Electronic Technology Design and Workshop

Presented and updated by
Przemek Sekalski
DMCS room 2

2007

Electronic Technology Design and Workshop

Lecture 2

Electronic circuit analysis

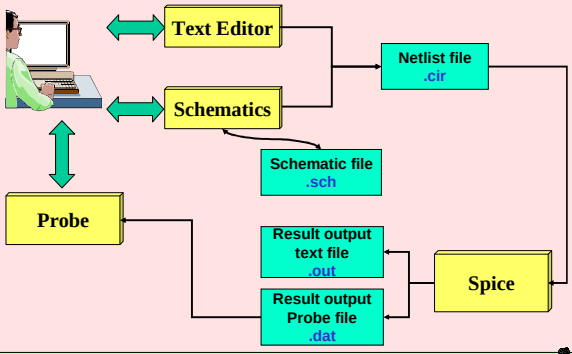
Behind the stage by the example of MicroSim Spice

ETDW course road map

- ✓ Schematic edition, libraries of elements
- ✓ Circuit simulation & netlist generation
- ✓ Microelectronics - full custom design and simulation
- ✓ Microelectronics - simple layout synthesis
- ✓ Hardware description languages - behavioural description
- ✓ Logic & sequential synthesis - programmable logic devices
- ✓ PCB design – auto-routing

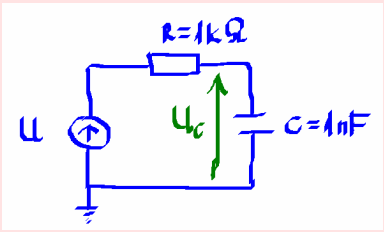
🗐 Project - bringing the pieces together

MicroSim - Data Exchange Organization



How to move the project from the head to PC ?

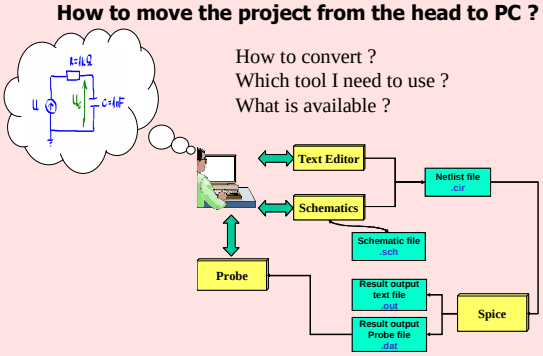
Our “innovative” circuit



Let’s try to put our idea into the MicroSim

How to move the project from the head to PC ?

How to convert ?
Which tool I need to use ?
What is available ?



MicroSim Text Editor

The **netlist** describes the connectivity of an electronic design.

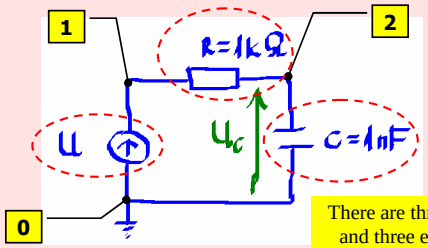
To create the netlist there is need to define the **nodes**. Between the nodes the elements/parts are located.

Each time a part is used in a netlist, this is called an **instance**. Thus, each instance has a definition.

These definitions usually list the connections that can be made to that kind of device, and some basic properties of that device, i.e. **model**, **parameter's values**, etc.

These connection points are called **ports** or **pins**, among several other names.

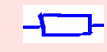
MicroSim Text Editor – the nodes



There are three nodes and three elements

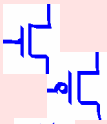
The ground node is called zero node by default

MicroSim Text Editor – the elements


Resistor **R**


Capacitor **C**


Solenoid, Coil **L**


MOS Transistors **M**

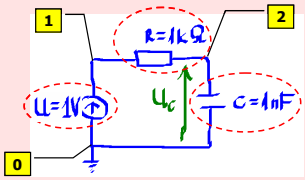

Diode **D**


Voltage source **V**


Current source **I**

... and many others

MicroSim Text Editor – the instances



Three elements corresponds to three instances

Asterisk are used to put the comment line

```
* Schematics Netlist *
R 1 2 1k
C 0 2 1n
V 1 0 DC 1
```


MicroSim Text Editor – the instances and libraries

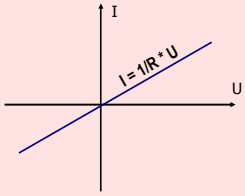
- What does it mean the C, R and V for MicroSim ?
- What does it mean the DC ?
- Are there any other shortcuts, abbreviation symbols ?
- How and why the MircoSim understands the lines ?

