects of same type.

Six zoom type icons

- **Zoom Show All** - show whole work area.
- **Zoom - Objects** - display an area to include all objects., the size depends on the number and distribution of objects.
- **Zoom - Selected Objects** - display an area to include all *selected* objects.
- **Zoom In** - decrease the area shown. The size of the objects shown increases.
- **Zoom Out** - increase the area shown. The size of the objects shown decreases.
- **Zoom - Mark Area** - zoom into a marked area on the screen
- **Refresh** (Mentor Graphics logo) - refresh and renewal of content of screen
- **Layer Icon** - open the Layers menu to change the layer displayed.
- **Undo** - function to undo your *transparent (see Help)* actions one after the other, right back to the starting state of your current session.
- **Redo** - function to page forwards again through UNDO actions.
- **Overview Windows** - zoom quickly to exactly the part of the work area you want, fast way to place the cursor onto part of board invisible on screen.
- **Online Help** - access to manual including description of INTEGRA system functions.

Two vertical column of tools:

- **Select** - click when you want to select objects first, then apply functions to the selected objects.
- **Deselect** - deselect all currently selected objects.
- **Copy** - copying of select objects into the place indicated with cursor.
- **Move** - shift of select objects in into the place indicated with cursor.
- **Delete** - removal of select objects.
- **Rotate** - rotate of select objects anticlockwise at cursor position.
- **Fill Component Pool** - include all unplaced components in the component pool.
- **Place Component** - place unplaced components in the layout, after you select the component to place from the list of unplaced components.
- **Begin Trace** - start trace, change trace direction or layer, end trace.
- **Modify** - modify the selected object attributes.
- **Draft Text** - place user-defined text on selected layer.
- **Fix** - block (fix) position of select objects.
- **Unfix** - unblock position of select elements;
- **Change Layer** - click on the icon to move objects to another layer.
- **Route Guide** - click on the icon to convert selected guide into a trace.
- **Measure From** - measurement of distance between two points indicated by cursor.
- **Close** - shut the currently active tool.

The short abstract from Help files given above can be used as a guide to find more information in the Online Help of INTEGRA system. Many menus' items have submenus not referenced here. Students are kindly advised to preview documentation accompanying INTERGRA Station system.

3.3.2. Design project steps in the INTEGRA Station system

The PB design procedure consists of few typical subsequent steps. described below. Description of special situations not mentioned one may find in the INTEGRA Online Help. To select menu item click left mouse button, the same button is used to confirm an operation. Clicking right mouse button will display so-called **Pulldown menu**. This menu contents depends on executed operation and gives an attempt to additional functions related to this operation.

Defining work area. From menu **Setup>Definitions** choose option **Work Area** and write down work area using mils as units (l mil = 1/1000 of inch).
Defining board outline. The board outline is a borderline limiting the area the components and traces are placed in. The board outline has to fit inside the work area (marked with a default white line). From menu **Insert>Draw>Board Outline** choose a standard form (**Rectangle** - rectangle, **Polygon** - shapeless polygon, **Circle** - circle), it may be adjusted later on to get really any final shape of the board. The default color is purple.
Placing elements on the board. The simplest way to dispose elements is click the icon **Place Component** and choose a suitable element from the list. Use cursor to position the element on the board. Before placing is confirmed the element may be rotated or mirrored (press R or M keys accordingly). keyingIn moment of placing of element possible modification of his arrangement is on plate np. To confirm placing press left mouse button.
Another way of placing elements is using menu **Insert>Component Pool>Define** to define an area called **Component Pool** for initial disposing of elements (Note - this area has to be marked outside the **Work Area**) and then moving them into the board with the help of function **Insert>Component Pool>Fill**. This method allows the designer to see the component package shape at the moment of placing it on the board.

On edges of work area one may see red diagrams illustrating density of traces along the axes. They should be as flat as possible - it helps routing traces.
When disposing elements the highest priority should have mechanical elements, displays, signaling LEDs, microswitches, etc. elements, which have to fit to the device cabinet or needs temporary service. After placing elements of such a kind it is a good practice to mark them and apply the function **Fix** what preserve them from subsequent moving, rotating or deleting.

Defining widths of signal and power supply traces. To do this use **Setup>Design Rules** and select an option **Nets**, what allows defining trace width for each individual net. Standard trace width is 8 mils, is but for supply paths (particularly in projects including larger quantity of elements) using of much wider traces, e.g. 20 - 50 mils is recommended. For defining of parameters for larger number of traces, the group of traces should be created (**Net Class**). It may be done in the same menu by choosing bookmark **Classes** and adding new class using button **Add**. Under the bookmark **Clearance** the following parameters of group of traces can be set:

- **Trace** - distances between traces and other objects,
- **Via** - distances between vias and other objects,
- **Thru-Pin** - distances between traditional component pins and other,
- **SMD-Pin** - distances between SMD pins and other objects,
- **Area** - distances between copper areas and other objects.

The parameters listed above can be set for two different trace classes choosing the bookmark **Class-To-Class**.

Manual routing of traces. To begin trace route click the icon Begin Trace, next click the start trace pin. having the left mouse button pressed draw the trace segment. To begin next segment click at the point of change. The trace color depends on selected layer. During drawing one may change some options selected from the **pulldown menu** invoked by clicking the right mouse button (observe shortcut keys marked on menu items names):

- **Change Layer** - change layer of drawing (possible use of shortcut key **L**),
- **Direction** - change direction of last segment,
- **Any Angle** - new segments can be placed at any angle to the previous segment,
- **90°** - new segments can only be placed at 90° to the previous segment,
- **45°** - new segments can only be placed at 45° to the previous segment;
- **Arc** - new segments can only be placed as arcs,
- **Oops** - remove the last segment you put;
- **Wider** - increase the width of the trace one step up of the widths defined,
- **Narrower** - decrease the width of the trace one step down of the width defined,
- **Cancel** - complete the trace at the last point defined and exits routing

The trace routing may be performed automatically if any of **Tools>Auto>Autoroute....** options is selected. The choices are:

- **Autoroute - Select Nets...** - select the nets to autoroute from the list of nets used in the project,
- **Autoroute - All** - autoroute all the selected nets throughout the layout,
- **Autoroute - Area** - autoroute the nets in the selected area
- **Autoroute - Component** - autoroute the connections to the selected components

For complicated and dense packed layouts the autorouter often can not route 100% of connections. In such a case a completion of work needs more attempts, typically done by hand.

3.4. Auxiliary literature.

INTEGRA Station Help file in the INTEGRA install directory.

3.5. Schematic diagram of electronic circuit

The schematic diagram of the PCB designed in the experiment is shown in Fig.3.1. The diagram is placed in the **Integra3 I\projects\User\mils** directory under the name **lpe_cw7**.

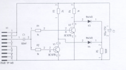


Fig. 3.1. Schematic diagram of the designed PCB

3.6. EXPERIMENTAL PART

The students should design two variant PCB boards for the circuit shown in Fig. 3.1.

Task 1. Place the circuit components in a way similar to shown in Fig.3.2. The PCB parameters are to be as below:

1. PCB dimensions: 4x2"
2. signal trace width: 0.05" (50 mils)
3. supply trace width: 0.12" (120 mils)

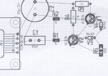


Fig.3.2. Suggested initial components placing for circuit from Fig.3.1

Task 2. Route manually traces in layout from Fig.3.2. Print resulting PCB and attach the copy to the report.

Task 3. Remove manually routed traces and repeat routing with the AUTOROUTE facility. Print resulting PCB and attach the copy to the report.

Michał Ramotowski

ELECTRONIC LABORATORY

Instruction sheet No. 4

SIMULATION OF LINEAR CIRCUITS

(The text is freely based on the book:
'Laboratorium Podstaw Elektroniki, Materiały Pomocnicze, Część II',
with the kind permission of the author
Marek Pawłowski)

Experiment 4

SIMULATION OF LINEAR CIRCUITS

Experiment goals:
- explanation of difference among emulation and simulation of an electronic circuit, their advantages and drawbacks,
- familiarize with a simple tool for simulation analog electronic circuits and systems.

1. Realization of the experiment.

1.1. Demonstration of the simulator possibilities. An introduction to the circuit description and simulation.

1.2. Investigation of the amplifier with a bipolar transistor.

Description of the investigated circuit. Simulation parameters and its documentation. Realization of simulation and making the report.

1.3. Investigation of the transient states in simple transistor inverters. Description of the investigated circuit. Simulation parameters and its documentation. Realization of simulation and making the report.

1.4. Investigation of the amplitude characteristic of the operational amplifier with a negative feedback.

Equipment on the laboratory bench:
- terminal of the school computer network,
- MicroSim PSPICE-DEMO software,
- Norton Commander program

2. Process of construction of an electronic device.

Process of construction of an electronic device begins with formulating project requirements – technical conditions. On the base of project technical conditions a block diagram is created containing basic functional block devices with description of their operation and cooperation with other blocks. Exact specifying of requirements and detailed description of each block function permits an independent design of device blocks by different designers. Their work results in a full schematic diagram of the project.

Following step of project design is verification of the project. Several years ago the verification of an electronic project needed to construct its model. Cost and time of realization of printed circuits forced assembling a model on universal boards. Apart from the cost of such process there were sometimes problems with making corrections (no needed components at hand).

At present there are programs making possible simulation of operation of a designed circuits or systems. Simulation permits to avoid most steps of physical modeling.

Simulation of full project or its selected parts relies on calculations of mathematical model of a circuit or system. An accuracy of simulation results depends on accuracy of these mathematical models of electronic components and devices. The component modeling is not a trivial work, it needs experience and skills.

2.1. Differences among emulation and simulation of a circuit

As it was said above, even with a help of a simulator there is necessity to build the physical model of the designed device. But even at the phase of real measurements, the nowadays technology may give a great help for testing the project. Suppose that you design a device controlled with a microprocessor containing preprogrammed ROM memory and some RAM memory. Testing such a device with an embedded microprocessor system is extremely hard task having in mind necessity of programming the memory again and again. The solution is the emulation of the microprocessor system with a help of computer. Using this method you may plug-in into your system a probe from the external computer that emulates the operation of full microprocessor system.

Then the **simulation** means *modeling* of electronic system in a computer *before* assembling it, whereas the **emulation** means testing physical device with a help of specially programmed computer that *emulates* an operation of some crucial blocks of a tested device.

3. Simulation of electronic circuits with the PSPICE simulator

According to *The American Heritage® Dictionary of the English Language, Fourth Edition Copyright © 2000 by Houghton Mifflin Company*, simulation is:

'Representation of the operation or features of one process or system through the use of another: *computer simulation of an in-flight emergency.'*

Simulator of electronic circuits analyses the mathematical model this of this circuit and calculates its parameters required by the designer.

3.1. Models of electronic elements

The computer representation of an electronic element (a 'model')) is typically a set of differential equations describing responses of the element to external signals the element is subjected to. The process of creating a model is not a simple task because a real passive components - resistors, capacitors, inductors are not 'pure' ones. For example, a real resistor is mainly characterized by its resistance value, but its has also some residual inductance and capacitance. What's more, such components are generally nonlinear, i.e. their values depends on the voltages or currents in them, what is especially true for all semiconductor devices. But trying to reflect all known properties of a real element in its model complicates its mathematical representation and subsequently increases the time of calculations and computer memory requirements. So, in everyday engineering practice some reasonable compromise between model complexity and its accuracy has to be taken. In order to do such a compromise models in simulators' software packets are always parameterized giving the user some freedom in selection of model accuracy.

Models of most frequently used components types are typically built-in into the simulator and are identified by specifying the model type name.

In PSPICE the model describing statement has a form:

.model model_name model_type_name([optional parameter list])

As a **model_type_name** the PSPICE accepts, among the others:
- **RES** - resistor,
- **CAP** - capacitor,

- **IND** - inductor,
- **CORE** - non-linear magnetic core,
- **D** - semiconductor diode,
- **PNP** - bipolar transistor PNP,
- **NPN** - bipolar transistor NPN,
- **NJF** - junction JFET transistor with channel n,
- **PJF** - junction JFET transistor with channel p

Optional parameters' values are declared by specifying the parameter name following by its value. If parameters' list is omitted, their default values are assumed (given below in brackets). For statistical analysis the parameter may be given a tolerance by using the keyword **DEV**. As na example consider the model statement for a resistor.

