TUTORIAL 5 : TRANSIENT RESPONSE OF AN R-L-C CIRCUIT

Table of contents **page**

1 Introduction

This tutorial shows you how to simulate the transient response of an R-L-C circuit after the switching-off of a circuit breaker. The figure 1.1 shows the example we are going to study. The simulation will proceed in two phases. First, from initial conditions set to zero and the circuit breaker switched-on, the circuit will reach a steady state with zero voltage over the inductor and zero current through the capacitor. In a second stage, from the steady state reached in stage one, the circuit breaker is switched-off and we study the transient behavior.

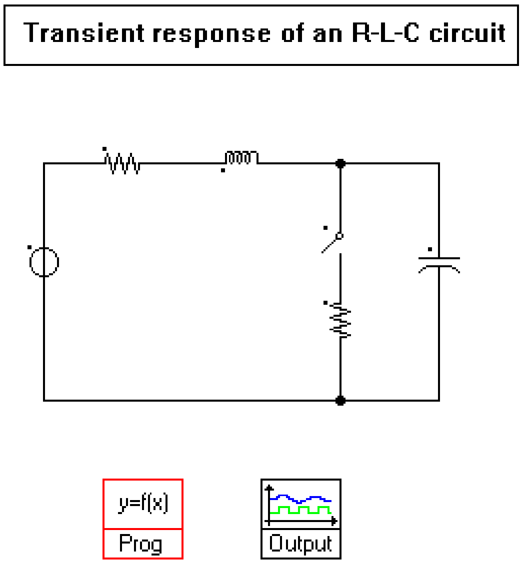

Figure 1.1: *studied example*

2 Construction of the circuit

2.1 Construction of the power part

All submenu commands you have to select are represented in **Bold** in one line. Make sure you are in the **Edit** mode. Select the following submenus and place the elements on the editing grid as shown in figure 2.1.1. After having selected a submenu, you can place the corresponding element several times on the grid without going back to this submenu.

Elements	**Elements 1ph**	**Voltage Supply**	
Elements	**Elements 1ph**	**Resistor**	(2 times)
Elements	**Elements 1ph**	**Inductor**	
Elements	**Elements 1ph**	**Circuit Breaker**	
Elements	**Elements 1ph**	**Capacitor**	

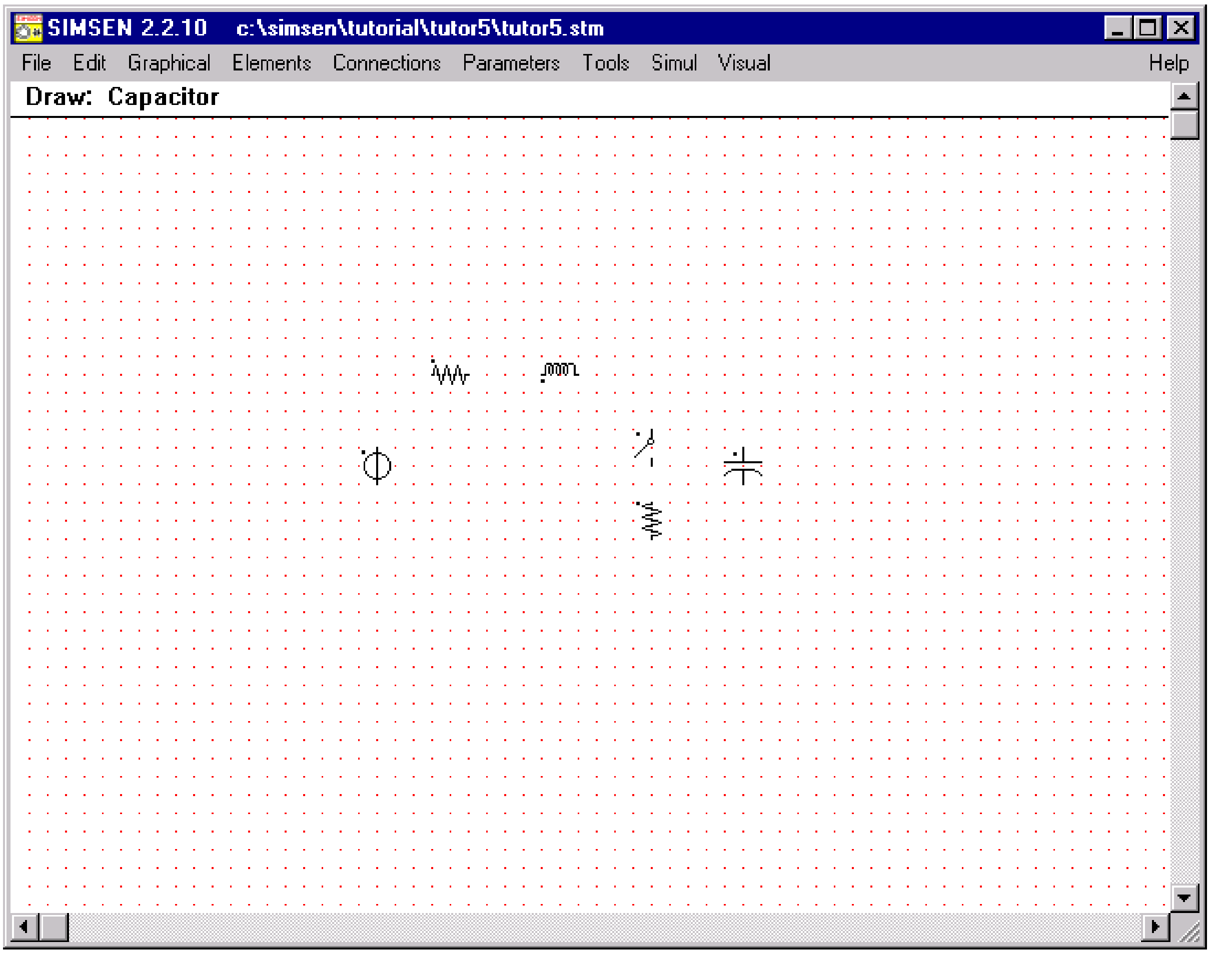

Figure 2.1.1: *Selecting and placing elements on the editing grid*

2.2 Electrical connections (wires and crossings)

You have placed your elements on the editing grid. Now you have to connect them, according to the desired topology. Select the submenu:

Connections Wire

Wire the elements as shown in figure 2.2.1.

