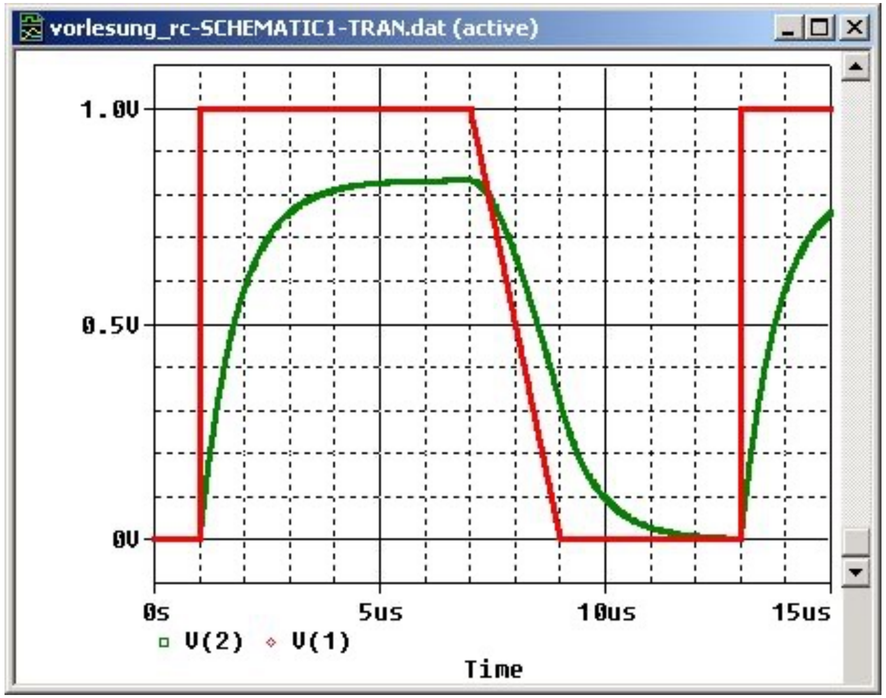
Analogue Simulation



Prof. Dr. P. Fischer

Lehrstuhl für Schaltungstechnik und Simulation
Uni Heidelberg

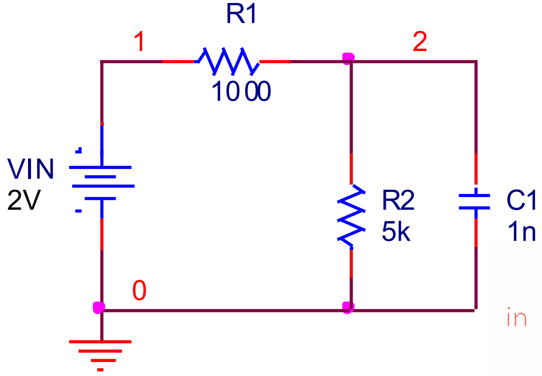
Simulators

- There are many programs around to simulate analogue circuits. One of the first (1968, UC Berkeley, IBM) was SPICE
  - **S**imulation **P**rogram with **IC E**mphasis
- All simulators use a text file ('netlist') to describe the circuit
  - Main elements are ***components*** which connect ***nodes***

    Component (Name/Type) <nodes> <parameters>
  - The syntax varies (slightly)

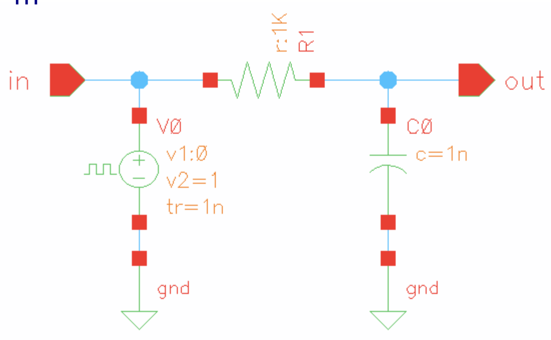
PSPICE on PC

```
* Netlist Demo
VIN 1 0 DC 2
R1 1 2 1000
R2 2 0 5K
C1 2 0 1N
```



Cadence

```
// Library name: CCS2013
// Cell name: LowPass
// View name: schematic
R1 (out in) resistor r=1K
C0 (out 0) capacitor c=1n
V0 (in 0) vsource mag=1 type=pulse val0=0 val1=1 period=1u delay=0 rise=1n \
        fall=1n width=200n
```



Models

- The behavior of the components is described by 'models'
  - These are rather trivial for capacitors, resistors
    (they can contain temperature dependence, for instance)
  - They can be VERY complicated for transistors…

- Models are (should be) provided by the manufacturer of the components (Op-Amps, transistors..)
  - We will uses models of the chip technology UMC 180 nm

- The significance of a simulation results depends on the quality of the models!
  - Do not trust your result too much!
  - Agreement (simulation vs. reality) to within 10% is very good!
  - Make sure your circuit is robust and does not rely on perfect agreement to simulation!