Resistor model **RES**.

In a resistor model RES exists following parameters (in brackets the *default* values are given - these values are taken by parameters if not explicitly specified):
- R - resistance multiplier (1),
- TC1 - linear temperature dependence coefficient (0),
- TC2 - square temperature dependence coefficient (0).

A temperature dependence of resistance is then given in the form:

resistance_value(T) = resistance_value * R * [1 + TC1*(T-T0) + TC2*(T-T0)]

Then the model library statement for the user defined resistor model MRES will have a form:

```
. model MRES RES( R=1 DEV 5% TC1=. 02 TC2=. 0045)
```

Similar model statements may be created for the other passive components - capacitors and inductors.

Semiconductor diode model **D**.

Diode model statement syntax can be used for p-n junction, Schottky barrier and Zener diodes. The basic model parameters are listed below (in brackets are given default values and their units)
- IS - saturation current (1.0E-14, ampere),
- RS - series resistance (0, ohm),
- CJO - junction capacitance at zero polarization (0, farads),
- VJ - diffusion potential (1, volt),
- BV - reverse breakdown voltage (100, volt),
- IBV - reverse diode current at reverse diode voltage V_d = BV(1.0E-3, ampere).

An example of diode model statement (for explanation of the multiplier suffixes 'p' and 'm' see p. 4.3.2):

```
. model MYDIODE D( Is=880.5E-18 Rs=. 25 Cjo=175p Vj=.75 Fc=.5 Bv=4.7 Ibv=20.245m)
```

Bipolar transistor model (**NPN** and **PNP**)

The bipolar transistor model used in the PSPICE is a modified Gummel-Poon charge control model. The full nonlinear Gummel-Poon model contains a lot of components and is described with few tenths of parameters. For most applications only few of them are important:

- IS - saturation current of reference junction (1.0E-16, A),
- RB - base region resistance (0, ohms)
- BF - ideal forward current gain Ic/Ib coefficient (100, A/A),
- BR - ideal reverse current gain Ic/Ib coefficient (1, A/A),
- VAF - forward Early voltage (∞, V),
- VAR - reverse Early voltage (∞, V),
- TF - forward transit time (0, s),
- TR - reverse transit time (0, s),
- CJC - collector-base junction capacitance at zero - polarization (0, F),
- CJE - emitter-base junction capacitance at zero - polarization (0, F),

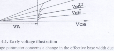


Fig. 4.1. Early voltage illustration

The Early voltage parameter concerns a change in the effective base width due to voltage change at the collector-base junction. This effect is visible on characteristics IC(VCE) as an increase of the collector current IC resulting from increase of VCE at VBE constant. A method of obtaining the Early voltage parameter VAF is shown in Fig. 4.1.

An example of the transistor model statement for the bipolar transistor used in the experiment:

. model QNL npn(BF=220 RB=100 CJC=5PF VAF=150 CJE=12P TF=.5N TR=.5U)

All the **.model ...** statements may be placed in a special text files named ‘libraries’. Such libraries may contain models of elements grouped according to their properties, applications, and so on. For example you may find there libraries of BJTs, FETs, diodes, cores, capacitors, operational amplifiers, switches, and whatever else you could imagine. The files have standard extension **.LIB** and may be searched by the simulator in order to find referenced model name.

Operational amplifier (μA741) model

The operational amplifier μA741 model is an example of the special simulator model construction commonly named a ‘subcircuit’. The subcircuit is typically a more or less complicated network consisting of many active and passive components, which is sewed into the main circuit in few connection points. The subcircuit may have other subcircuits nested. In our case the μA741 model (or ‘macromodel’ as one may say) is quite complicated circuit which is accessible in the library EVAL.LIB included in the PSPICE package. What is inside the model subcircuit is listed in the Appendix A.

The subcircuit calling statement is of the form:

Xname N1 N2 N3 N4 N5 UA741

As it is seen above, the subcircuit name must begin with the prefix X after which come node numbers connecting the subcircuit to the main circuit. The node number order is extremely important because under position of the external node is hidden the function of corresponding position node in the subcircuit. The statement ends with the name of the model.

For the operational amplifier μA741 the meaning of node order is as follows:

N1 - noninverting input,
N2 - inverting input,
N3 - positive power supply,
N4 - negative power supply,
N5 - amplifier output.

4.3.2. The description syntax of the simulated circuit

In order to be properly interpreted by the simulator program, the simulated circuit has to be described with a special language. The statements and commands of this language are placed in an ordinary text file, with standard extension **.cir**. The file may be prepared with any DOS text editor.

The first line of the file is the title line of the job, which is placed in the header of the PSPICE output file and it should contain the project name and the designer's name. The subsequent lines contain the information about component types, their positions in the circuit, the component models. There are special command lines provoking the analysis type and setting the analysis control parameters. Other command lines control the output format. The user, what is especially useful in solving the 'no convergence' problems, can also set some parameters of the simulation algorithm.

Every statement line has to begin from new line and can be continued in following lines, however the continuation sign ' +' must occupy the first character position on each continuation line.

The comment lines must begin with the character '*'. The empty lines are allowed. The input file ends with the command **.end**. Immediately after this line the title line of the next task may placed, and so on.

Writing numbers in the PSPICE input statements simplify special scaling suffixes that may be used instead of standard exponential notation. They are listed below:

tera **T** = 1E+12, giga **G** = 1E+9, mega **MEG** = 1E+6, kilo **K** = 1E+3,
mili **M** = 1E-3, mikro **U** = 1E-6, nano **N** = 1E-9, piko **P** = 1E-12.

The unit symbol (as A - for ampere, V - for volt, etc.) may be omitted. For example, the value of inductor 150mH may be written as 150m, or 15uA as 15u.

Note: All text in the PSPICE input file may be written using either capital or small characters, so that in the above example the text '150mH' may be written as '150MH' or '150M'.

Electronic components description syntax

The component description statement line has the form:

component_name node_numbers value or model_name_reference optional_parameters

The **component_name** must begin with the **type prefix** - a letter unique for given element type (**R** - for resistor, **C** - for capacitor, **L** - for inductor, **Q** - for transistor, etc.) and consist of capital or small letters without spaces. Only the first eight characters of the name are recognized and they must be unique for the given element.
The **node_numbers** is a list of natural numbers (or *net labels* in some versions of PSPICE) assigned to circuit nodes the given element is connected to. At least one of all circuit nodes must be assigned to 0 - this is a reference node with the ground potential. All node voltages in the simulated circuit are calculated in respect to this ground node. The node numers order determines the assumed direction of the current through the element or assumed voltage (or current direction) polarity of independent or controlled sources.
The **value** is simply a value of the element, e.g. 120k (120 kΩ for a resistor).
The **optional_parameters** are any optional parameters allowed for given element.

Examples of elements' descriptions:
A resistor R120 placed between nodes 102 and 0 has resistance of 120kΩ with 5% tolerance and with linear temperature coefficient of 0.001:
R120 102 0 120K DEV 5% TC1=0. 001
A capacitor CE between nodes 12 and 34, having the user model MYCAP has capacitance of 10nF:
CE 12 34 MCAP 10NF
An inductor L105 with the inductance of 250μH is placed between nodes 124 and 0:
L105 124 0 250UH
A transistor Qinput having the model QNL, whose collector is connectd to the node 105, the emitter to the node 109 and the base to the node 134:
Qinput 105 134 109 QNL

Note: The node order for a transistor determines its terminal assignment. The first in order is the collector node, the next is the base node, the third is the emitter node, the fourth (optional) is the substrate node. For diodes as the first node the anode is assumed, as the second - the cathode. Similar convention is applied to other active elements (jfets, mosfets).

Signal sources description syntax

The PSPICE allows two kinds of signal sources: independent sources and controlled ones. Both sources' types may be voltage or current sources. The controlled sources may be controlled with voltage or current and may be linear or nonlinear. The controlled sources are not used in this experiment and are not described here.

The independent voltage source syntax:

Vxxxxxx N+ N- <<DC value> <AC <amp_value <phase_value>>> time_dep_function>

The independent current source syntax:

Ixxxxxx N+ N- <<DC value> <AC <amp_value <phase_value>>> time_dep_function>

Vxxx or **Ixxx:** The name of the independent source must begin with the prefix type letter: **V** - for voltage source or **I** - for current source (there may be small letters used as well).
N+ N- : node numbers the source is connected to. The order of nodes determines the source polarity - the first one is positive, the second is negative. The current sources produce current flowing out of the first node and return current flows in the second node.
DC value (optional): if the DC keyword is specified, the value is used for the DC solution of the circuit and the bias solution for active devices. Typically used for power supply sources declaration.
AC amp_value <phase_value> (optional): if the AC keyword is specified, the amp_value is the amplitude (magnitude) value of the small signal AC source used in the small signal AC analysis. Optionally the initial phase of this source signal may be specified. This source is ignored in other analysis types.
time_dep_function (optional): used for description of time dependent source in large signal transient (**.TRAN**, see below) analysis. There are few predefined time dependent sources in the PSPICE:
- trapezoidal pulse:
PULSE(V1 V2 TD TR TF PW PER)

Fig. 4.3. Trapezoidal pulse

V1 - initial voltage,
V2 - pulse voltage,
TD - delay time (0),
TR - rise time (TSTEP),
TF - fall time (TSTEP),
PW - pulse width (TSTOP),
PER - period (TSTOP).

.TSTEP, .TSTOP , .TRAN command parameters (see below).

- damped sine wave:
SIN(V0 VA FREQ TD THETA)
V0 - DC component voltage,
VA - amplitude of the sine wave,
TD - delay time (0),
FREQ - frequency in Hz(1/TSTOP),
THETA - coefficient of damping in 1/s(0).
The coefficient of dumping may be found from an: THETA = 2 * FREQ * ln(A1/A2), where A1 and A2 are amplitudes of consecutive periods of the wave.

- piece-wise linear approximation:
PWL({Ti, Vi})
Ti, Vi - coordinates of consecutive points of the curve: Ti - time, Vi - corresponding voltage at Ti. There may be any number of data pairs. The statement is used to define signal of any shape (e.g. sawtooth, staircase, triangle. etc.)

Examples of voltage and current sources declarations:
DC voltage source 10V (e.g. power supply):

```
VIN1 102 0 DC 10
```

Small signal AC current sources of 20mA amplitude and initial phase 90°:

```
Iin1 104 106 AC 20M 90.0
```

Large signal sinusoidal source of 120kHz frequency, 2V amplitude and having 3 V DC component:

VIN2 107 0 SIN(3 2 120K)

Rectangular wave current source of 1MHz frequency, 150μA pulse value, initial value of 1mA, delay 150ns and of 70% duty-cycle:

IWE2 145 234 PULSE(1M 150U 150NS 0.0S 0.0S 300NS 1US)

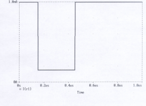


Fig.4.4. Pulse shape declared with PWL form

The shape of the current pulse declared with the PWL form is shown in Fig. 4.4. It has been obtained from the PSPICE as a result of the input file 'test.cir' given below:

```
test
I1 0 1 DC 5m AC 5m Pulse(1m 150u 150n 0 0 300n 1u)
r1 1 0 1
.op
.tran 2n 1u
.probe
```

If you run the test example and carefully analyze the output file you may see some interesting facts. First of all, the 'bias (or operational) point solution', whose results are invoked with the '.op' command, depends only on the current value declared after 'DC' keyword. Secondly, the DC current does not have any meaning for the resulting pulse initial value 1mA - the initial transient solution ignores the 'DC' value using the value declared with the PWL form. The AC declared value is completely ignored, it has a meaning only if the '.AC' command is used invoking a small signal frequency analysis.