The answer is ...

MODEL's LIBRARIES

MicroSim Text Editor – the libraries and models

- From electrical point of view each element might be described by the current **I** which goes through the element and voltage **U** measured over it.



The current-voltage characteristic of the resistor

Variables and constraints

U = R * I

Element parameter

MicroSim Text Editor – the instance

R25 1 2 1k

- Name of the model
- The element parameter list
- The element name
  Must be different for each element (letters and numbers are allowed, no space!)
- The element nodes
  indicate which current and voltage need to be used for calculation

MicroSim Text Editor – the Voltage Source instance

The voltage source might be DC (direct current) or AC (alternating current). Moreover the voltage might change in time.

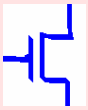
DC – constant voltage amplitude in time, frequency is zero
VInput 1 0 DC 10

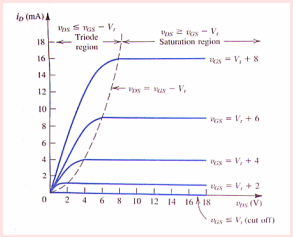
AC – constant voltage frequency in time,
VSource 1 0 AC 1 SIN(0V 0.2V 10kHz)

PULSE – defined trapezoid voltage pulse
V203 1 0 PULSE(-1mV 1mV 2ns 2ns 2ns 50ns 100ns)

Model_name/element_name nodes parameters

MicroSim Text Editor – the NMOS transistor instance






```
.MODEL N NMOS(LEVEL = 3
+ TOX   = 1.8E-8        NSUB   = 1.268366E16
+ GAMMA  = 0.3246848     VTO    = 0.6378109
+ DELTA  = -0.6127926    UO     = 532.5567276
+ ETA    = 0.0688848     THETA  = 0.0806494
+ VMAX   = 1.923644E5 KAPPA  = 4.192879E-3 RSH = 654.5658635
+ IS     = 1.126041E-14 NFS    = 3.598738E11   XJ    = 1E-9
+ WD     = 4.21328E-7    CGDO   = 2.907231E-10  CGSO  = 2.907231E-10
+ CGBO   = 3.586291E-9   CJ     = 2.37583E-4    PB    = 0.6279748
+ MJ     = 0.381556      FC     = 0.5           CJSW  = 7.38563E-10
+ MJSW   = 0.148419     LD     = 8.861389E-9 )
```


MicroSim Text Editor – the libraries

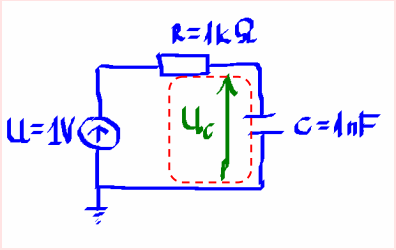
The libraries are the set of models i.e. analogue basic components, diodes, transistors, etc

There are possibility to write down your own model and build your own libraries

To include the given library into to project there is need to type the command **.INCLUDE {name of the library}**

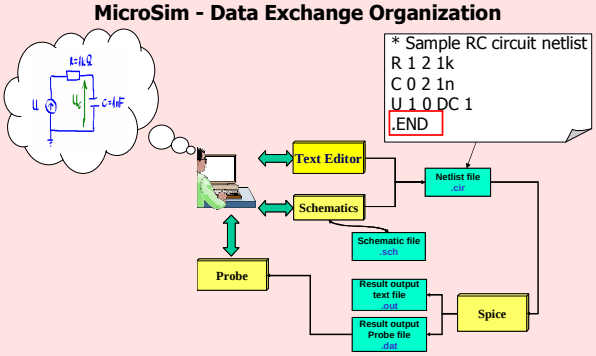
To write your model using the predefined one type
.MODEL {name} {type}

MicroSim – the simulation



How to get the results ?

MicroSim - Data Exchange Organization



SPICE - general information

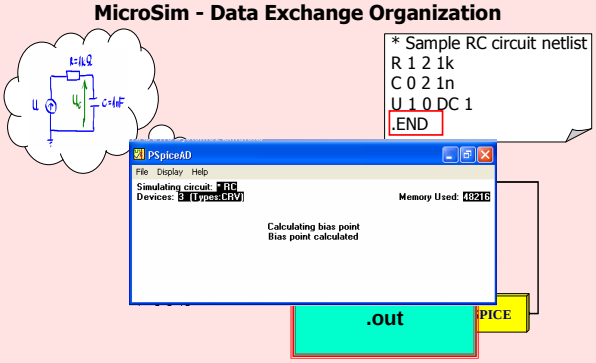
Simulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis is a general purpose analogue circuit simulator.