Elements of Simulation (in the netlist)

- The netlist contains
  - the circuit components
  - a ground node (with net name '0')
  - voltage / current sources to supply the circuit
  - pulse generators to 'stimulate' the circuit

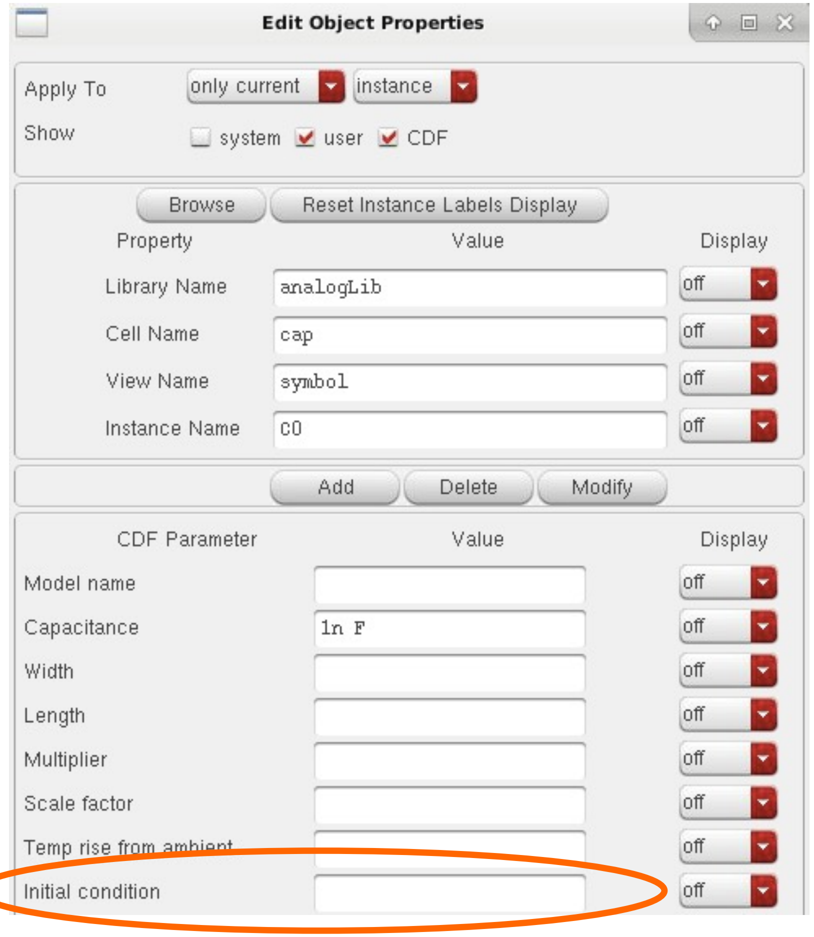
- Common errors:
  - No supply voltage (in active circuits)
  - Current in circuit is zero ('operation point' is wrong)
  - No ground node defined (voltages are 'strange')
  - No input stimulus (source may be there, but its parameter not set, see later)
  - Component values are completely off (Ampère instead uf µA)
  - Typos in parameters ('10 us s')

RUPRECHT-KARLS-
UNIVERSITÄT
HEIDELBERG

COMPONENTS

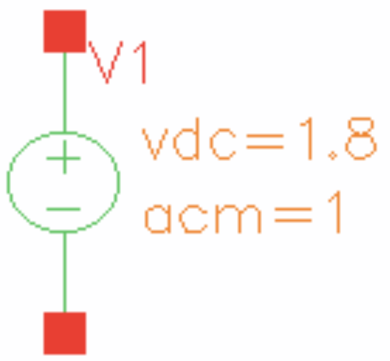
Passive Components (in analogLib)