4.3.3. The PSPICE command statements controlling simulation process and its output formats.

All PSPICE command statements start begin with the '.' (dot) character. The are placed in the input file with standard extension '.cir'. The output information from the program is placed into a file with standard extension '.out'.

Library declaration statement - declares model library files to be searched by the PSPICE. If path_name is ommited the NOM.LIB is assumed.
. LIB <path_name>

Model declaration statement - defines component model used in simulated problem.
.model model_name model_type_name([optional_ parameter_list])
model_name name has to be unique in entrance - gathering {harvest}. Is influenced {suggests} so that began from letters identifying type model. See sec. 4.3.1.

Tabular listing of output voltages or currents
. PRINT analysis_type out_var .... out_var ....

analysis_type - one of the following:
DC - dc transient characteristic,
AC - frequency domain ac small signal characteristic,
NOISE - noise analysis of the small signal model of the circuit,
TRAN - time domain large signal transient characteristic
out_var - output variable declared in the V(N1,N2) form for voltages, N1 and N2 are node numbers of the observed branch (if one of nodes is omitted, it is assumed 0). For currents the I(comp_name) form is used, where comp_name is the name of components whose current is to be printed.

In a case of an AC analysis the output variable type prefix V or I can be appended with an additional letter qualifying variable type:

R - real part, **I** - imaginary part, **M** - magnitude, **P** - phase, **DB** - 20*log (magnitude).

If the additional letter is omitted, the magnitude is assumed.

Print-plots of output voltages or currents
. PLOT analysis_type out_var [range].... out_var [range]....
The syntax is as for **.PRINT...** command with an exception that after specified out_var its range may be optionally declared. If omitted the full range is assumed. One .PLOT instruction can output up to eight variables. Each graph is assigned a unique character to plot. Where plots are superimposed the x character is printed.
Note. The .PLOT command is nowadays out of use. Instead most PSPICE packages incorporates the PROBE - a powerful graphical postprocessor capable of producing high quality graphs on bubble jet or laser printers.

Temperature declaration
.TEMP t1 t2 ... tn
t1, t2, tn - list of temperatures for which an analysis is to be performed. If .TEMP is omitted in the input file, the PSPICE assumes the temperature specified in OPTIONS.
Note. Most models of active elements have their parameters temperature dependent. But an user should remember that the declared temperature concerns the temperature inside the component,

not the an ambient temperature, which may be quite different. What's more, the PSPICE assumes that all circuit components have the same temperature, what is not necessary true.

Number of columns interpreted in input file and send to screen or printer
.WIDTH [IN=in_val] [OUT=out_val]
in_val may have maximum value 80, **out_val** may take 80 or 133. Default value is 80.

Analysis type control statements

To perform required analysis of the circuit an adequate command should be placed into the input file (*.cir). One or more analysis' types can be specified.

. OP - placing this command in the input file results in printout all detailed data about operational point of the circuit, model parameters of active components, small signal models of active devices and nonlinear controlled sources.

.DC sweep_type nr_of_int source_name init_value last_value incr_value

The command shown above is typically used for calculation of dc transfer characteristic from the inpurt port where the source (voltage or current) is connected to any node or branch in the circuit.
sweep_type - possible values are **OCT** or **DEC**.
nr_of_int - number of intervals per octave or decade,
source_name - name of source to be swept,
init_value - initial value of sweep variale,
last_value - last value of sweep variale,
incr_value - if neither OCT, nor DEC is specified, then **LIN**ear sweep is assumed, and incr_value is its increment.

.AC sweep_type nr_of_int init_value last_value

The '.AC' command invokes the small signal frequency domain analysis. To perform this analysis at least one ac source must exist in the circuit.
sweep_type - possible values are **OCT**, **DEC** or **LIN**,
nr_of_int - number of intervals per octave or decade, if LINear sweep is used then as a nr_of_int a frequency step is used,
init_value - initial value of frequency,
last_value - last value of frequency,

.TRAN TSTEP TSTOP <TSTART <TSTEPMAX>>

The '.TRAN' command invokes the large signal time domain analysis. The analysis is performed with an automatic time step control
TSTEP - print or plot output interval,
TSTOP - stop time of analysis,
TSTART - start time of print or plot output results,
TSTEPMAX - analysis time step limit, default value TSTOP/50.
The analysis always starts at time T=0. If TSTART is declared then results are sent to the output file beginning from this moment.

Program run controls

.OPTIONS par_name[[=value] ...

There are 32 run control parameters. To most important from them belong s:

NOMOD - stops printing models' parameters in output file,

NOPAGE - no paging of output,

OPTS - prints OPTIONS parameters,

ABSTOL=value - defines maximum absolute error of current balance at ciruit node, default value = 1pA,

VNTOL=value - defines maximum relative error of voltage calculations, default value = 1mV,

TNOM=value - changes the nominal temperature for simulation, default value = 27° C.

LIMPTS=value - defines maximum numbber of lines in PRINT or PLOT output, default = 201.

CPTIME=value - defines maximum CPU time for PSPICE run, default = 1E8 s.

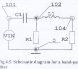


Fig.4.5. Schematic diagram for a band-pass filter

```
Band-pass filter
VIN 101 0 AC 7.5
 C1 101 102 10NF
L1 102 104 250UH
R1 102 0 10K
R2 104 0 1K
.OPT NOMOD NOPAGE
.AC DEC 5 5KHZ 10MegHZ
.PRINT AC V( 101) V( 102) V( 104)
.PROBE
.end
```

Fig. 4.6. Input file for the band-pass shown in Fig.4.5.

4.3.4. Example of the PSPICE input file preparation

As an example of the PSPICE simulator application a simple pass-band filter circuit shown in Fig.4.5 is used. The filter consists of two filters: high-pass filter C1R1 and low-pass filter L1R2 investigated in the Experiment 2, in a chain connection. The filter elements values are as follows: C1=10nF, R1=10kΩ, L1=250mH and R2=1kΩ. The magnitude of the input voltage source VIN amplitude equals 7.5 V and its frequency varies from 100Hz to 10MHz. The signals to be calculated are voltages at nodes 102 and 104 - the outputs of RC and LR filters respectively.

The input file for the example may have a form shown in Fig.4.6. The first line is a title line. The second line declares the input voltage source with amplitude 7.5 V. The next four lines specify the element values. All component lines contain information about the circuit topology - node numbers - that the components are connected to. The '.OPT' line restricts printing of models data and excess paging of the printout. In the '.AC' command there is an instruction to perform a calculation with logarithmically varied frequency spaced 5 intervals per decade. The frequency range extends from 5 kHz up to 10 MHz (note the multiplier suffix 'Meg' in the upper frequency limit specification. The last lines control the presentation of results. The '.PRINT' command produces the tabular print of the frequency characteristic, while the .PROBE command results in generation of special data file (typically in binary format) which is subsequently read by automatically invoked graphic postprocessor PROBE. The PROBE not only can create an elegant graphic plot of the requested variable but may additionally perform a lot of math processing on the data file e.g. averaging, integrating, differentiating, multiplying, dividing, and many more. The input file as in Fig.4.6 should be saved

```
FREQ V( 101) V( 102) V( 104)

5. 000E+03 7. 500E+00 2. 064E+00 2. 064E+00
7. 924E+03 7. 500E+00 3. 106E+00 3. 106E+00
1. 256E+04 7. 500E+00 4. 410E+00 4. 409E+00
1. 991E+04 7. 500E+00 5. 714E+00 5. 711E+00
3. 155E+04 7. 500E+00 6. 689E+00 6. 680E+00
5. 000E+04 7. 500E+00 7. 243E+00 7. 221E+00
7. 924E+04 7. 500E+00 7. 504E+00 7. 447E+00
1. 256E+05 7. 500E+00 7. 613E+00 7. 469E+00
1. 991E+05 7. 500E+00 7. 649E+00 7. 300E+00
3. 155E+05 7. 500E+00 7. 645E+00 6. 850E+00
5. 000E+05 7. 500E+00 7. 616E+00 5. 989E+00
7. 924E+05 7. 500E+00 7. 574E+00 4. 743E+00
1. 256E+06 7. 500E+00 7. 538E+00 3. 408E+00
1. 991E+06 7. 500E+00 7. 517E+00 2. 290E+00
3. 155E+06 7. 500E+00 7. 507E+00 1. 485E+00
5. 000E+06 7. 500E+00 7. 503E+00 9. 477E-01
7. 924E+06 7. 500E+00 7. 501E+00 6. 007E-01
1. 256E+07 7. 500E+00 7. 500E+00 3. 797E-01
```

Fig.4.7. Tabular output from ex41.cir file

in DOS text format under a name e.g. 'b_pass.cir' to the folder where the PSPICE shell file PS.EXE exists. To start the program one should type 'ps b_pass.cir', or only 'ps'. If only 'ps' command is used then the input file name should be opened from the **File** menu of the PSPICE shell. Then from the **Run** menu the option **Simulator** should be clicked. The tabular output invoked by the '.PRINT' command is sent to the text output file 'b_pass.out'. The essential fragment is shown in Fig.4.7.

The graphic output produced by the PROBE postprocessor is shown in Fig.4.8. Three curves are displayed: the input signal, the high-pass filter output and the resulting output from the low-pass filter. The high-pass filter output shows some flat maximum, what is not visible if the high-pass filter is simulated separately. The effect results from the fact that the high-pass part (CR) of the filter is loaded by its low-pass (LR) part.

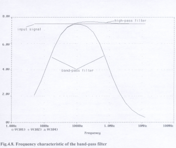


Fig.4.8. Frequency characteristic of the band-pass filter

4.4. Experimental part

Task 4.4.1. Band-pass amplifier

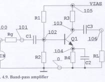


Fig. 4.9. Band-pass amplifier

A schematic diagram of a simple small signal band-pass amplifier is shown in Fig. 4.9. The components' values are given below:
R1 = 620kΩ, **R2** = 1.2MΩ, **R3** = 2kΩ, **R4** = 1.5kΩ, **C1** = 100nF, **C2** = 200μF, **C3** = 100nF,
Q1 - NPN transistor with parameters:

- ideal forward current gain - 220,
- collector-base junction capacitance - 5pF,
- base-emitter junction capacitance - 12pF,
- forward transit time - 0.5ns
- reverse transit time - 0.5us
- forward Early voltage - 150V.

Supply voltage **VZAS** = +12V, resistor **RL** = 1kΩ, resistor **Rg** = 1kΩ.
Input signal - sine wave, 0.1V amplitude.

Create the input file **lpex1.cir** for the above circuit containing:

- title line "Transistor band-pass amplifier *[your team number]*"
- **.model** statement for the transistor Q1,
- commands for the DC operating point data output and small signal AC analysis for 10Hz - 100MHz frequency range,
- PROBE postprocessor invoking command

Write down your input file using the template given below:

```
Band-pass amplifier .........
*
.model .........................................
VIN ..........................................
VZAS ...........................................
*
Q1    .........................................
C1    .........................................
C2    .........................................
C3    .........................................
R1    .........................................
R2    .........................................
R3    .........................................
R4    .........................................
RL    .........................................
Rg    .........................................
*
..........................................
.............................................
................................................
.END
```

Start PSPICE simulator, using its internal editor type-in the created input file, correct errors if needed and run it (use 'RUN/SIMULATOR'). Check results with PROBE. Adjust frequency range to obtain reasonable shape of the frequency characteristic, find lower and upper cut-off frequencies, bandwidth and small signal gain for f = 1kHz, note them in your report

For those who want to know. Double the collector-base capacitance CJC in the transistor model, repeat simulation. Observe and note the result. Next change the C2 value to 100uF and repeat simulation. Observe and note the result. Make a comment.

Task 4.4.2. Simple inverter

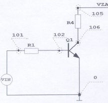


Fig. 4.10. Simple inverter

A schematic diagram of a simple inverter is shown in Fig. 4.10. The components' values are given below:

R1 = 30kΩ, **R4** = 1kΩ,
Q1 - NPN transistor (use previous model),
Supply voltage **VZAS** = +12V.
Input signal - single rectangular pulse, low level 0V, high level 5V, width 1u.