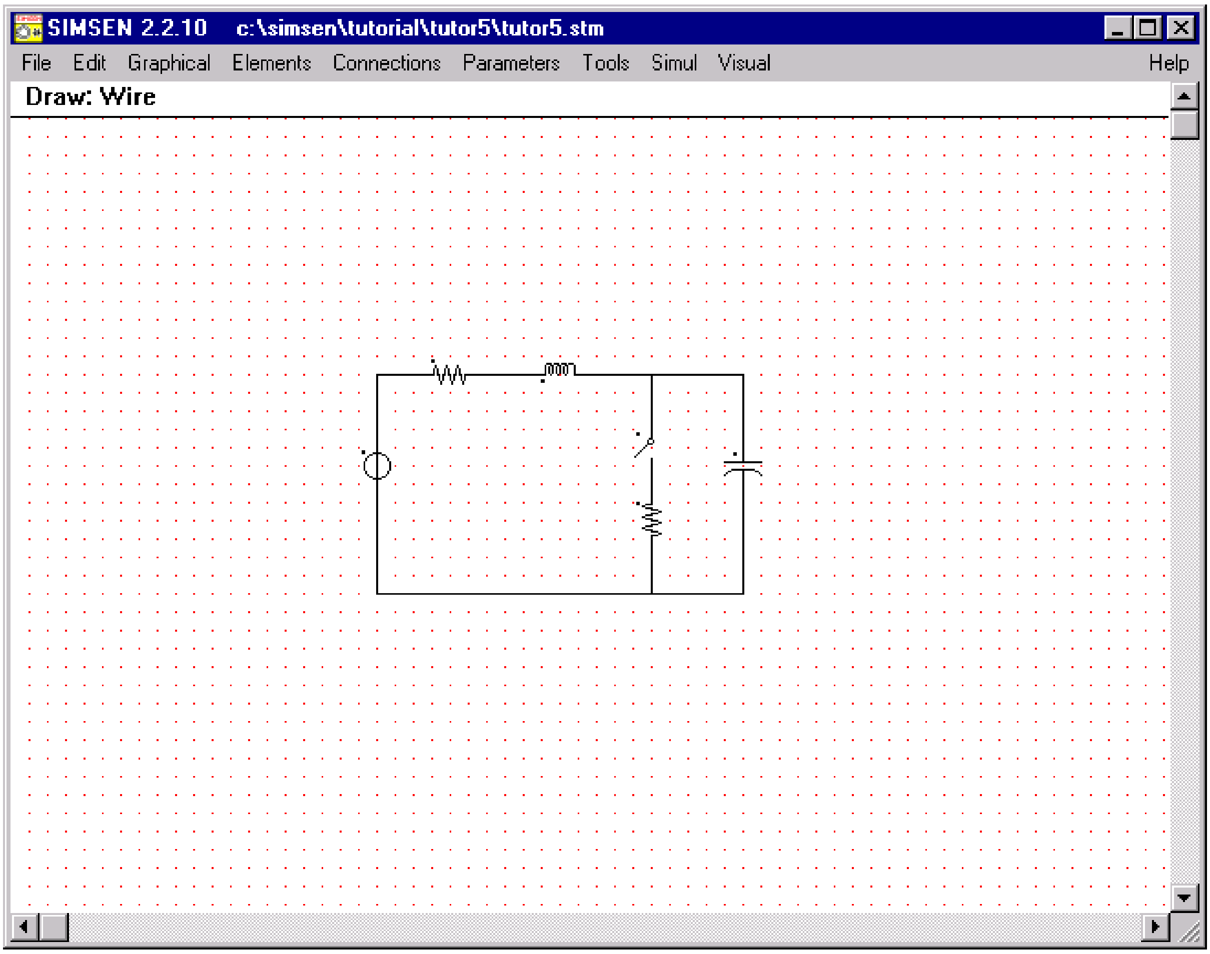

Figure 2.2.1: *Wiring the elements*

Now add the crossing points to specify the electrical connections between wires. Select the submenu:

Connections Crossing

Place the crossing as shown in figure 2.2.2.