It is a powerful program that is used in integrated circuit (IC) and board-level design to check the integrity of circuit designs and to predict circuit behaviour.

The input data are written in ***.cir*** files as the netlist.

The details about SPICE will be given in the next lecture.

MicroSim - Data Exchange Organization



SPICE - .out file

```
**** 09/29/107 13:30:13 ****** NT PSpice 8.0 (July 1997) ****** ID# 87788 ****
 * RC
 ****     CIRCUIT DESCRIPTION
******************************************************************************
R 1 2 1k
C 2 0 1n
V 1 0 DC 1
**** 09/29/107 13:30:13 ****** NT PSpice 8.0 (July 1997) ****** ID# 87788 ****
 * RC
 ****     SMALL SIGNAL BIAS SOLUTION       TEMPERATURE =   27.000 DEG C
******************************************************************************
 NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE     NODE   VOLTAGE
(  1)   1.0000  (   2)   1.0000

   VOLTAGE SOURCE CURRENTS
   NAME       CURRENT
   V          0.000E+00
   TOTAL POWER DISSIPATION   0.00E+00  WATTS
      JOB CONCLUDED

      TOTAL JOB TIME          .05
```

Date and general information about the SPICE version

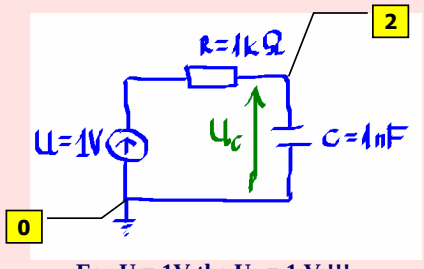
Circuit description netlist

Simulation constraints

Simulation results Voltage in all nodes

General infos about simulation, i.e. power dissipation, simulation time

The answer is ...



For U = 1V the U_c = 1 V !!!
Because U_c = V(2)-V(0) = 1 – 0 = 1V
No current flow through capacitor ;)

DC analysis

The input voltage varies from -5V to +5V with step 0.1V.

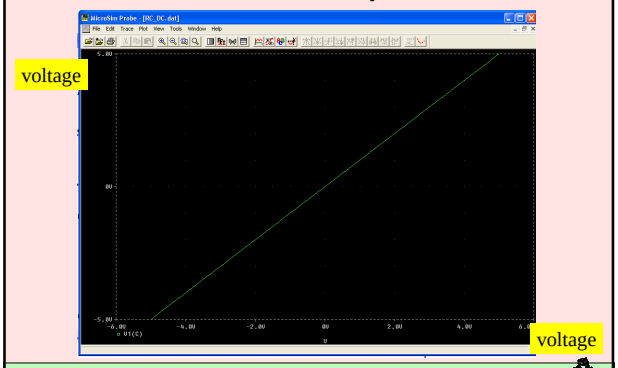
There is need to perform 100 times the same calculation with different settings for voltage source **V**.

It might be easily done by SPICE just by using the directives **.DC** which is responsible for DC analysis.

.DC {name of the source} {min} {max} {step}
.DC V -5 5 .1
.PROBE

Probe is a tool for simulation results visualisation

PROBE DC Analysis



AC Analysis

The frequency of input voltage varies from 1 Hz to 100 MHz.

With constant step i.e. 10 Hz there is 10 millions step !!!

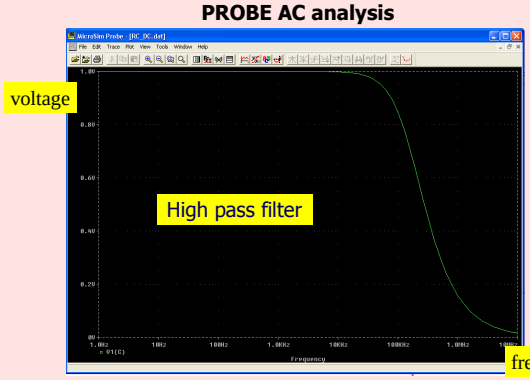
Lets make 10 steps for each decade
(10 20 30 …100 200 300… 1k 2k 3k… 10k 20k 30k… 100k …….. 100Meg)

The SPICE directive is **.AC**

```
.V 1 0 DC 1 AC 1
.AC [LIN][OCT][DEC] {points} {start freq} {end freq}
.AC OCT 10 1 100Meg
.PROBE
```

And check the results once again

PROBE AC analysis



Transient Analysis

We would like to know what will happen when the voltage will arise from 0 to 1 V and back to 0 V.