Create the input file **lpex2.cir** for the above circuit containing:

- title line "Transistor inverter *[your team number]*"
- **.model** statement for the transistors Q1,
- commands for transient analysis in 0 - 5us range, max step 50ns,
- PROBE postprocessor invoking command

Write down your input file using the template given below:

```
Transistor inverter
*

.model QNL .......................................

VIN ....................................

VZAS ....................................
```

```
Q1 ........................................
R1 ........................................
R4 ........................................
..........................................
..........................................
.END
```

Start PSPICE simulator, using its internal editor type-in the created input file, correct errors if needed and run it (use 'RUN/SIMULATOR'). Check results with PROBE. Observe input and output waveforms. Draw the waveforms, mark delay time, rise time, recovery time, fall time, note them in your report

For those who want to know. Add 3pF capacitor across collector-base junction (simulating spread capacitance). Observe and note the result. Make a comment.

Task 4.4.3. Operational amplifier

A schematic diagram of an inverting operational amplifier is shown in Fig. 4.11. The components' values are given below:

R1 = 1kΩ, **R2** = 10kΩ, **RL** = 10kΩ

XOP - operational amplifier of uA741 type,

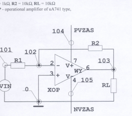


Fig. 4.11. Inverting operational amplifier

Supply voltage **PVZAS** = +15V, **NVZAS** = -15V
Input signal - sine wave, 0.1V amplitude.

Create the input file **lpex3.cir** for the above circuit containing:

- title line "Inverting operational amplifier *[your team number]*"
- library declaration for EVAL.LIB,
- commands for the DC operating point data output and small signal AC analysis for 10Hz - 200kHz frequency range,
- PROBE postprocessor invoking command

Write down your input file using the template given below:

Inverting operational amplifier

*

....

VIN ....

VPZAS ....

VNZAS ....

XOP ....

R1 ....

R2 ....

RL ....

....

....

....

.END

Start PSPICE simulator, using its internal editor type-in the created input file, correct errors if needed and run it (use 'RUN/SIMULATOR'). Check results with PROBE. Draw the frequency characteristic, mark bandwidth limits, find small signal gain for f = 1kHz, note them in your report. Modify sine input signal for large signal time domain analysis (f = 1kHz). Using transient analysis mode find input signal amplitude for which the output signal begins to be distorted. Change the load resistor value to 100Ω, repeat last simulation. Make a comment.

4.5. Supplementary literature

Michał Ramotowski

ELECTRONIC LABORATORY

Instruction sheet No. 5

AC VOLTAGES AMPLIFICATION WITH TRANSISTORS

(The text is freely based on the laboratory sheet:
'Wzmacnianie napięć zmiennych
za pomocą tranzystorów'
with the kind permission of the author
Bogusław Kalinowski)

Experiment 5

AC VOLTAGES AMPLIFICATION WITH TRANSISTORS

Goals of the experiment:
- familiarize with simplest active elements - bipolar and junction field effect transistors (JFET),
- presentation of application of these components for voltage amplification.

Scope of the experiment:

1. Introduction to experiment (overview of transistors: bipolar and JFET).

2. Static transistors' characteristics measurement
2.1. Measurement of the input and transfer characteristic of the bipolar transistor.
2.2. Measurement of the output characteristic of the bipolar transistor.
2.3. Measurement of the transfer characteristic of the JFET transistor.
2.4. Measurement of the output characteristic of the JFET transistor.

3. Observation of the transfer characteristic of the bipolar transistor amplifier.

4. Measurement of the frequency characteristic of the bipolar transistor amplifier.

Equipment on the laboratory bench:

M-3 - laboratory module for measurement of transistor static characteristics and a transistor amplifier,
MX-620 - universal digital meter,
OS-9040 - 2 channel oscilloscope,
MX-9000 - multifunction device,
Resistors - 100 Ω, 1 kΩ, 10 kΩ, 1 MΩ,
Connection cables: BNC-BNC - 3 pcs, BNC-banana - 1 pcs, banana-banana - 4 pcs.

5.1. Overview of the active elements

5.1.1. Bipolar transistors

Transistors are simplest twoport electronic elements designed for construction of signal power amplifying circuits. From a principle of operation point of view there are two main groups of transistors: bipolar and field effect.

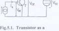


Fig.5.1. Transistor as a current amplifier

Basically, the bipolar transistors may be considered as a current amplifiers, as shown in Fig. 5.1. The transistor output circuit current I_C is controlled by the base current IB through a nearly linear dependency

$$I_C = h_{FE} \cdot I_B \qquad (5.1)$$

where: h_{FE} , sometimes denoted as h_{21E} or β_F, is called as a static current amplification factor. The formula (5.1) mentioned above is 'nearly' linear because the h_{FE}

parameter depends in some way on the collector current I_C. The h_{FE} parameter has rather wide tolerance range, and depends heavily on temperature. Then design work on the bipolar transistor amplifier is not a trivial task and a simulator program, like PSPICE or other, may give a great help. The collector current I_C dependency on the base-emitter voltage V_{BE} is of the form

$$I_C = I_S\,[\exp(V_{BE}/V_T) - 1] \qquad (5.2)$$

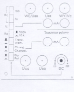
where: $V_T = kT/q = 25.8$ mV at T = 20°C, I_S - saturation current.

The input characteristic of the bipolar transistor is then an exponential function. Everything what has been said above is valid for so called 'active' region of the transistor operation. Outside the active region you may find a 'cut-off' region where the collector current I_C does not flow, and a 'saturation' region where the collector current value is determined by an external circuitry. Bipolar transistors are of two types: n-p-n and p-n-p, and, actually, quantitatively most of them are manufactured on the base of silicon, as a discrete components, or as a building parts of integrated circuits (IC's).

5.1.2. Field-effect transistors (FET)

There are two main groups of FETs: a junction FET, and an insulated gate FET (commonly named 'MOSFET' - Metal-Oxide-Semiconductor-Field-Effect-Transistor, or shortly 'MOS'). The junction FETs (JFETs) are mainly used a discrete elements and in an analog integrated circuits, the MOS types are basic components of all digital ICs.

The FETs are elements of the transconductance type, i.e., their output current is controlled by the gate-source voltage. The leakage gate current is of order of nanoamps (JFET) or picoamps (MOS) and, except of very special instrumentation cases, is negligible. The FETs may be of *n*-channel, or *p*-channel type. The channel may be built-in (depletion type), or induced (enhanced type) - each of them of the of *n*-, or *p*- type. Most applied MOSFETs are of enhanced type, commonly named PMOS, or NMOS, after the type of the drain current carrients.



5.2. MEASUREMENT MODULE M-3

The module contains the bipolar transistor with switchable resistors in the base and collector circuits, the RC transistor voltage amplifier, the JFET transistor, a set of sockets and switches that help to configure the measured circuit. The front panel of the module is shown in Fig.5.2 and its schematic diagram in Fig.5.3.

The RC voltage amplifier with a bipolar transistor is shown in Fig.5.3 as a block marked 'WZM'. The schematic diagram of this amplifier is shown in Fig.5.4.

The effective voltage gain of this amplifier for midfrequency range may be expressed as :

$$|k_{uu}| = |V_{wy}/E_g| = R_L\, g_m\, [R_{we}/(R_{we} + R_G)] = R_L\, g_m,$$

$$\text{for } R_G \ll R_{we} \qquad (5.3)$$

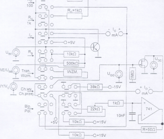


Fig. 5.3. Schematic diagram of the measurement module M-3

where:
$R_L = R_C \parallel R_{load}$,
$g_m = I_C/V_T$ - the transconductance at the operational point of the (bipolar) transistor I_C,
R_{we} – input resistance of the amplifier:

$$R_{we} = r_{wetrans} \parallel R_B, \quad r_{wetrans} = \beta_F V_T / I_C, \quad R_B = R_{B1} \parallel R_{B2}$$

R_G = resultant internal resistance of a signal source:

$$R_G = (R_{wygen} + R_g)$$

If a FET transistor is installed in the module the basic expression for effective voltage gain remains the same. Of course, g_m is defined in the other way than above:

$$g_m = \Delta I_{DS} / \Delta V_{GS}$$

where: ΔI_{DS} is a change of drain-source current I_{DS} caused by ΔV_{GS} change of gate-source voltage V_{GS}.

Besides, for the FET transistor field working as a small signal amplifier, i.e. with gate-drain and gate source junctions reverse polarized, its input resistance $r_{wetrans}$ is of order of gigaohms so may be treated as an open circuit.

5.3. EXPERIMENTAL PART

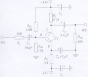


Fig.5.4. Schematic diagram of the voltage amplifier with a bipolar transistor inside the M-3 module (the WZM block)

5.3.1. Measurements of static characteristics of transistors

5.3.1.1. Measurement of input and transfer characteristic of the bipolar transistor

a) Connect measurement circuit as in Fig.5.5. using:
- the measurement module M-3 configured with the bipolar transistor, resistors $R_B = 500k\Omega$, $R_C = 100\ \Omega$.
- two multimeters as voltmeters V1 and V2.

b) Measure static characteristics: $I_C = f(V_{BE})$, $I_C = f(I_B)$, $I_B = f(V_{BE})$.
The method: for some values (say e.g. 10) of V_{BB} voltage (set with the potentiometer marked as 'DC') measure V_{BB}, V_{BE}, V_C (collector voltage in respect to the ground) and place them in the previously prepared table. Calculate I_B and I_C using formulas:

$V_{RB} = V_{BB} - V_{BE}$;
$I_B = I_{RB} - I_{V2} = V_{RB}/R_B - V_{BE}/10^7$; $\frac{V_{CE}}{R_B - V_{BE}} = I_B$
$V_{RC} = V_{CC} - V_C$; $I_C = V_{RC}/R_C$ 500kΩ
$I_C = \frac{V_{CC}}{R_C}$

Calculated values place in the table.

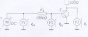


Fig.5.5. Schematic diagram of the circuit for measurement of input and transfer characteristics of the bipolar transistor

c) Draw measured characteristics on separate graphs.
Note! Voltages VBB and VC measure using the same voltmeter V1 reconnecting its 'hot' terminal. During measurements of static characteristics the voltmeter V2 ***may not*** *be disconnected.*

5.3.1.2. Measurement of the output characteristic of the bipolar transistor

a) Modify the measurement circuit in Fig.5.5 in accordance with the schematic shown in Fig.5.6. The collector voltage V_C is set with the potentiometer marked as 'DC' over the range of 0 to +10V. The resistor RB stays without changes ($R_B = 500k\Omega$).

b) Measure the collector current I_C dependence on the collector – emitter voltage V_{CE} for the base current value fixed. Results of measurements place in the previously prepared table. Estimate the base current value of the transistor according to formula: $I_B = (V_{BB} - V_{BE})/R_B$, where V_{BE} is to be measured at the socket 'U$_{BE}$" for I_C corresponding to $V_{CE} = 5$ V. Calculated I_B value place in the

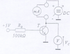


Fig.5.6. Schematic diagram of the circuit for measurement of the output characteristic of the bipolar transistor

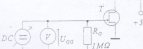


Fig.5.7. Circuit diagram for measurement of the transfer characteristic of the FET transistor

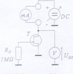


Fig.5.8. Measurement of the output characteristic of the JFET transistor

headline of the table.

c) Draw the output characteristic $I_C = f(V_{CE})$ of the bipolar transistor.

5.3.1.3. Measurement of transfer characteristic of the JFET transistor

a) Connect measurement circuit as in Fig.5.7 using:
- the measurement module M-3 configured with the JFET transistor and pressed switch 'Ch.wy/Ch.przej.' -
two multimeters – one as mA and one as V.

b) Measure static characteristic: $I_D = f(V_{GS})$. Results of measurements place in the previously prepared table and draw the graph.

c) Find the slope of the drain current characteristic $g_m = \Delta I_{DS}/\Delta V_{GS}$ at $V_{GS} = -0.5V$.