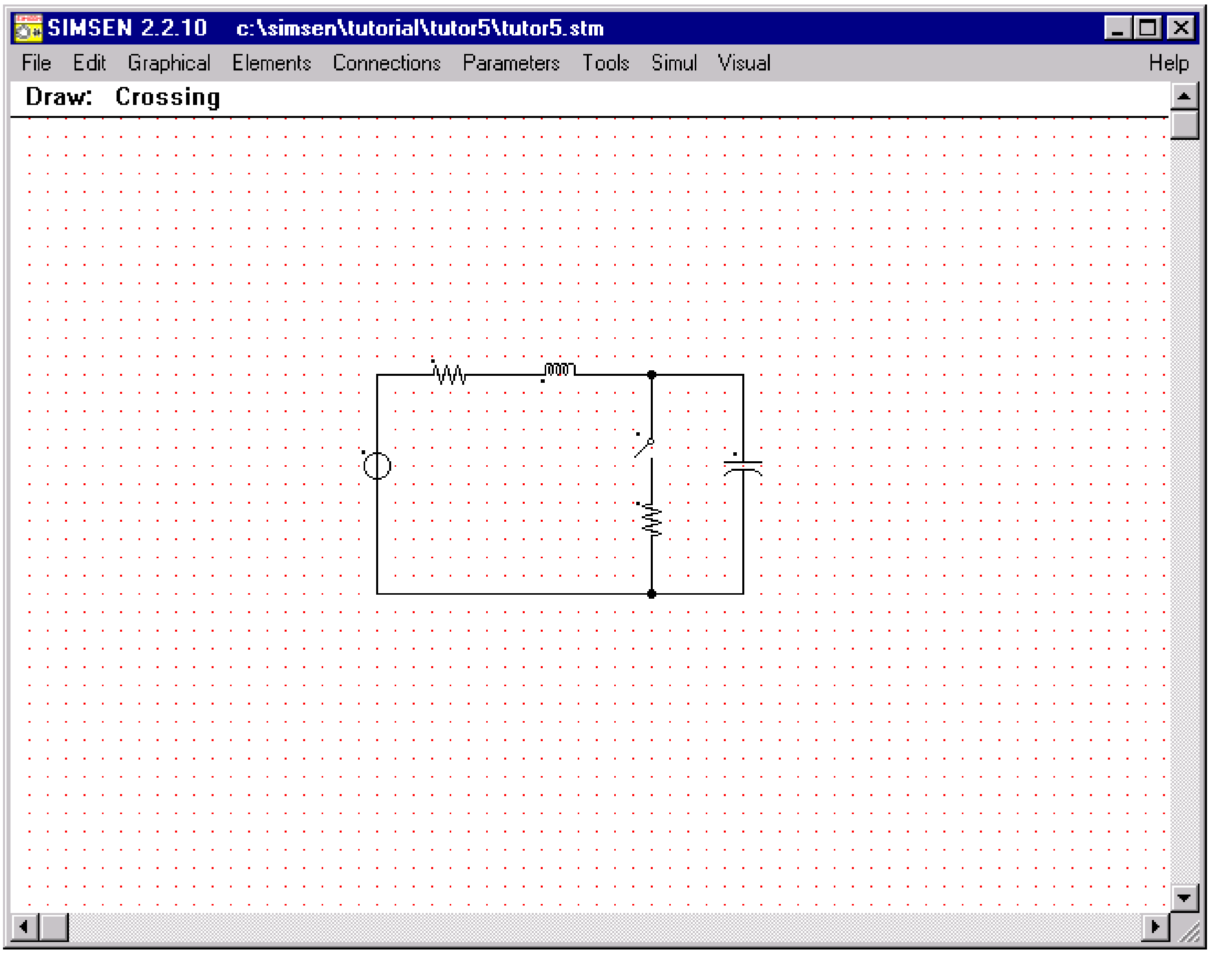

Figure 2.2.2: *Adding crossings*

2.3 Parameters of the power part

Before adding the regulation part it is useful to specify the parameters of the power part. To recognize the elements, the figure 2.3.1 shows you the power part element's names.