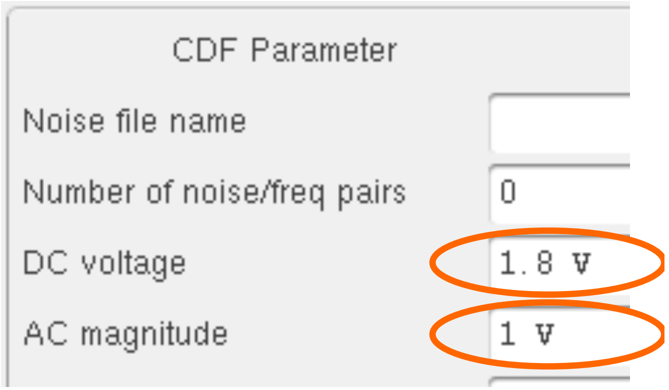
- Resistors
  - Very simple. Can add model. Can add temperature dependence. Can generate noise during simulation
- Capacitors
  - Can define an 'initial' condition, i.e. its voltage when simulation starts



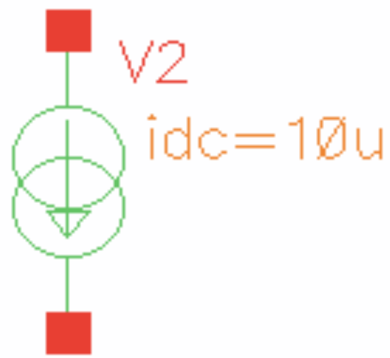
(Independent) Sources

- DC **voltage** source **vdc**
  - ideal source (R_0 = 0)
  - Must SET voltage
  - (Can also add AC)





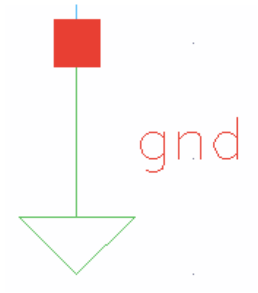
- DC **current** source **idc**
  - ideal
  - Must SET current
  - Second pin provides –I0
    Not really needed (best short to ground)
  - (Can also add AC)



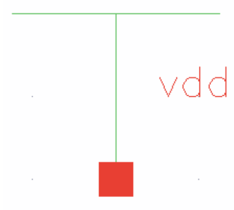
Global Nets (analogLib → Sources → Globals)

- There are special symbols which connect the pin to some 'global' net.
  - Global nets end with '!' in Cadence (e.g. gnd!, VDD!)

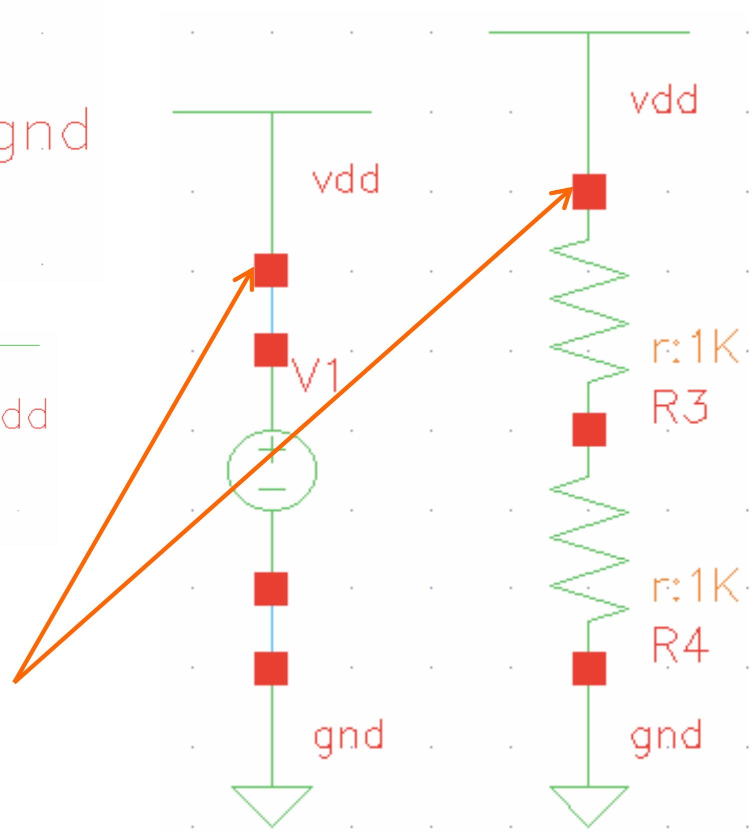
- Most important: cell **gnd**
  - Connects the pin to net gnd! (= net 0)