5.3.1.4. Measurement of the output characteristic of the JFET transistor

a) Connect measurement circuit as in Fig.5.8.

b) Measure the drain current dependence $I_D = f(V_{DS})$. The V_{DS} voltage measure at socket U_{DD} on the front panel of the M-3 module. Results of measurements place in previously prepared table, and then draw the graph.

5.3.2. Observation of the transfer characteristic of the RC amplifier with a bipolar transistor

a) Connect measurement circuit as in Fig.5.9. Set the oscilloscope to the XY mode (time-base speed selector in position XY). Set the function generator to produce a triangle wave with parameters: lower peak value 0 V, peak-to-peak amplitude of such value that the transistor is not deeply saturated (all the transfer characteristic should be well visible. Frequency of the signal of order of a few hundred Hz, $R_B = 10$ kΩ, $R_C = 1$ kΩ.
b) Sketch the displayed waveform. Mark regions where the transistor is cut-off, active or saturated.

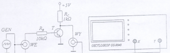


Fig.5.9. Circuit for observation of the transfer characteristic of the RC amplifier

5.3.3. Measurement of the frequency characteristic of the RC amplifier with a bipolar transistor

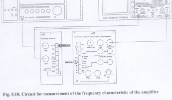


Fig. 5.10. Circuit for measurement of the frequency characteristic of the amplifier

a) Connect measurement circuit as in Fig.5.10. Signal from functional generator should be a sine wave. Adjust the signal amplitude to obtain possibly large undistorted output signal.

b) For some values of signal generator frequency selected in a range 100 Hz - 1 MHz (use frequency meter to check), measure the peak-to-peak value of output signal amplitude and note it in the previously prepared table. The input amplitude should be kept constant (observe it on the oscilloscope channel 1), its value place in the header of the table.

c) Calculate the voltage gains of the amplifier for linear and log scales and write it into the table.

d) Draw the voltage gain G_v [dB] of the amplifier versus frequency i.e. its amplitude characteristic.

e) Write a comment what elements of the amplifier are responsible for the fall of amplification at low frequencies, and what - at high ones.

Michał Ramotowski

ELECTRONIC LABORATORY

Instruction sheet No. 6

OPERATIONAL AMPLIFIER AND NEGATIVE FEEDBACK

(The text is freely based on the laboratory sheet:
'Wzmacniacz operacyjny i sprzężenie zwrotne',
with the kind permission of the author
Bogusław Kalinowski)

Experiment 6

OPERATIONAL AMPLIFIER AND NEGATIVE FEEDBACK

Goals of the experiment:

- presentation of an integrated circuit (IC) used for electric signals amplification,
- practical application of negative feedback theory.

Scope of the experiment:

1. Observation of signal at the operational amplifier ('opamp') output in a case of feedback loop opened.

2. Observation of signal at opamp output in a case of feedback loop closed.

3. Measurement of amplifier parameters in inverting configuration.
    3.1. Measurement of DC voltage gain.
    3.2. Measurement of amplitude characteristic.
    3.3. Measurement of maximum speed of output voltage change.

4. Measurement of amplifier parameters in noninverting configuration.
    4.1. Measurement of DC voltage gain.

Equipment on the laboratory bench:

M-4 - laboratory module of an operational amplifier,
MX-620 - universal digital meter,
OS-9040 - 2 channel oscilloscope,
MX-9000 - multifunction device,
Connection cables: BNC-BNC , BNC-banana, banana-banana.

6.1. RÉSUMÉ OF THEORY

6.1.1. Operational amplifier

An operational amplifier ('opamp') is an active electronic component typically used for amplification of electric signals. Nowadays it is manufactured only in form of integrated circuit and has three active signal terminals: two inputs - inverting and noninverting, and one output. Its graphic symbol is shown in Fig.6.1. The output signal of the opamp depends on input signals according to the formula:

$$V_{out} = A(V_{in(+)} - V_{in(-)}) \qquad (6.1)$$

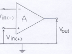


Fig.6.1. Graphic symbol of an operational amplifier

The differential signal gain A (a differential signal is a difference between signal at invering and noninverting inputs of the opamp) is typically large, and for most opamps exceeds 10^4 V/V. The gain of a common signal (this is part of signal, which changes similarly on both inputs of the opamp) should be possibly close to zero. In a technical data of opamps instead of a common signal gain is given a value of relation of gains for differential and common signal. This relation is called as coefficient of suppression of common signal and denoted as **CMRR** (Common Mode Rejection Ratio).

The important parameter of an opamp is the frequency, for which the differential gain magnitude equals unity. Such a frequency is denoted as f_T, or GBW, and is called as gain-bandwidth product or unity gain frequency.

In a design practice a few more opamp parameters are used:
- maximum speed of output voltage change,
- admissible values of supply voltages (usually operational amplifiers are powered symmetrically)
- admissible values of voltages: differential and common,
- input offset voltage,
- input bias current and input offset current,
- output voltage swing.

An operational amplifier only in special situations works in such a way that signals on its inputs are independent from its output signal. Such a configuration is called as "open feedback loop". The normal way of the operational amplifier application is "closed feedback loop" configuration, i.e. there is a connection between output and inverting input, as in Fig.6.2. The circuit in Fig.6.2. is a voltage amplifier with gain (for very low frequencies) given by expression:

$$|k_v| = |V_{out}/V_{in}| = R_2/R_1 \qquad (6.2)$$

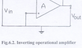


Fig.6.2. Inverting operational amplifier

One should note that amplification of this voltage amplifier, at least for DC voltages and low frequency AC voltages practically does not depend on the gain A of the operational amplifier. However, its upper limiting frequency depends on frequency f_T of the operational amplifier according to the following expression (for $|k_v| >> 1$):

$$f_{3dB} \approx f_T / |k_v| \qquad (6.3)$$

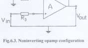


Fig.6.3. Noninverting opamp configuration

Let's turn attention that we have so-called band-to-gain exchange, characteristic for one-pole circuits with feedback - if we increase the amplifier gain then the bandwidth will be proportionally decreased.

The third very important parameter of the discussed voltage amplifier is the maximum speed of the output voltage change. This speed is limited by internal construction of the operational amplifier IC and results in two effects: at first - the higher the frequency the smaller the undistorted output amplitude of the sinusoidal signal, at second - the switching speed of the opamp in nonlinear applications is very limited.

The circuit in Fig.6.2 has yet one interesting feature: the potential of the inverting input is almost equal zero (because due to high value of the gain A the differential input signal is very small and the noninverting node is grounded). Then this inverting node is called as 'virtual ground'. So whole input voltage Vin appears on the resistor R_1 what becomes an input resistance of the amplifier. The current flowing through R_1

$$I_{R1} = V_{in}/R_1 \qquad (6.4)$$

is equal to the current flowing through R_2 because no current flows into the input of the operational amplifier. It directly leads to the expression (6.2).

After modification of the circuit, as in Fig.6.3, the amplifier becomes a noninverting one, i.e. it does not invert the signal phase. Its voltage gain (for DC and very low frequencies) is given by expression:

$$||k_v| = |V_{out}/V_{in}| \approx R_2/R_1 + 1 \qquad (6.5)$$

The bandwidth and maximum speed of the output voltage change for this configuration is practically the same as for inverting configuration, however its input resistance is very large.

6.1.2. Negative feedback

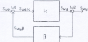


Fig.6.4. Feedback principle

The circuit constructed in such a way that there is a signal back-flow from its output to its input is called as a circuit with *feedback*. For a purpose of analysis, such a circuit may be considered as connection of two blocks shown in Fig.6.4. These two blocks are not loading each other. One of the blocks is called as a four-terminal, or two-port amplification block, and is marked with the symbol *k*. The second block is usually called as a four-terminal, or two-port feedback block and is mark with the symbol *β*. The introduced symbols are also used as proportionality factors in relations between output and input signals.

$$S_{wy} = k\,S_{we,k}\,; \qquad S_{wy,\beta} = \beta\,S_{wy} \qquad (6.6)$$

The input node W1 is a summing node:

$$S_{we} + S_{wy,\beta} = S_{we,k} \qquad (6.7)$$

Combining (6.6) and (6.7), and denoting with the symbol k_f the resulting gain (or more generally - the transmittance) of the feedback system, we will receive:

$$k_f = S_{wy}/S_{we} = k/(1 - k\beta) \qquad (6.8)$$

The expression (6.8) is a very general one. The appearing quantities k, β, k_f can be voltage gains, current gains, transimpedances or transresistances depending on input and output signal types of both blocks.
If $|1 - k\beta| > 1$, then $|k_f| < |k|$, and we call the feedback 'negative'. If an opposite case we have a positive feedback. The circuits with positive feedback, for which $k\beta = 1$ or $|k\beta| > 1$, they can produce selfoscillations, or they can be generators.

For negative feedback circuits usually $|k| >> 1$, $|\beta| < 1$, $- k\beta >> 1$. Then $k_f \approx 1/\beta$ so the circuit transmittance almost fully depends on twoport β.

Let's apply the introduced theory to the circuit in Fig.6.3. Her we have: k = A, $\beta = R_1/(R_1+R_2))$. The input signal V_{in} is connected to the noninverting input of the operational amplifier, and the feedback signal βV_{out} to the inverting one. Then the sign of β should be changed from positive to negative. Finally, having in mind that for low frequencies $|A| >> 1$, we obtain:

$$k_f = k_u = \frac{A}{1 - A(-\beta)} = \frac{A}{1 + A\beta} \approx \frac{1}{\beta} = \frac{R_1 + R_2}{R_1} = 1 + \frac{R_2}{R_1} \qquad (6.9)$$

6.1.3. The measurement module M-4

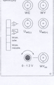


Fig. 6.5. Front panel of the M-4 module

The M-4 module is used for investigation of the operational amplifiers. The module front panel is in Fig.6.5 and its schematic diagram in Fig.6.6. The module contains an universal operational amplifier of type µA741, the feedback twoport with resistors 1kΩ and 10kΩ, the 10kΩ potentiometer for balancing the opamp, the operation mode switch, the DC source adjustable from 0 to 1.2V, and a set of BNC sockets. Pressing the first button from the top connects both opamp input to the ground and opens the feedback loop. Such configuration is used for measurement of the input offset voltage, or to balance the amplifier.
Pressing the second button (from the top) connects the opamp output to its inverting input and the noninverting input to the ground. The output voltage is then equal to the input offset voltage (why?). The third button switches the amplifier to the inverting mode with -10V/V gain (see Fig.6.6). The fourth button sets the noninverting mode with the gain of +10V/V.

6.2. EXPERIMENTAL PART

6.2.1. Observation of the output signal of the operational amplifier with the feedback loop open

a) Connect both inputs of the operational to the ground, leave the feedback loop open.
b) Measure the output voltage of the operational amplifier with a DC voltmeter.

c) Comment, why the output voltage of the operational amplifier is not equal to 0. Is it possible to correct it?

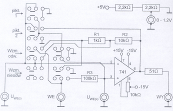


Fig. 6.6. Schematic diagram of the M-4 module

6.2.2. Observation of the output signal of the operational amplifier with the feedback loop closed

a) Press the second from the top button on the front panel of the M-4 module..
b) With a help of Fig.6.6 draw the resulting configuration of the operational amplifier.
c) Measure (with a DC voltmeter) the output voltage of the operational amplifier.
d) Write, what the measured value results from.

6.2.3. Measurement of amplifier parameters in inverting configuration

6.2.3.1. Measurement of the DC gain

a) Press a button on the front panel of the M-4 module to select the inverting configuration of the opamp.. Connect the DC voltage source of value about 0.4 V to 1.2 V to the amplifier input. Use a T-connector to facilitate measurement voltage of the loaded source (comment it).
b) Measure voltages:
- at the amplifier input V_{we},
- at the virtual ground node $V_{wa(-)}$,
- at the amplifier output V_{wy}.