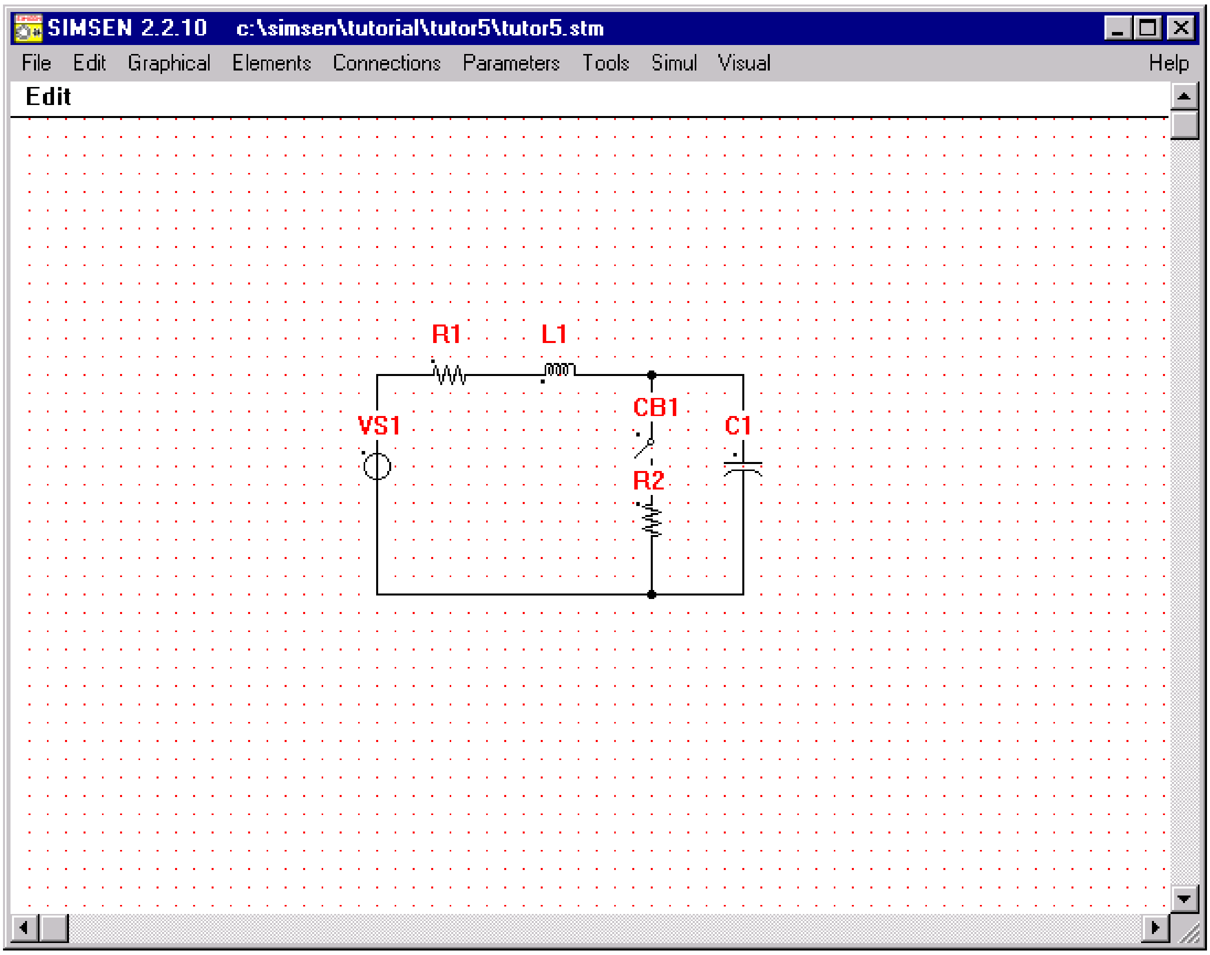

***Figure 2.3.1:** Power part element's names*

When introducing the parameters for each element you can call the related help file by clicking on the corresponding command on the bottom right of the window. Thus you can benefit from more information about the current element. Select the submenu:

Parameters Elements

Directly click on the element you want to parameterize. Thus you can open the related window for each element. The next pages will show you, with **bold characters**, the parameters you have to introduce for each element of the power part.

2.3.1 Voltage Supply 1ph **VS1**

```
GENERAL DATA :

Name     = VS1
Comment =
Writing = SI

RATED VALUES :

Sn      [VA] =  0.00000000000E+0000
Un       [V] =  0.00000000000E+0000
Fn      [Hz] =  0.00000000000E+0000

PARAMETERS :

type     [1] =  1.00000000000E+0000
U        [V] =  1.00000000000E+0001
Fs      [Hz] =  0.00000000000E+0000
Ths    [deg] =  0.00000000000E+0000

INITIAL CONDITIONS :

I        [A] =  0.00000000000E+0000
```

Explanation:

Each element of the system must have a different name. The value **1** of **type** parameter corresponds to a DC voltage supply. The rated values Sn, Un, Fn are used in two cases: either the user will use the Load-Flow program Inisim and the frequency is required or the user wishes to get output results in per unit ('Writing = PU') and the three rated values are required. As 'Writing' is by default set to SI (International System), you can omit the input of rated values. The initial conditions are the values that will be used to start the next simulation. The current explanation is valid for all the elements of the power part.

2.3.2 RLC Impedance 1ph **R1**

```
GENERAL DATA :

Name     = R1
Comment =
Writing = SI

RATED VALUES :

Sn    [VA]  =   0.00000000000E+0000
Un     [V]  =   0.00000000000E+0000
Fn    [Hz]  =   0.00000000000E+0000

PARAMETERS :

R    [Ohm]  =   4.00000000000E+0001
L      [H]  =   0.00000000000E+0000
C      [F]  =   0.00000000000E+0000

INITIAL CONDITIONS :

I      [A]  =   0.00000000000E+0000
UC     [V]  =   0.00000000000E+0000
```

2.3.3 RLC Impedance 1ph **L1**

```
GENERAL DATA :

Name     = L1
Comment =
Writing = SI

RATED VALUES :

Sn   [VA]  =   0.00000000000E+0000
Un    [V]  =   0.00000000000E+0000
Fn   [Hz]  =   0.00000000000E+0000

PARAMETERS :

R   [Ohm]  =   0.00000000000E+0000
L     [H]  =   1.00000000000E-0001
C     [F]  =   0.00000000000E+0000

INITIAL CONDITIONS :

I     [A]  =   0.00000000000E+0000
UC    [V]  =   0.00000000000E+0000
```

2.3.4 Circuit Breaker 1ph **CB1**

```
GENERAL DATA :

Name     = CB1
Comment =
Writing = SI

RATED VALUES :

Sn     [VA]  =  0.00000000000E+0000
Un      [V]  =  0.00000000000E+0000
Fn     [Hz]  =  0.00000000000E+0000

PARAMETERS :

Ron  [Ohm]  =  0.00000000000E+0000
Roff [Ohm]  =  1.00000000000E+0008
dT   [sec]  =  1.00000000000E-0006

INITIAL CONDITIONS :

I       [A]  =  0.00000000000E+0000

STATE OF PHASES :

a     [1]  =  1.00000000000E+0000
```

Explanation:

The **dT** parameter corresponds to the time constant of the circuit breaker model (**dT=L/R**). The initial state of the circuit breaker is set to **1** (switched on).

2.3.5 RLC Impedance 1ph **R2**

```
GENERAL DATA :

Name     = R2
Comment =
Writing = SI

RATED VALUES :

Sn    [VA]  =   0.00000000000E+0000
Un     [V]  =   0.00000000000E+0000
Fn    [Hz]  =   0.00000000000E+0000

PARAMETERS :

R    [Ohm]  =   1.00000000000E+0002
L      [H]  =   0.00000000000E+0000
C      [F]  =   0.00000000000E+0000

INITIAL CONDITIONS :

I      [A]  =   0.00000000000E+0000
UC     [V]  =   0.00000000000E+0000
```

2.3.6 RLC Impedance 1ph **C1**

```
GENERAL DATA :

Name     = C1
Comment =
Writing = SI

RATED VALUES :

Sn    [VA]  =   0.00000000000E+0000
Un     [V]  =   0.00000000000E+0000
Fn    [Hz]  =   0.00000000000E+0000

PARAMETERS :

R    [Ohm]  =   0.00000000000E+0000
L      [H]  =   0.00000000000E+0000
C      [F]  =   1.00000000000E-0008

INITIAL CONDITIONS :

I      [A]  =   0.00000000000E+0000
UC     [V]  =   0.00000000000E+0000
```

2.4 Simulation parameters

Select the following submenu command:

Parameters Simulation

Fill the main file as follows:

```
COMMENT :


SIMULATION PARAMETERS :

Time min                          [sec] =   0.0000000000000
Time max                          [sec] =   0.0100000000000
Integration step                  [sec] =   0.0000010000000
Precision for immediate events      [%] =   1.0000000000000
Precision for simultaneous events   [%] =   1.0000000000000
Integration process                 [-] = RK45
Write in output files every         [1] = 1
Initial conditions from           [E/M] = E
Disturbances activated            [Y/N] = NO

CONSTANT DATA :


PARAMETERS :


INITIAL CONDITIONS :


DISTURBANCES :
```

Explanation:

Simulation of the circuit behavior during **10ms** with an integration step of **1 us**.