- Others, for instance **vdd**

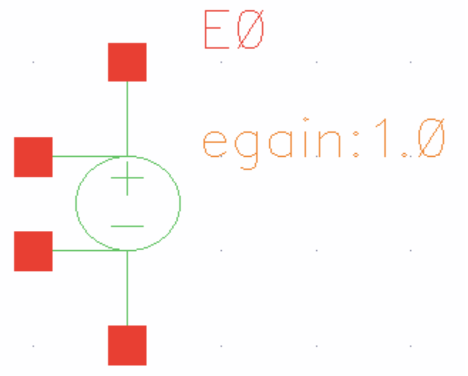


  - The upper nets in the schematic are connected!

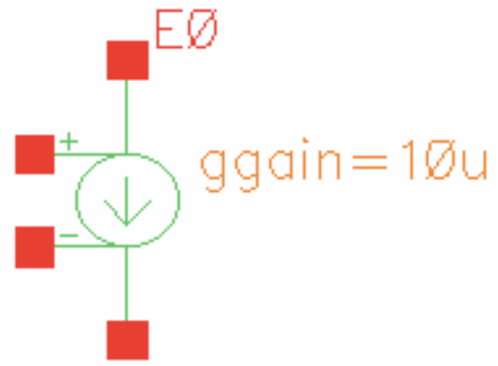


(Dependent) Sources

- There are voltage / current sources which can be controlled by a voltage (or current)
- These are very useful in simulation to simplify stuff
- Example1: Voltage Controlled Voltage Source VCVS
  - Vout = gain * Vin
  - Gain has no dimension (volt / volt)
  - This is an (ideal) voltage amplifier



- Example2: Voltage Controlled Current Source VCCS
  - This is an idealized model of a transistor!
  - Gain is in Ampère / Volt (i.e. Siemens) ('ggain' in Mhos in Cadence)



'Active' (independent) Sources

- There are sources which act as signal generators (in 'tran' simulations, see later):
  - Vpulse generates rectangular signals
  - Vsin generates sine waves
  - Vpwl generates piece-wise-linear signals

- These have several parameters that must be set.
  - Check out!

- They ALSO have DC and AC components for simple simulations

RUPRECHT-KARLS-
UNIVERSITÄT
HEIDELBERG

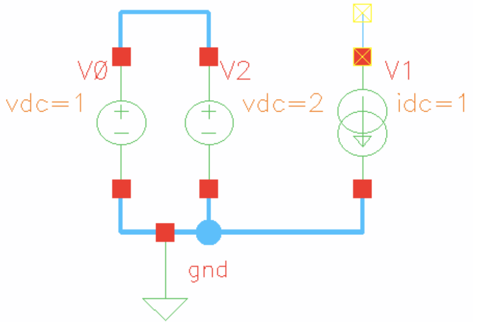
Simulation Types

.OP analysis

- This is a **static** analysis, i.e. **time plays no role**
  - Imagine you 'switch on the circuit' and observe the steady state
  - All capacitors are removed from the circuit
  - Inductors are reduced to their ohmic resistance
- All nonlinearities (diodes, MOS,..) are considered
  - It is a 'large signal analysis'
- An iterative procedure is used to find a stable steady state
  - Initial conditions (of capacitors or from the simulation window) are taken into account

- .OP is done 'automatically' before other simulations start

- .OP can fail
  - Shorted voltage sources
  - Open current sources



DC Analysis

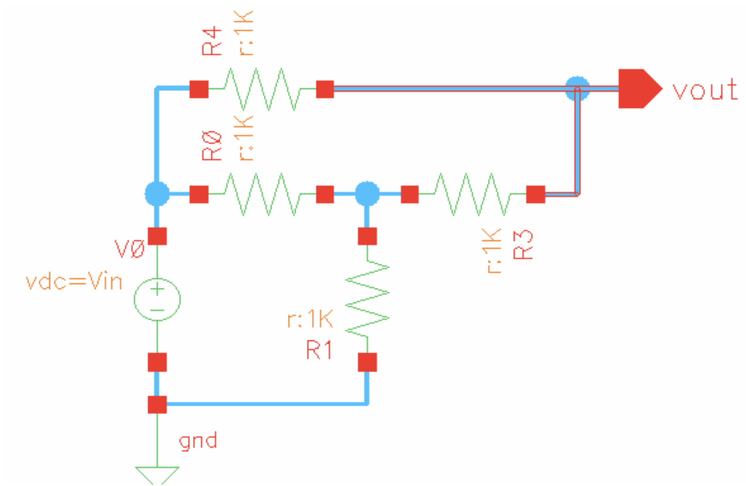
- This is just a series of .OP analysis
- One parameter is changed in each analysis
  - voltages
  - temperature
  - component values (design variables)