Note results.
c) Calculate the real value of the voltage gain $|k_v| = |V_{wy}|/V_{we}$. Note and comment results.
d) Switch off the power from the M-4 module. Measure and note resistance of the resistor R_1. Use measured k_v and R_1 values for calculation of the resistor R_2 value.

e) Calculate the input resistance of the circuit using the formula: $R_{we} = V_{we}/I_{we}$, where $I_{we} = (V_{we} - V_{we(-)})/R_1$. Comment the result.

6.2.3.2. Measurement of the amplitude characteristic of the amplifier vs. frequency

a) Replace the DC voltage at the input of the inverting amplifier (see circuit in Fig.6.2) with sine wave signal source and the input of the one channel of the oscilloscope. The output signal of the amplifier connect to the second channel of oscilloscope. Set the signal amplitude from the generator (for small frequency, for example 1 kHz), to obtain at the amplifier output an undistorted signal of about 2.5 Vpp (peak-to-peak).
b) Measure the amplitude characteristic of the amplifier. Results of measurements and calculations place in a table.
c) Draw the magnitude of the gain vs. frequency using log-log scale.
d) Estimate from the graph and find experimentally the upper limiting frequency of the amplifier.

6.2.3.3. Measurement of the maximum speed of the output voltage change

a) Using the same measuring circuit as before set the signal generator to rectangular waveform.
b) Increase the generator signal until the output signal from the amplifier starts saturate.
c) Measure the speed of leading and trailing edges of the amplifier output signal. Results of measurements place in a table.

6.2.4. Measurement of the noninverting voltage amplifier parameters

6.2.4.1. Measurement of the DC gain

a) Press a button on the front panel of the M-4 module to select the noninverting configuration of the opamp.. Connect the DC voltage source of value about 0.4 V to 1.2 V to the amplifier input. Use a T-connector to facilitate measurement voltage of the loaded source.
b) Measure voltages:
- at the amplifier input V_{we},
- at the noninverting input $V_{we(+)}$,
- at the amplifier output V_{wy}.

Note results.
c) Calculate the real value of the voltage gain $|k_u| = |V_{wy}|/V_{we}$. Note result.
d) Switch off the power from the M-4 module. Measure and note resistance of the resistor R_3.
e) Calculate the input resistance of the circuit using the formula: $R_{we} = V_{we}/I_{we}$, where $I_{we} = (V_{we} - V_{we(+)})/R_3 - V_{we(+)}/10^7$. Comment received result.
f) Does the bias current of the operational amplifier have any meaning on the result of above calculations?

Michał Ramotowski

ELECTRONIC LABORATORY

Instruction sheet No.7

GENERATORS OF SINUSOIDAL AND RECTANGULAR VOLTAGES

(The text is freely based on the laboratory sheet:
'Generatory napięć sinusoidalnych i protokątnych',
with the kind permission of the author
Bogusław Kalinowski)

Experiment 7

GENERATORS OF SINUSOIDAL AND RECTANGULAR VOLTAGES

Goals of the experiment:

- presentation of some methods of periodical signals generation.

Scope of the experiment:

1. Measurement of amplitude characteristic of a divider with parallel resonance circuit.
2. Observation of voltages in a sinusoidal oscillations generator with an LC circuit.
3. Observation of voltages in a quartz generator with a CMOS inverter.
4. Observation of voltages in a relaxation generator with an operational amplifier.

Equipment on the laboratory bench:

M-5 - laboratory module with generators,
MX-620 - universal digital measure;
OS-9040 - 2-channel oscilloscope,
MX-9000 - multifunction device,
Measurement and supply cables,
Oscilloscope probes - 2 pcs.

7.1. RÉSUMÉ OF THEORY

7.1.1. Parallel resonance circuit

Parallel resonance circuit consists of an inductor L in parallel connection with a capacitor C, as in Fig.7.1. The energy of alternating current flowing through the inductor and capacitor is partially lost (thermal energy, energy of radiation, etc.). This losses of energy, appearing mainly in the coil, they are modeled by conductance G parallel to the coil L, as in Fig.7.1, or by a resistance in series connection with the coil L (this case is not shown). Here only the parallel model is discussed.
Note! Both models are not equivalent and none of them does completely describe the real resonant circuit.

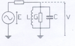


Fig.7.1. Parallel resonant circuit

The voltage across the resonant circuit connected as a part of a voltage divider, as in Fig.7.1, is given by the formula:

$$V = E\frac{Z}{Z+R} \tag{7.1}$$

whereas the impedance of this circuit is given by:

$$Z = \frac{1}{j\omega C + G + \frac{1}{j\omega L}}; \qquad \omega = 2\pi f \tag{7.2}$$

The impedance magnitude is equal to:

$$|Z| = \frac{1}{\sqrt{G^2 + \left(\omega C - \frac{1}{\omega L}\right)^2}} \tag{7.3}$$

Under circumstance: |Z| << R, the formula (7.1) may be simplified as follows:

$$V = \frac{E}{R} Z = IZ \tag{7.4}$$

If the condition |Z| << R is fulfilled, then the current I supplied by the source E is almost independent on the magnitude |Z|. Then measuring the voltage V vs. frequency we can obtain the frequency characteristic of |Z|. The frequency characteristic of the impedance magnitude |Z| is shown in Fig.7.2. One may observe from this graph that there is distinct emphasizing of signals with frequencies close to some special frequency f_0, defined as follows:

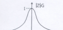


Fig.7.2. Magnitude of the parallel resonant circuit impedance vs. frequency

$$f_0 = \frac{1}{2\pi\sqrt{LC}} \tag{7.5}$$

The frequency f_0 is called as a *resonant frequency* of the circuit, at this frequency the imaginary part of the circuit complex admittance Y equals 0:

$$B(\omega) = \omega_0 C - \frac{1}{\omega_0 L} = 0 \tag{7.6}$$

The resonant frequency is an essential parameter of the resonant circuit.

The second most important parameter of resonance's circuit is its *quality factor*, *Q factor*, or *factor of merit*, defined as a relation of energy stored in the circuit (the coil and the capacitor are energy storing elements) to the energy lost during period of signal of frequency f_0. The *quality factor* is usually denoted with the symbol Q for the discussed model of circuit is expressed as follows:

$$Q = \frac{1}{\omega_0 LG} = \frac{\omega_0 C}{G} = \frac{R_d}{\omega_0 L} = R_d \omega_0 C \tag{7.8}$$

where: $$\omega_0 = 2\pi f_0; \qquad R_d = \frac{1}{G} \tag{7.9}$$

Note:
a) Rd is called as a dynamic resistance of a parallel resonant circuit.
b) The 'sharpness' of the resonance curve depends on the Q-factor. The larger Q, the more "sharp' (selective) the resonance curve is.
c) For the resonant frequency the magnitude of current in the coil and in the capacitor is Q times larger the external current supplied from the signal source.
d) For the resonant frequency the parallel resonant circuit is seen from the outside as a resistance (there is no phase shift between current and voltage); for frequencies f < f_0 the circuit is of inductive nature, and of capacitive character for f > f_0. The resonant circuit is mainly used for selection of signals of frequency near to f_0 and to attenuate signals of frequency far away from f_0.

7.1.2. Notes on linear theory of generation

If in a circuit with a feedback (see Experiment No.6) a signal returning from the output to the input has the same amplitude and phase as the input signal, then the input signal can be removed without stopping the oscillations. The amplitude of the oscillations will be dependent on the amplitude of temporary stimulating signal, and their frequency - on the feedback loop selectivity. On the base of linear theory of generation this is the case when twoports k and β are linear and kβ = 1. However, such a circuit can not be realized practically because a smallest change of any circuit parameter has to cause either decay of oscillations or rising them to infinity.

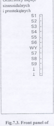


Fig.7.3. Front panel of the M-5 module

7.1.3. Notes on nonlinear theory of generation

The real generator begins oscillations itself after switching on the power supply. Typically, the generator circuit includes an amplifier with a positive feedback, such that for the frequency f_0, or for a whole bandwidth, the product $k\beta$ is a real number, and that for small signal amplitudes the inequality $k\beta > 1$ is valid. Therefore the signal amplitude returning from the amplifier output to its input is larger than the input signal amplitude, what makes the growth of the self-oscillations amplitude.

In circuits with a selective feedback loop as the amplitude is raising the product $k\beta$ gets smaller, until kβ = 1 and the amplitude become stable. Depending on the generator circuit configuration, the value of the k factor may be function of the amplitude (the most frequent case), or the value of the β factor (e.g. in a generator with Wien bridge).

In circuits with a wide-band feedback loop (relaxation type generators) the product $k\beta$ becomes much larger than unity when the circuit switches from one state to the other. During the rest of the cycle $k\beta = 0$ and the energy is stored or dispersed in reactances of the circuit

7.1.4. Measurement module LPE/M-5

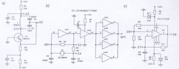


Fig.7.4. Schematic diagrams of the generators

The front panel of the LPE/M-5 module is shown in Fig.7.3, whereas schematic diagrams of the generators are placed in Fig.7.4. The module contains three different generators:

a) sine wave LC generator with a capacitive divider (Colpitts' type),
b) quartz generator with a CMOS inverter,
c) relaxation generator with an operational amplifier.

7.1.4.1. LC generator (Fig.7.4a)

The active element of the LC generator is a bipolar transistor in resonant amplifier circuit. The collector load of the transistor is a parallel resonant circuit with elements L, C2, C3. The feedback signal is fed to the emitter through the capacitive voltage divider C2, C3. The transistor emitter here is the input of the resonant amplifier in a common base configuration. The element values of the resonant circuit and start operating point of transistor were chosen to obtain open loop gain for small signal amplitudes greater than 2 V/V. Such a measures assure a reliable start of oscillations. Frequency of the output signal is close to resonant frequency of the resonant circuit.

The process of reaching steady state of oscillations relies on shifting the transistor bias point in a direction toward collector current cut-off when the signal amplitude at the emitter input rises. This way of amplitude limiting is called a limiting on input nonlinearity of the amplifier. The collector current flow only through a part of signal period and it has a shape of a pulse being a part of the sine wave. The collector current spectrum contains a lot of harmonics of the fundamental frequency f_0. The selective resonant circuit shorts current harmonics with frequencies other than f_0 so the collector voltage has a shape very close to the sine wave.

7.1.4.2. Quartz generator (Fig.7.4b)

The quartz generator investigated in the experiment is also of Colpitts type (the capacitive divider is formed with capacitors C5 and C6). The resonant circuit consists of capacitors C5, C6 and a quartz resonator Q. The quartz resonator works here as the frequency dependent inductance. At any changes of generator frequency the quartz inductance changes its value toward reduction of this frequency change. The frequency comes back to its steady state value (of course there is left a very small frequency error).

The resistor connected between output and input of the inverter settles the inverter operating point somewhere in a middle of its active transfer characteristic where the voltage gain reaches its maximum. The resistor placed between the inverter output and the quartz resonator causes that the resonant circuit supply of this generator is of current character type, not of the voltage type. Let's note that the inverter output is rather of low resistance type, what is not suitable to supply a parallel resonant circuit, even in such impedance transforming configuration.

In this circuit the limiting of the oscillations' amplitude happens as a result of the inverter transfer characteristic nonlinearity, or more precisely, as a result of both-side limitation of its output voltage due to limited dynamic range of the inverter output stage.

7.1.4.3. Relaxation generator (Fig.7.4c)

This generator uses an idea of switching asymptotic value of voltage across periodically recharged capacitor. The generator output signal has a form of rectangular wave. The circuit works as follows.