3 First simulation

Click the **Simul** menu to go enter simulation mode. Then, click the **Sim** menu and answer **Yes** to the next question. The simulation starts. When the simulation is terminated, the simulation window looks as shown in figure 3.1.

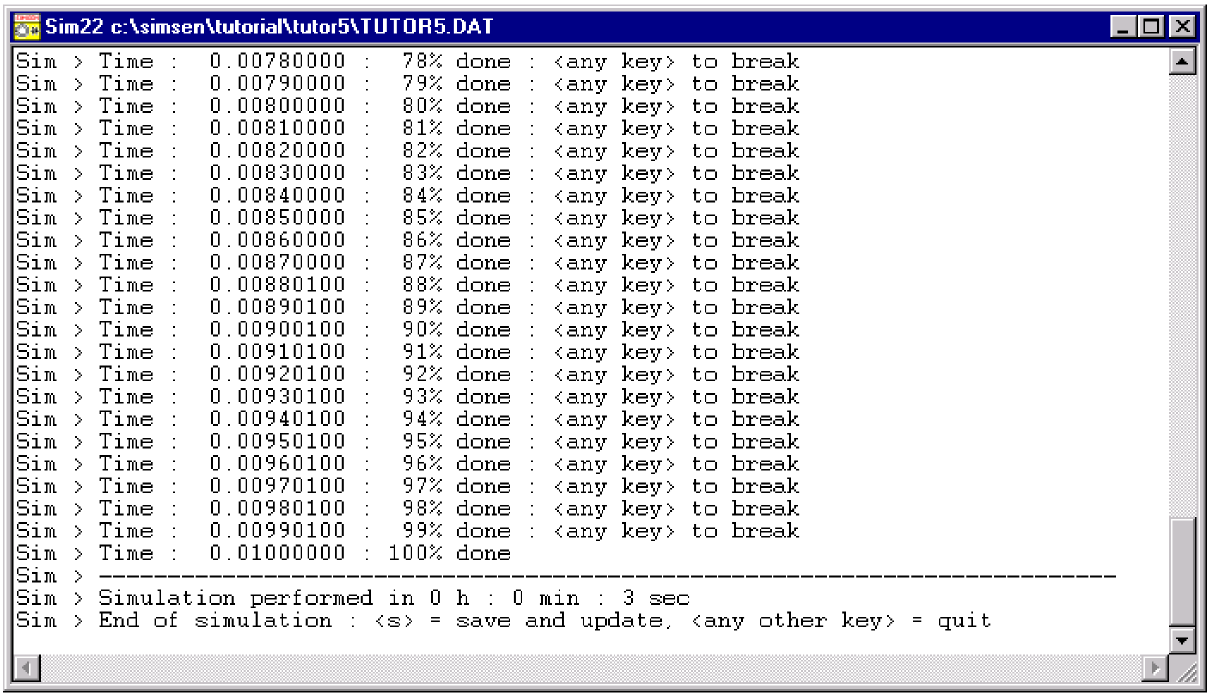

***Figure 3.1:** End of the simulation*

Please wait until you can read the last simulation message
'End of simulation: <s> = save and update, <any other key> = quit'.

Then, save the last calculated point by **pressing the <s> key**, to use it as initial condition for the next simulation.
The window closes automatically.

4 First results

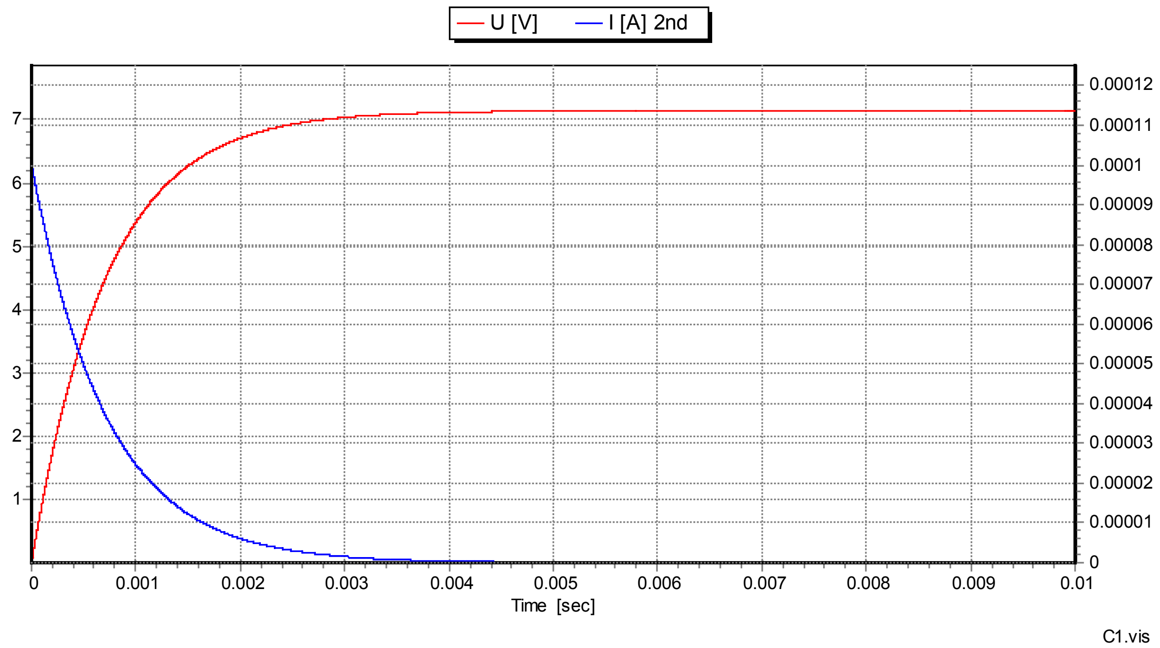

Figure 4.1: *Voltage and current of the capacitor*

5 Construction of the regulation part

5.1 Construction of the elements

Before constructing the regulation part, you must go back to **Edit** mode, by clicking the Editor menu. Then, select the following submenus and place the elements on the editing grid as shown in figure 5.1.1.