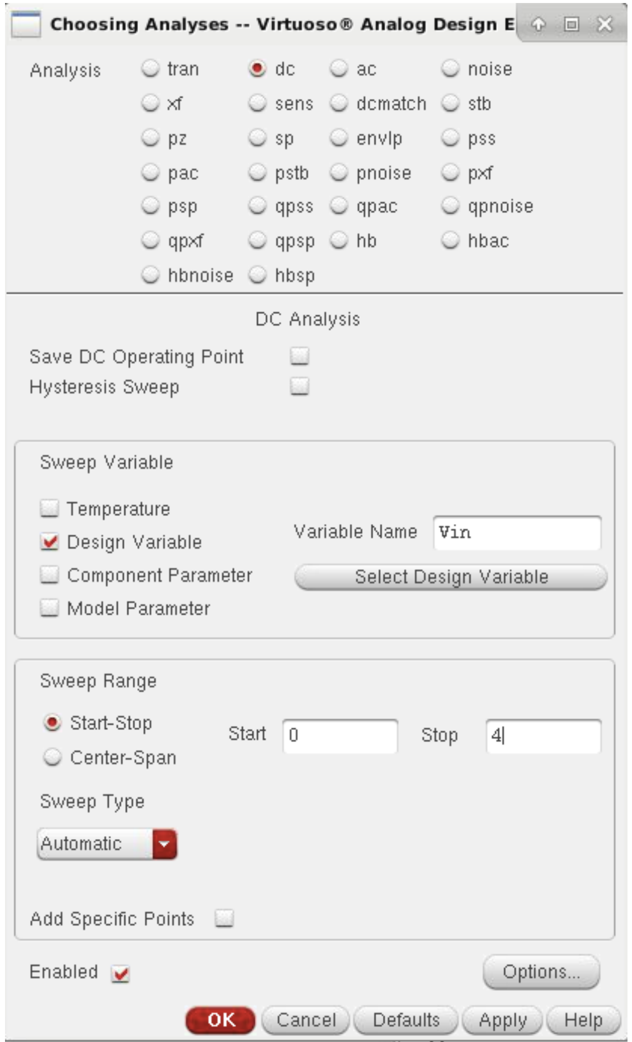
- It uses the 'DC voltage' and 'DC current' of sources

- Used mainly to determine the operation point of a circuit

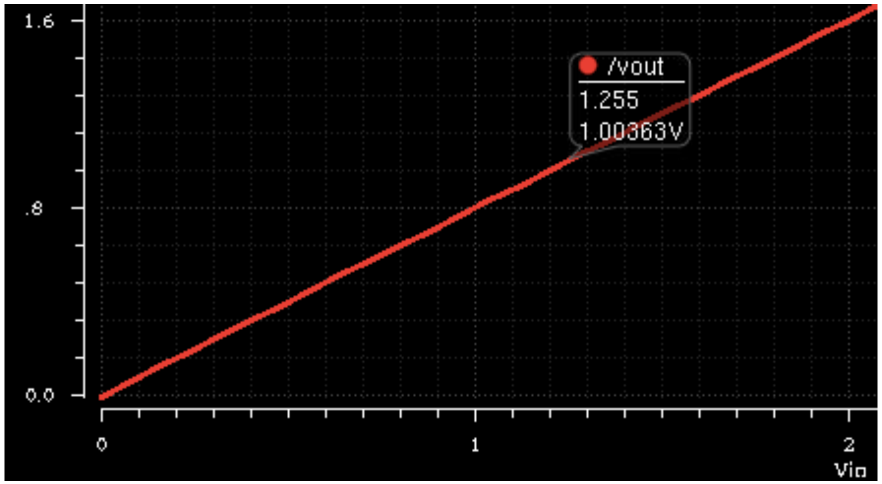
DC Analysis: Example 1

- Which voltage *Vin* is required in this circuit to get *vout*=1V ?