Let the output voltage of the operational amplifier is almost equal to $V_{zas(+)}$. The capacitor C8 is charged from the voltage source $V_{zas(+)}$ through the resistor R11 and the channel resistance of the MOS transistor placed between the output of the operational amplifier and the voltage source $V_{zas(+)}$. At this time to the noninverting input of the operational amplifier is fed with a part of the amplifier output voltage through the voltage divider R7, R9. When the capacitor voltage reaches the same level as at the noninverting input the operational amplifier switches rapidly to its opposite state, where the MOS transistor connecting the amplifier output and $V_{zas(+)}$ cuts-off and the MOS transistor connecting the amplifier output and $V_{zas(-)}$ starts conducting (the speed of switching is limited by the maximum speed of output voltage change, i.e. a slew-rate parameter). The output voltage becomes close to $V_{zas(-)}$. The capacitor C8 discharges to $V_{zas(-)}$ source through the resistor R11 and the channel resistance of the amplifier output transistor. Switching the output voltage results in the step voltage change at the noninverting input of the operational amplifier. This is a new asymptote that the capacitor voltage is now going to reach. At the moment when the capacitor voltage reaches the same level as at the noninverting input the operational amplifier switches rapidly to its opposite state and the new work cycle begin.

For $V_{zas(+)} = V_{zas(-)}$ the generator signal period is given by:

$$T = 2 \cdot R11 \cdot C8 \cdot \ln\frac{1+\alpha}{1-\alpha} \tag{7.10}$$

where

$$\alpha = \frac{R7}{R7+R9} \tag{7.11}$$

7.2. EXPERIMENTAL PART

7.2.1. Measurement of the amplitude characteristic of the attenuator with the parallel resonance circuit

a) Using M-1 measurement module connect a circuit as in Fig.7.5. Select the components' values: L=250 µH, C=10 nF, R=100 kΩ, GEN - sine-wave generator with the output signal voltage E ≈ 16

V_{pp}.

b) Estimate the resonant frequency value f_0 of investigated resonant circuit.

c) Measure the voltage across the resonant circuit V vs. frequency of the generator signal keeping its output voltage value constant. Write results of measurements in a table and make a graph.

d) Find experimentally the resonant frequency value f_0 of investigated resonant circuit

e) Find experimentally 2Δf bandwidth of the investigated resonant circuit, i.e. the difference of frequencies, for which the magnitude value of the resonant circuit voltage goes down -3dB in respect to its maximum value (for the circuit resonant frequency).

f) Estimate the Q-factor of the resonant circuit (keep in mind that the circuit is loaded with a connection cable and an input impedance of the oscilloscope):

$$Q = f_0/2\Delta f$$

g) Change the shape of input signal to a triangle, and then to a rectangle. Write in the report, what signal appears on the output for input signal frequency near to f_0, $f_0/3$, $f_0/5$ of signal frequency.

Fig.7.5. Measurement of the magnitude of parallel resonant circuit impedance vs. frequency

7.2.2. Observation of voltages in the sine wave LC generator circuit

a) With a help of the LC generator schematic diagram (Fig.7.4) calculate the resonant frequency $f_{0(LC)}$ of the LC circuit used in the generator.

b) Place the M-5 module in the power supply rack and switch the power on..

c) Using oscilloscope probes, plugged into both inputs of the oscilloscope, obtain synchronized display of signals appearing in measurement points Sl and S2.

d) Sketch observed waveforms in common coordinate axes providing every curve with a name and sensitivity value of the oscilloscope channel (or mark double scale onto voltage axes).

e) Measure and note the generator frequency f_0 using the oscilloscope probe connected to the digital frequency meter.

f) Explain in conclusions:

- whether signals in points Sl and S2 are, or should be, shifted in phase?
- how the generator behave in a response to, say, two times enlarged value of C3 capacitance?
- which elements of the generator decide about its initial operating point parameters, especially about initial value of the emitter current?
- how the generator will react on loading of the resonant circuit?

7.2.3. Observation of voltages in the quartz generator with a CMOS inverter

a) Using oscilloscope probes, plugged into both inputs of the oscilloscope, obtain synchronized display of signals appearing in measurement points S4, S5, S6 and WY.
Note! Signals in points S4, S5 and S6 should be observed and sketched in respect to the signal in point WY.
b) Sketch observed waveforms providing every curve with a name and sensitivity value of the oscilloscope channel.
c) Comment obtained results. In peculiarity explain, does the generator show sensitivity to touching some of its points with the probe.

7.2.4. Observation of voltages in the relaxation generator with an operational amplifier

a) With a help of the relaxation generator schematic diagram (Fig.7.4) calculate the frequency f_g of oscillations produced in the circuit.
b) Measure and note the generator frequency $f_{g\ meas}$ using the oscilloscope probe connected to the digital frequency meter.
c) Using oscilloscope probes, plugged into both inputs of the oscilloscope, obtain synchronized display of signals appearing in measurement points S7, S8 and S9.
d) Sketch observed waveforms providing every curve with a name and sensitivity value of the oscilloscope channel.
e) Explain in conclusions:
- whether the simultaneous change of supply voltages $V_{zas(+)} = |V_{zas(-)}|$ will change the frequency of the generator signal?
- is it possible to design the generator in such a way that somewhere inside of it would exist a triangle voltage signal?

Hint: The initial part of an exponential signal can be considered as of linear shape.

Michał Ramotowski

ELECTRONIC LABORATORY

Instruction sheet No. 8

POWER SUPPLY BLOCKS: RECTIFIERS AND REGULATORS

(The text is freely based on the laboratory sheet:
'Zasilanie układów elektronicznych'
with the kind permission of the author
Bogusław Kalinowski)

Experiment 8

POWER SUPPLIES OF ELECTRONIC EQUIPMENT

Goals of the experiment:
- investigation of a linear power supply with adjustable regulated level of stabilized output voltage and adjustable current limitation,
- investigation of a switching power supply for an IBM PC computer.

8.1. Scope of the experiment:

1. Investigations of the linear power supply.
    Operations of the linear power supply.
    Output voltage and current as functions of the load.
    Regulation ranges of the loaded and unloaded power supply.
    Load current limit level.
    Ripple filter operation.
    Battery back-up of the power supply.

2. Investigations of a switching power supply.
    Waveforms' dependency on the load current.
    Control pulses' parameters versus the load current.

Equipment on the laboratory bench:

LPE/M-6 - the linear power supply module,
LPE/M-7 - the loading module,
LPE/M-8 - the line transformer module - 9V/1A;
LPE/M-9 - the switching power supply for an IBM PC computer module
M-4650B - the universal digital meter,
MX-9000 - the multifunction device,
OS-9040 - the 2-channel oscilloscope,
- decade resistor 0.1Ω - 10kΩ/2A,
- connecting wires at least 8 pcs. , shorts (30 mm) 8 pcs.

8.2. General remarks

All electronic devices need power supply. The power may be taken from power line or from battery, and is delivered to the powered circuitry through the more or less complicated power unit. The power unit may deliver an (almost) constant voltage or current, independent on the input voltage or load conditions, then it is called as a regulated power supply. In unregulated units their output voltage depends on the line voltage or load current. Typically, power supplies are voltage stabilizers and exhibit very low output impedance. The power unit may be designed for one output voltage or few output voltages. To prevent the power unit from burn when its output is shorted, commonly a fuse is used and/or there may be installed a special output current limiter, that cuts-off the output current when its value exceeds some predefined level.

The regulated power supplies may be of linear, or switching type. The linear, or of continuous operation units have rather low efficiency and good electromagnetic compatibility. They are used mainly for low power applications. The switching regulators have high efficiency, but produce relatively high level of electromagnetic noise. They are ideal for high power, high load current applications like power units for computers, printers, TV sets, etc.

8.3. Power supplies - common blocks

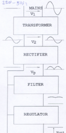


Fig.8.1. Block diagram of a linear power supply

Most of the professional and consumer electronics is designed to take power from the mains. The alternating line voltage (220V, +10%, -20%, 50Hz, in Poland) used to power an electronic equipment almost always has to be converted to one or more DC voltages of values required by the particular device. The classic and most simple way to obtain needed DC voltage from the alternating voltage line is to use a transformer and a rectifier. However, we can achieve the same results using transformerless power units, but their use is limited to some special areas assuring proper safety conditions. We begin our power supply studies with the classic constructions.

Such designs for a low-level supply voltage application typically contains a power step-down transformer, possibly optimized for minimal energy losses. But because of a compromise between allowable losses and weight and dimensions of the transformer results in that the real low power transformer efficiency rarely exceeds 80%. .

Typical blocks of the classic power supply are: power transformer, rectifier and filter. These blocks are shown in schematic diagram in Fig.81. The last block shown in the Fig.8.1.- the regulator - will be discussed later on.

The primary winding of the power transformer is connected to the mains line (220V, 50Hz), the secondary winding feeds a rectifier. Of course, there may be more secondary windings, and more rectifiers connected to them.

Denoting the primary winding voltage as V1 and secondary winding voltage as V2 we have

$$V2 = V1 \cdot (z2/z1)$$

where z1 and z2 are numbers of turns of primary and secondary windings respectively.

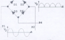


Fig. 8.2. Full-wave rectifier (Graetz configuration)

At the output of the transformer we still have an alternating voltage V2. To convert it to the DC voltage the rectifier should be used. The simplest rectifier is a single diode - it conducts only for half-periods of the input alternating signal forming so called half-wave rectifier. The are known other configurations of the diodes producing same polarity output for both half-periods of the input voltage. These are the full-wave rectifiers. One possible solution is shown in Fig.8.2. This is so called Gratez configuration.

The output voltage of the transformer V2 is connected to the rectifier input (P1 - P2). During the first half-period of V2 (see the V2 signal waveform in Fig.8.2) diodes D1 and D4 are conducting forming the first half-period of the rectifier output signal VP (P3 - P4). During the

24,5
10
34,5

4,5
6,5

second half-period of V2 (see the V2 signal waveform in Fig.8.2) diodes D2 and D3 are conducting forming the second half-period of the rectifier output signal VP. The process continues producing the voltage VP of the same polarity. Connecting point P4 to the ground we have positive DC voltage at P3. Connecting point P3 to the ground we have negative DC voltage at P4.

Observing the rectifier output waveform in Fig.8.2 it is clear that this waveform is *not* the pure DC voltage. It *has* a DC component - its value equals (2/π) of the waveform peak (amplitude) value - and other components that have to be removed leaving only pure DC voltage. For this reason the *filter* block is used (see Fig.8.1). This block easily passes through it the DC component of the voltage VP and stops all unwanted signals, what we called as *ripple*. Inside the block typically are used low-pass filters, circuits of the form shown in Fig.2.1 of the Experiment 2 instruction sheet. Sometimes, especially if the last block in Fig.8.1 - the regulator - is used, only one capacitor of large capacitance may be used as a filter.

8.3.1. Linear regulator

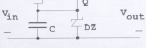


Fig.8.3. Simple series regulator

The output voltages of the simple power supply, i.e. consisting only the transformer, rectifier and filter depends on the input voltage (mains) and on the load current. In many applications, especially in measurement equipment, the supply voltage changes are undesired. To keep the output voltage independent, or at least, to diminish the changes to allowable level the voltage regulator, or voltage stabilizer, should be applied (see the lowest block in Fig.8.1).

There are many circuit ideas for voltage regulators. One of the simplest solutions is shown in Fig.8.3. The transistor Q, connected here as an emitter-follower, keeps its output voltage Vout very close to the voltage across the Zener diode DZ less the base-emitter voltage VBE. The voltage across the Zener diode is little dependent on the load current and on the input voltage Vin. The base-emitter voltage VBE of the transistor Q is little dependent on its emitter current, which is almost equal to the load current. The emitter current of the transistor Q is little dependent on its collector voltage. The net effect of all of it is that the changes of the output voltage Vout due to change of Vin, or the load current are considerably smaller than changes of the voltage Vin itself.