Elements	**Functions and Regulation**	**Program : Prog**
Elements	**Output**	

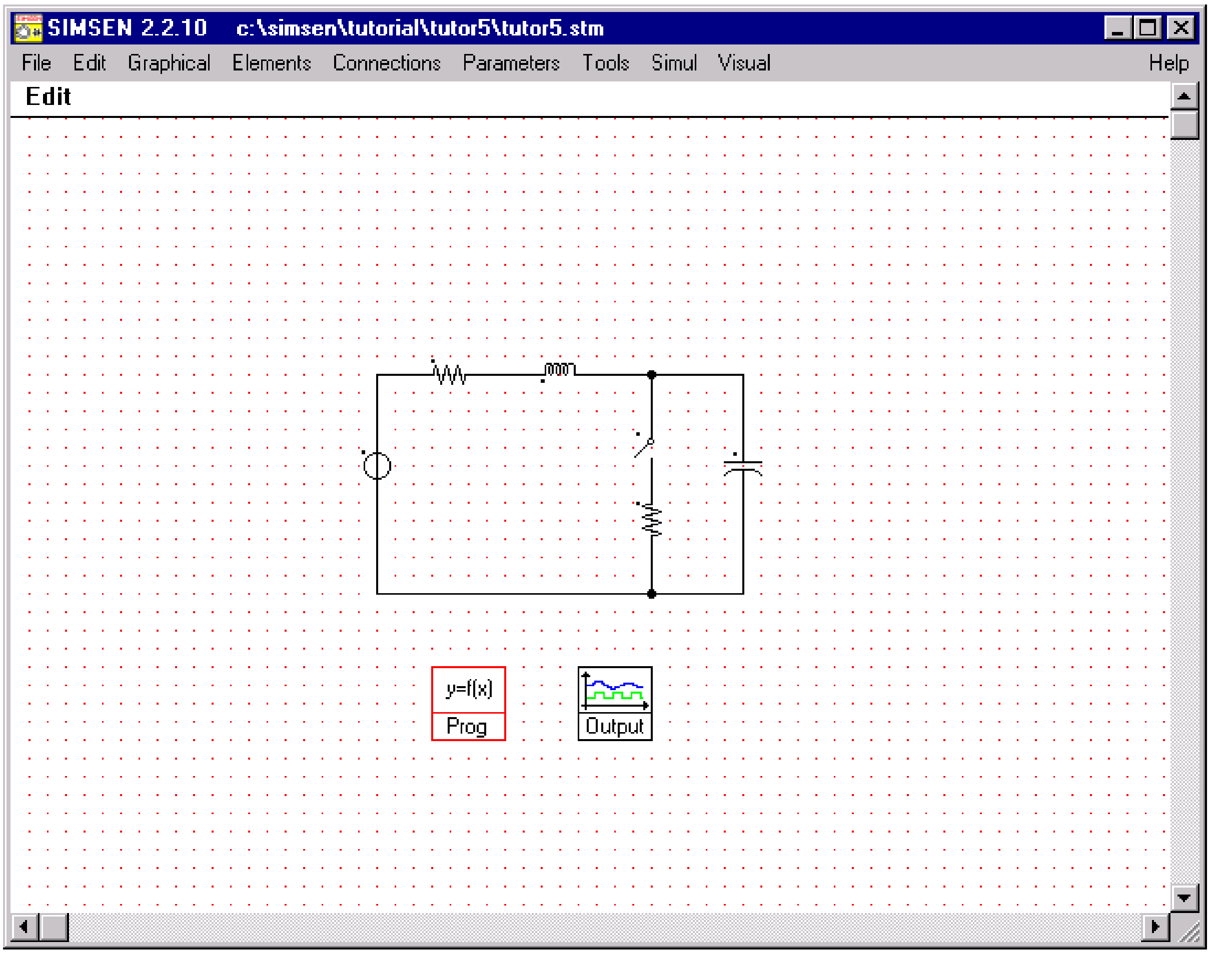

Figure 5.1.1: *Selecting and placing elements on the editing grid*

If needed, move the elements in order to align them like in figure 5.1.1.

5.2 Parameters of the regulation part

To recognize the elements, the figure 5.2.1 shows you the regulation part element's names.