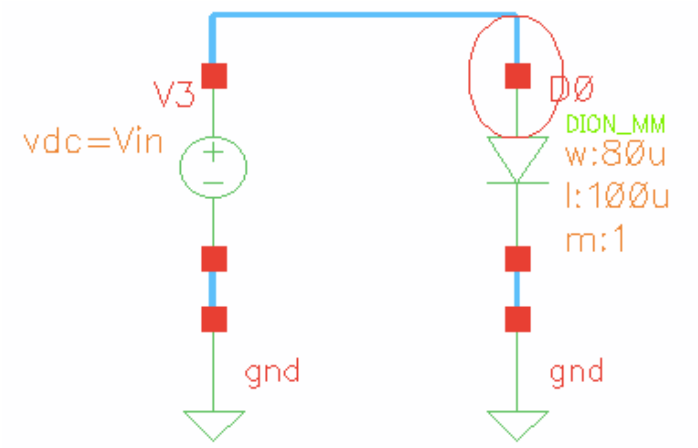


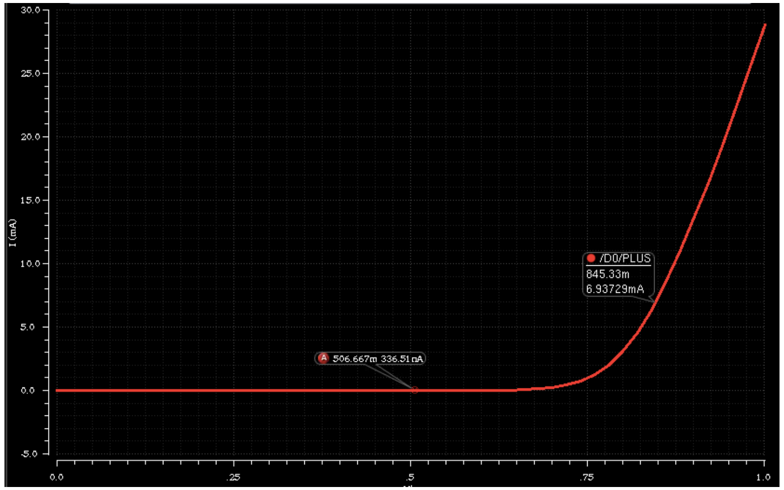
- Perform a DC sweep of Vin:



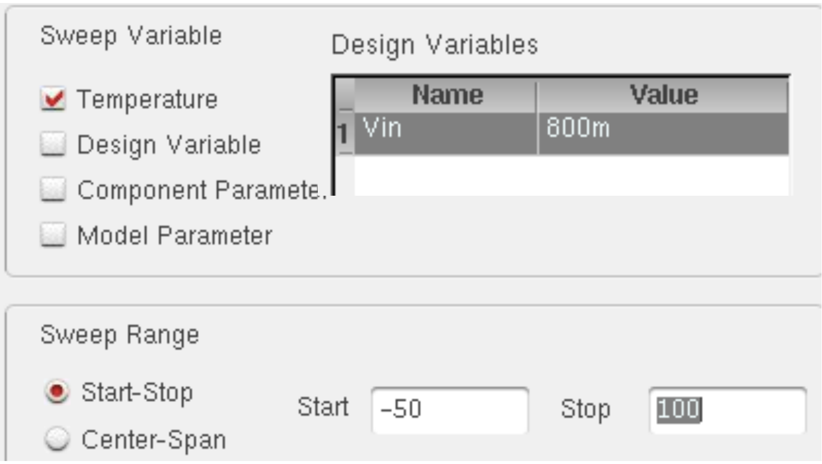
DC Analysis: Example 2

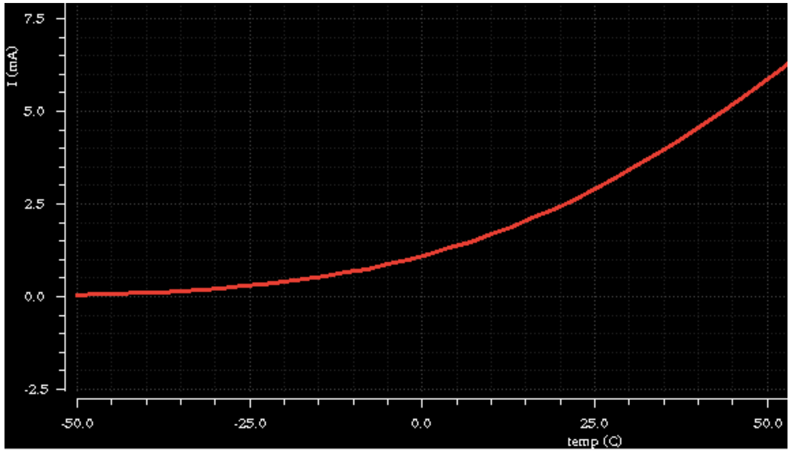
- How does the current in a diode depend on its voltage?