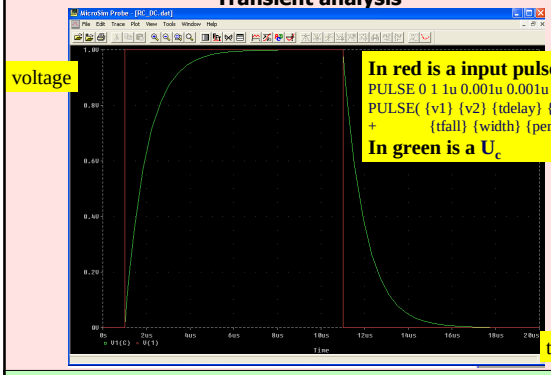
We need to perform a transient analysis (non-stationary conditions)

The SPICE directive is **.TRAN**

```
.V 1 0 DC 1 AC 1 PULSE 0 1 1u 0.001u 0.001u 10u 20u
.TRAN {print step value} {final time}
.TRAN 0.001u 20u
.PROBE
```

And check the results once again

Transient analysis



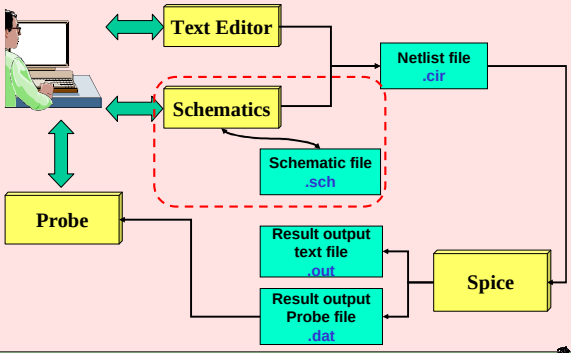
Other Analyses

- Temperature Analysis **.TEMP**
  .TEMP {value}* Examples: .TEMP 0 27 125
- Sensitivity Analysis **.SENS**
  .SENS {output variable}* Examples: .SENS V(9) V(4,3) I(VCC)
- Noise Analysis **.NOISE**
  .NOISE {output variable} {name} [{print interval}]
  Examples: .NOISE V(5) VIN
- Fourier Analysis **.FOUR**
  .FOUR {freq} {output var}*
  Examples: .FOUR 10KHz v(5) v(6,7)

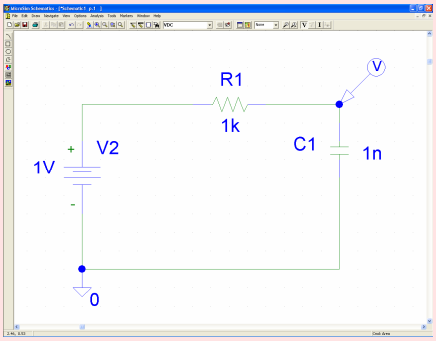
The Spice Units

F	E-15	femto
P	E-12	pico
N	E-9	nano
U	E-6	micro
M	E-3	milli
K	E+3	kilo
MEG	E+6	mega
G	E+9	giga
T	E+12	tera

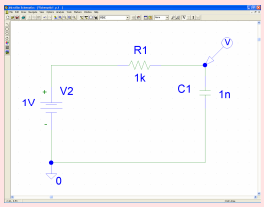
MicroSim - Data Exchange Organization



The MicroSim Schematics



The MicroSim Schematics - life is easy



One click

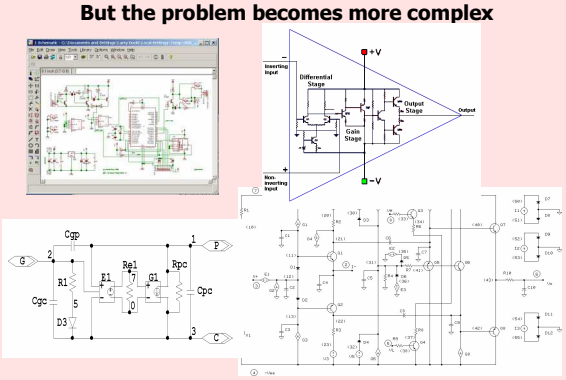
```
* Schematics Netlist *

R_R1      $N_0002 $N_0001  1k
C_C1      0 $N_0001  1n
V_V2      $N_0002 0 1V
```

The netlist is generated automatically

The simulation settings are accessible via GUI

But the problem becomes more complex



The SPICE is also available through WWW

http://spice.dmcs.p.lodz.pl/

The next lecture

The SPICE – behind the GUI

Thank you for your attention