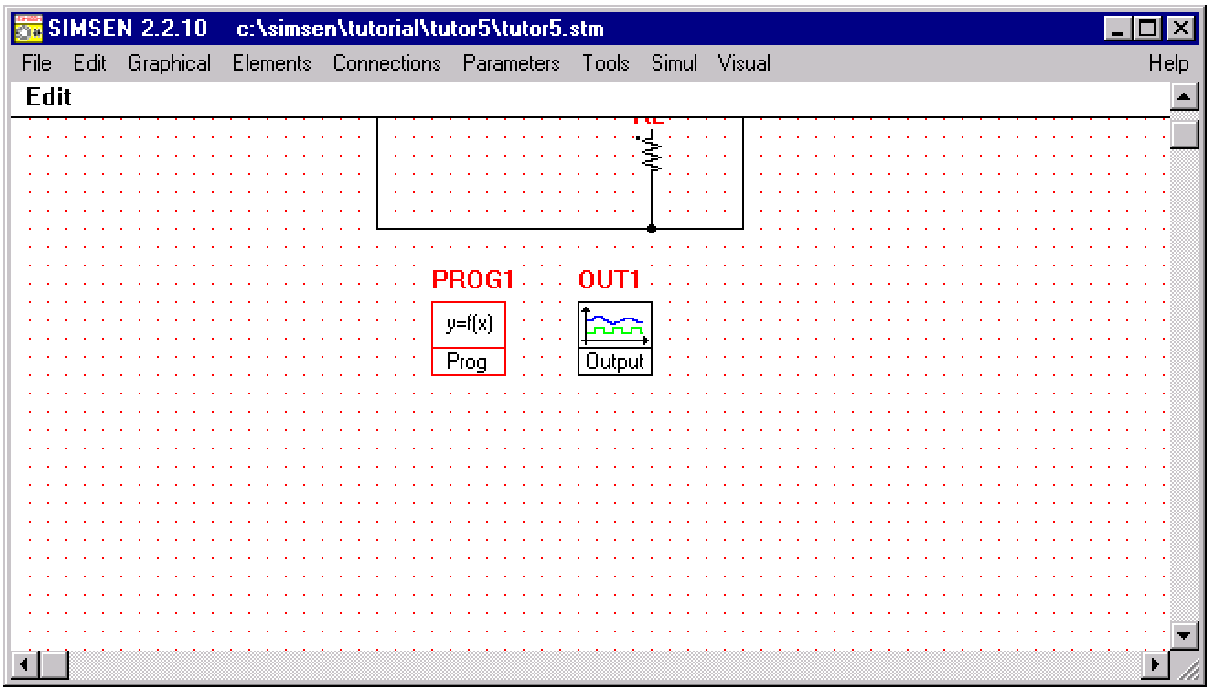

Figure 5.2.1: *Regulation part element's names*

5.2.1 Program : Prog **PROG1**

```
GENERAL DATA :

Name     = PROG1
Comment =
Writing = YES

REFERENCES X :

PROG1 Time 1 0 Time [s]  ;  x1 = Time

REFERENCES Y :


DATA :

y1=10+223*EXP(-x1/5E-3)*SIN(2*pi*x1/198.7E-6-0.74/180*pi)

PARAMETERS :

dT   [sec] =  0.00000000000E+0000

INITIAL CONDITIONS :

kdT  [sec] =  0.00000000000E+0000
y1     [1] =  0.00000000000E+0000
y2     [1] =  0.00000000000E+0000
y3     [1] =  0.00000000000E+0000
y4     [1] =  0.00000000000E+0000
y5     [1] =  0.00000000000E+0000
y6     [1] =  0.00000000000E+0000
y7     [1] =  0.00000000000E+0000
y8     [1] =  0.00000000000E+0000
y9     [1] =  0.00000000000E+0000
y10    [1] =  0.00000000000E+0000
y11    [1] =  0.00000000000E+0000
y12    [1] =  0.00000000000E+0000
y13    [1] =  0.00000000000E+0000
y14    [1] =  0.00000000000E+0000
y15    [1] =  0.00000000000E+0000
y16    [1] =  0.00000000000E+0000
y17    [1] =  0.00000000000E+0000
y18    [1] =  0.00000000000E+0000
y19    [1] =  0.00000000000E+0000
y20    [1] =  0.00000000000E+0000
```

Explanation:

In order to demonstrate the accuracy of the computation, this unit will calculate analytically the response of the circuit after a switching off of the circuit breaker. The voltage of the capacitor will be equal to a sinusoidal function having a magnitude decreasing with an exponential function, like the next equation:

$$UC(t) = A1 + A2 \cdot EXP(-t/T1) \cdot SIN(2 \cdot \pi \cdot t/T2 + \phi) \qquad (5.2.1)$$

With:

A1=10, A2=223, T1=5e-3, T2=198.7e-6, ϕ=-0.74°

The output of the Program will be used to compare the simulated voltage with the analytical value. Although that unit is explained in its related help file, it is good to repeat some rules about the Program unit. Each introduced line i containing input variables under the section – **REFERENCES X:** is assigned to the corresponding input **xi** (xi is time dependant => xi(t)). A calculation including **xi** and using some functions is done under the section – **DATA**.

Here, the unit reads the input **Time**, and creates the element x1. Then, the commands **EXP** and **SIN**, using **x1** allow to calculate the output signal of the Prog1 element :
y1 = 223*EXP(Time/5e-3)*SIN(Time*2*pi/198.7e-6-0.74/180*pi).

5.2.2 Output **OUT1**

```
GENERAL DATA :

Name     = OUT1
Comment =
Writing = YES

REFERENCES X :

C1     U  1 0 UC   [V]
PROG1 y1 1 0 UC*  [V]

- PARAMETERS :
```

Explanation:

This unit reads the value **U** of the capacitor **C1** and the output value **y** of the program **PROG1**. Even if the new Visual 2.2 allow multiple file open, this unit is useful to spare time by choosing the variables you want to output and by setting 'Writing = NO' in all other units, or simply by putting the variables you want to compare in the same file.

6 Second simulation

Select the following submenu command:

Parameters   Simulation

Fill the main file as follows:

```
COMMENT :


SIMULATION PARAMETERS :

Time min                        [sec] =   0.0000000000000
Time max                        [sec] =   0.0100000000000
Integration step                [sec] =   0.0000010000000
Precision for immediate events    [%] =   1.0000000000000
Precision for simultaneous events [%] =   1.0000000000000
Integration process               [-] = RK45
Write in output files every       [1] = 1
Initial conditions from         [E/M] = E
Disturbances activated          [Y/N] = YES

CONSTANT DATA :


PARAMETERS :


INITIAL CONDITIONS :


DISTURBANCES :

0:CB1.a=-1
```

Explanation:

Now the disturbances are activated (Disturbances activated [Y/N] = **YES**). Directly at the beginning of the simulation, the circuit breaker **CB1** will be switched OFF (**a** = **-1**).
You can now start the second simulation.