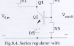


Fig.8.4. Series regulator with feedback

If the voltage stabilization obtained in the regulator from Fig.8.3. is not satisfactory then a bit more complicated circuit may be used, as in Fig. 8.4. This series regulator has a negative feedback loop with an amplifier of the error signal. It works as follows. Assuming that the Zener diode DZ keeps the emitter voltage of Q2 constant, any increase in Vout results in increase of collector current of Q2. Because the collector current of Q2 is supplied by resistor R1, the increase of Q2 current results in decrease of collector voltage of Q2 and subsequently of the base current of series transistor Q1. The emitter current of Q1 is getting smaller what counteracts the rise of the voltage Vout. In practical realizations of such type regulators the Zener diode current is additionally increased by adding a resistor between the emitter of Q2 and the input or output of the regulator. It assures better operation of the Zener diode if the emitter current of Q2 is relatively small.

Monolithic voltage regulator μA723

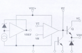


Fig.8.5. Block diagram of μA723 voltage regulator

Being a typical block of power supplies the voltage regulator is since many years produced in monolithic form under tenths of part names. One of the first designs of this type ICs was μA723 introduced by Fairchild. Its block diagram is shown in Fig.8.5.

The internal circuitry of the μA723 chip consists of three functional blocks: reference voltage source, error amplifier and current limiter. Two of these blocks one might note in previously described series regulator, shown in Fig. 8.4.

The basic element of the reference voltage source is thermally compensated Zener diode fed by a current source. The Zener diode voltage is internally buffered with an amplifier and its value equal to 7.15V is brought to pin VREF. This signal can be used as a reference voltage for the regulator block.

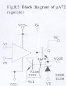


Fig.8.5a. Current limiter application

The error amplifier compares the signal proportional to the output voltage, connected to VI terminal (inverting input), with the signal proportional to the reference voltage connected to VN terminal (noninverting input). The amplified difference - the error signal - controls the regulating transistor Q. If the signal at VI is greater than at VN, the amplifier output voltage is negative and the emitter current of the regulating transistor Q is decreased causing the output voltage of the regulator to go down. In a reverse situation at both inputs, the output voltage goes up. The general rule in the system is that it automatically tends to achieve zero error signal. Both voltages at VI and VN may be taken either directly from their sources (the output and the VREF terminal), or with use of voltage attenuators, depending if the regulated output is greater or smaller than VREF (the reader is advised to prove it himself).

To prevent oscillations in the regulating loop the small capacitor (510 ÷ 1000pF) is typically connected between the amplifier output and the VCC- terminal.

In the μA723 chip there is a current limiter block. There are two terminal pins ILIM and ISEN and a transistor whose collector is connected do the output of the error amplifier. Normal use the ILIM pin is connected to the VOUT terminal and an additional resistor Rlim (current sensor) is connected between terminals ILIM and ISEN. The load is now connected to the pin ISEN. If the load current, flowing through Rlim produces across Rlim a voltage drop exceeding 0.6V, the limiting transistor starts conducting what decreases the base current of regulating transistor Q, and consequently, the load current.

The regulator µA723 can also be used to build power supplies with a negative output voltage. In some designs of such type the Zener diode DZ is used.

The maximum output current from µA723 unit is limited to 150mA. If the load current from power supply is expected to exceed this value, one or more power transistors may be added externally in parallel connection to the regulator.

Monolithic voltage regulator LM117/317

Another simple regulator circuit using LM317 IC is shown in Fig.8.5.b. The regulated output voltage is set according to the formula given below

$$V_{reg} = V_{ref} \cdot (R1 + R2)/R1$$

where $V_{ref} = 1.25V$, for $I_3 << I_{R1}$, and recommended R_1 value is 120Ω. The input voltage Vin should be greater than the expected regulated output by at least 2V (remember about input ripple component!).

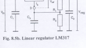


Fig. 8.5b. Linear regulator LM317

8.3.2. Switching regulator

Switching regulators are used when high energetic efficiency (up to 90%), low weight and dimensions are of main importance. Their major drawbacks are high level of EM noise and relatively slow response to the load changes.

The typical switching regulator block diagram is shown in Fig.8.6. There are:

- a dc-dc converter that converts the unregulated input voltage Vin to the regulated Vreg voltage proportionally to the duty-cycle of the control pulse train (f, w),
- a voltage divider R_{p1}, R_{p2}, whose values are selected to produce the error signal V_p

$$V_p = V_{reg} \cdot R_{p2}/(R_{p1} + R_{p2})$$

(note that for steady-state operation $V_p \cong V_{ref}$).

- a pulse width modulator (PWM) that controls the duty cycle of the control pulse train in a response to the voltage difference V_p - V_{ref}.

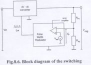


Fig.8.6. Block diagram of the switching regulator

Switching regulators may produce output voltage that is higher or lower than the unregulated input, and there may be the galvanic isolation between the output and input circuits, or may be not. In our experiment a very simple version of the switching regulator is examined (Fig.8.7). Its output voltage is lower than the input one, and there is no galvanic isolation between input and output circuits.

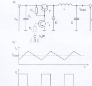


Fig.8.7. The dc-dc converter: a) equivalent diagram, b) waveform of the inductor current and corresponding control signal Vs(t)

Observe the control voltage V_s coming from the PWM modulator. At T_w part of the cycle transistor T_s conducts what saturates the switching transistor T. Than almost full input voltage V_{in} arrives at the left end of the inductor L. The diode D is reverse polarized and cut-off. The voltage at the right end of L is considered constant (large value of C). So the voltage difference across the inductor is almost constant and the inductor current IL is linearly rising until T_w time ends. The second part of the control cycle starts - T_d. Then the control voltage goes to zero, the Ts and T transistors are cut-off. Due to the selfinductance effect the inductor current continues its flow, with its value linearly falling, forcing the diode D to conduct. The voltage drop across the inductor now is reversed . At the end of T_d the cycle starts again. If the output voltage is much greater than the saturation voltage of the keying transistor T (– ca. 0.2V) and the diode forward voltage V_F (ca. 0.7V) than the output voltage may be expressed as

$$V_{reg} = T_w \cdot V_{in}/(T_w + T_d) = w \cdot V_{in}$$

where *w* denotes the duty-cycle value. The expression given above is valid only if the inductor current IL keeps its direction, i.e. its waveform in Fig.8.7b is always above zero line, or only touches zero line at its lower vertexes. The load value at which the inductor current direction tends to be reversed is called a 'critical load'. For higher load current the load is called 'overcritical' - its is normal mode of operation of the discussed switching regulator. In this operation mode the output voltage is insensitive to changes of the load. Quite different situation takes place when the load current goes below the 'critical' value to the 'undercritical' one. Because the inductor current can not change its direction, the rapid rise of the output voltage is observed when the load current is decreased. To avoid such situations, the regulators like that discussed should always work with an initial load internally connected, to prevent the regulator against damage when the nominal load is incidentally removed.

The 'brain' of the switching regulator, the PWM modulator is now produced in many types of Ics. One of the most popular is the TL494 of Texas Instr.

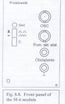


Fig. 8.8. Front panel of the M-6 module

8.4. Description of laboratory modules M-6, M-7 and M-8

Equipment on the laboratory bench:

- **M-6** - unregulated power supplies module,
- **M-7** - load module,
- **M-8** - regulated voltage and dc-dc converter module.
- MX-620B - universal digital meter;
- OS-9040 - 2-channel oscilloscope;
- MX-9000 - multifunction device (only digital multimeter is used).

8.4.1. Description of the M-6 module

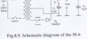


Fig.8.9. Schematic diagram of the M-6 module

The M-6 module is an unregulated power supply. Its front panel schematic diagram is shown in Fig.8.8 and schematic diagram in Fig.8.9. With this module one may observe an operation either half-wave or full-wave rectifier loaded with 10kΩ resistor. Also, a ripple filter in form of 1mF capacitor can be connected to the rectifier output. Another ripple filter R_FC_F may be investigated when the rectifier is loaded with the adjustable sinking current source. (the M-7 module).

Note. Do not load the rectifier with the current source if the C1 capacitor is not connected.

8.4.2. Description of the M-7 module

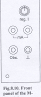


Fig.8.10. Front panel of the M-7 module

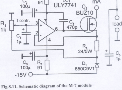


Fig.8.11. Schematic diagram of the M-7 module

The M-7 module is an absorbing current source. Its front panel schematic diagram is shown in Fig.8.10 and schematic diagram in Fig.8.11. The absorbed current may be adjusted in 0 to 200mA range with the help of an external miliammeter connected to the sockets 'mA'. The input socket 'Obc' should be connected to the loaded DC output.

8.4.2. Description of the M-8 module

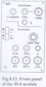


Fig.8.12. Front panel of the M-8 module

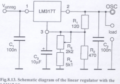


Fig.8.13. Schematic diagram of the linear regulator with the LM317 IC

The M-8 module (Fig. 8.12) contains two independent devices - continuous action (linear) regulator with the LM317 IC and the dc-dc converter with the control circuit producing a pulse waveform with adjustable duty-cycle. The schematic diagram of the linear regulator is shown in Fig.8.13. The output voltage may be adjusted in a narrow range with a variable resistor R3. The coarse setting is typically done by replacing the resistor R2.

The dc-dc converter schematic diagram is shown in Fig.8.14. Transistors T1, Ts and the diode D1 are switching elements, the NE556 IC produces switching pulses. Their duty-cycle is adjusted with the potentiometer R8 marked as 'szer.imp.' There are three 1Ω resistors (R1, R15, R3) that are used to observe the current waveforms in the circuit. Two of these resistors have both terminals ungrounded and waveforms across them can be displayed on the oscilloscope with a help of a special voltage amplifier (US1 LF356) having 10V/V gain. The monitored currents are:
- the keying transistor T1 current,
- the diode D1 current,
- the inductor L1 current.

The reference signal during observation of the currents is keying voltage marked in the schematic as 'Us', connected to one of the oscilloscope channels. The currents' signals are taken from sockets 'WY', or 'WY1' connected to the second channel of the oscilloscope and selected with switches on the front panel.

8.5. EXPERIMENTAL PART

8.5.1. Investigation of the unregulated power supply

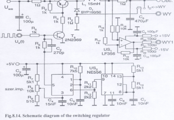


Fig.8.14. Schematic diagram of the switching regulator

a) Load the rectifier circuits in the module M-6 with the resistor 10kΩ. To do these connect the output socket 'pom.skl.stal.' to the ground. Then the R_F resistor become a load. Check if capacitor C1 is disconnected.

Observe and sketch the output signal for half-wave and full-wave rectifier.

b) Connect the capacitor C1. Connect the module M-7 to the output socket of the module M-6. Install current meter. Measure and draw on the same diagram the load characteristics $V_{out} = f(I_{load})$ for both rectifier configurations. Change the load current over the range 0 to 200mA. Evaluate the output resistance of both configurations for I_{load} = 100mA.
Measure and draw on the same diagram the peak-to-peak ripple voltage for both rectifier configurations.
Comment your observations.

8.5.2. Investigation of the linear regulator

a) Design and draw in your report a block diagram for measurement the load characteristic of the linear regulator. What modules are to be used and how you connect them.
Measure such characteristic over the load range of 0 to 200mA. Evaluate the output resistance of the linear regulator for I_{load} = 100mA. Measure the peak-to-peak ripple voltage at this point.
Comment your results.

8.5.3. Investigation of the switching regulator

a) Connect the dc-dc converter to the unregulated power supply (M-6). Load the converter with the load module M-7. Set the load current to 50mA. Set the switching pulse duty-cycle to w = 50%. Draw all available currents' waveforms and the output ripple voltage.

b) Still having w = 50% find experimentally the 'critical' current value.

c) Measure and draw the load characteristic of the converter for w = 50%. Mark 'overcritical' and 'undercritical' regions. The load current range 0 to 200mA.

d) Measure and draw the regulation characteristic of the converter $V_{out} = f(w)$ for I_{load} = 50mA.

Comment your results.

8.6. Auxiliary Literature.

1. Thomas L.Floyd " Electronic Devices", Fourth Edition, Prentice Hall.

POLSKO-JAPOŃSKA WYZSZA SZKOŁA
TECHNIK KOMPUTEROWYCH

Michał Ramotowski

ELECTRONIC LABORATORY

WYDAWNICTWO
PJWSTK