At the end of the simulation, wait until you can read the last simulation message :
'End of simulation: <s> = save and update, <any other key> = quit'.

Close the simulation window without saving the last calculated point by **pressing <any other key>**.

7 Second results

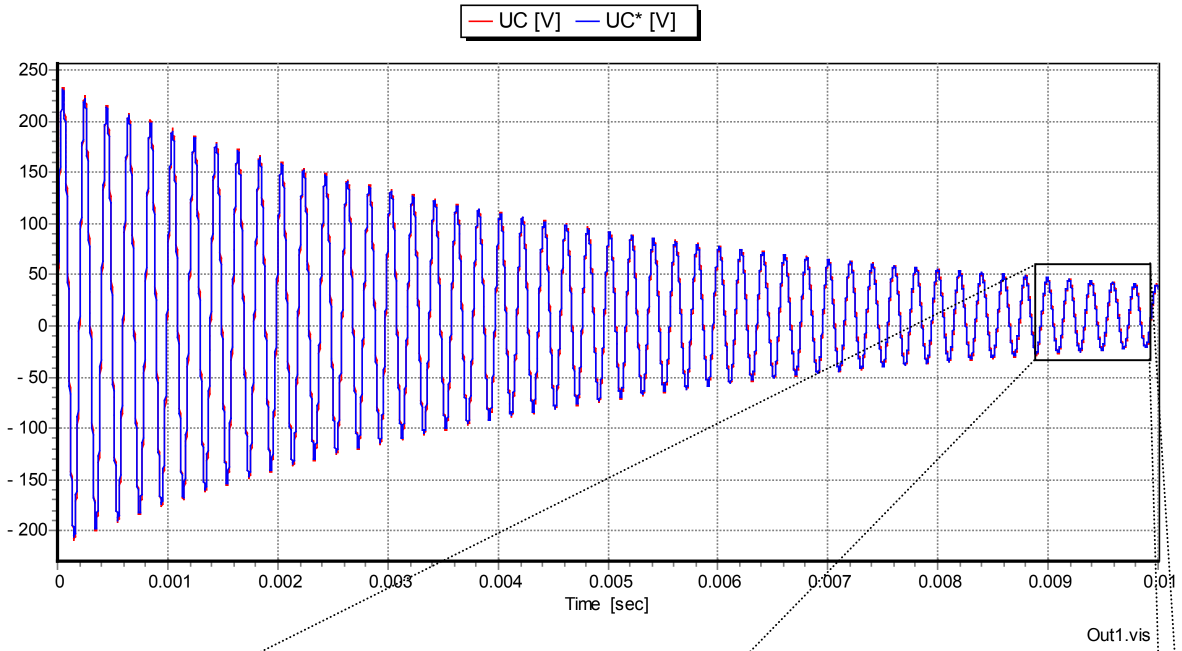

Figure 7.1: *Simulated and calculated voltages of the capacitor*

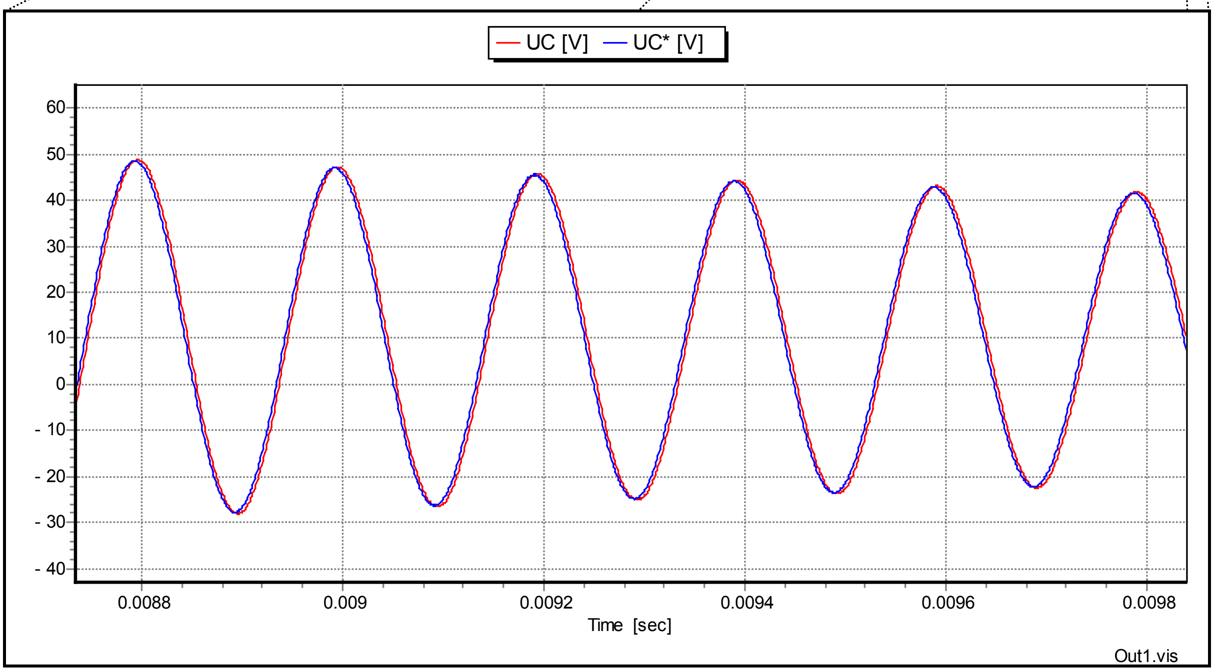

Figure 7.2: *Simulated and calculated voltages of the capacitor (Zoom)*

End of Tutorial