- How does the diode current at 0.8V depend on temperature?





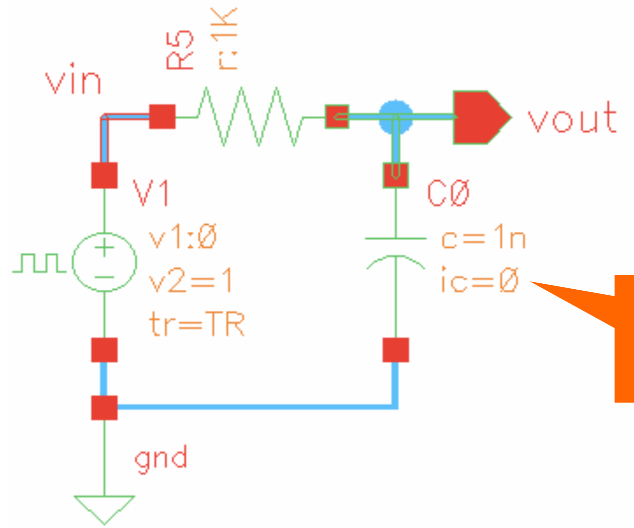
Transient Analysis

- Large signal analysis (all nonlinearities are considered)
- Generates voltages/currents as a function of time
  - Like an oscilloscope trace…
  - All caps / inductors are considered (including voltage dependencies…)
  - Most complicated (slowest) analysis
- Changes in circuit are updated in small time steps
  - Modern programs adapt the time steps to the activity in the circuit
  - NOTE: very fast signals require very fine time steps -> simulation becomes slow.
  - DO NOT make signals (outputs of vpulse) faster than needed
- Operation point is determined by a preceding .OP analysis

Transient Analysis: Example

- How does a low pass react to a triangular input ?

Voltage 1	0 V
Voltage 2	1 V
Period	2*TR s
Delay time	0 s
Rise time	TR s
Fall time	TR s
Pulse width	0 s

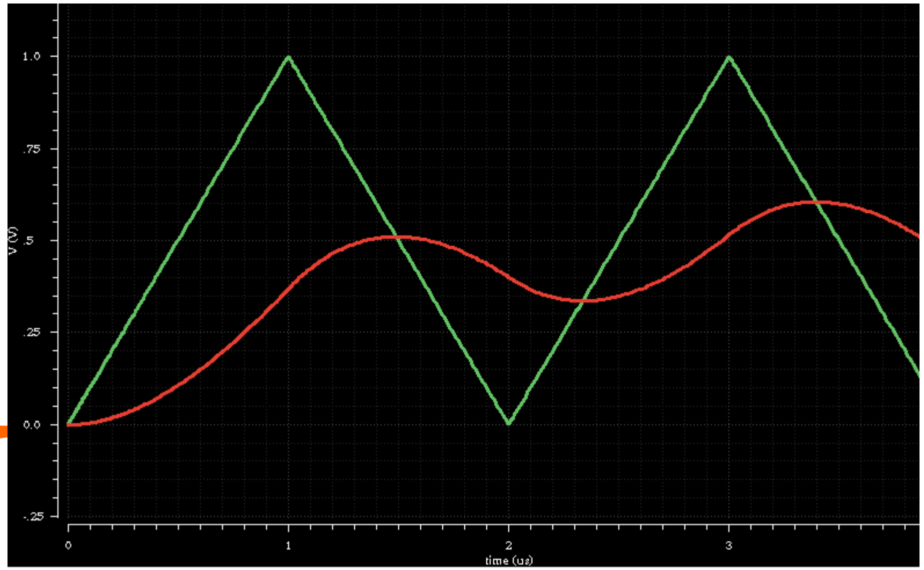


Note initial condition

- τ_{LP} = 1µs

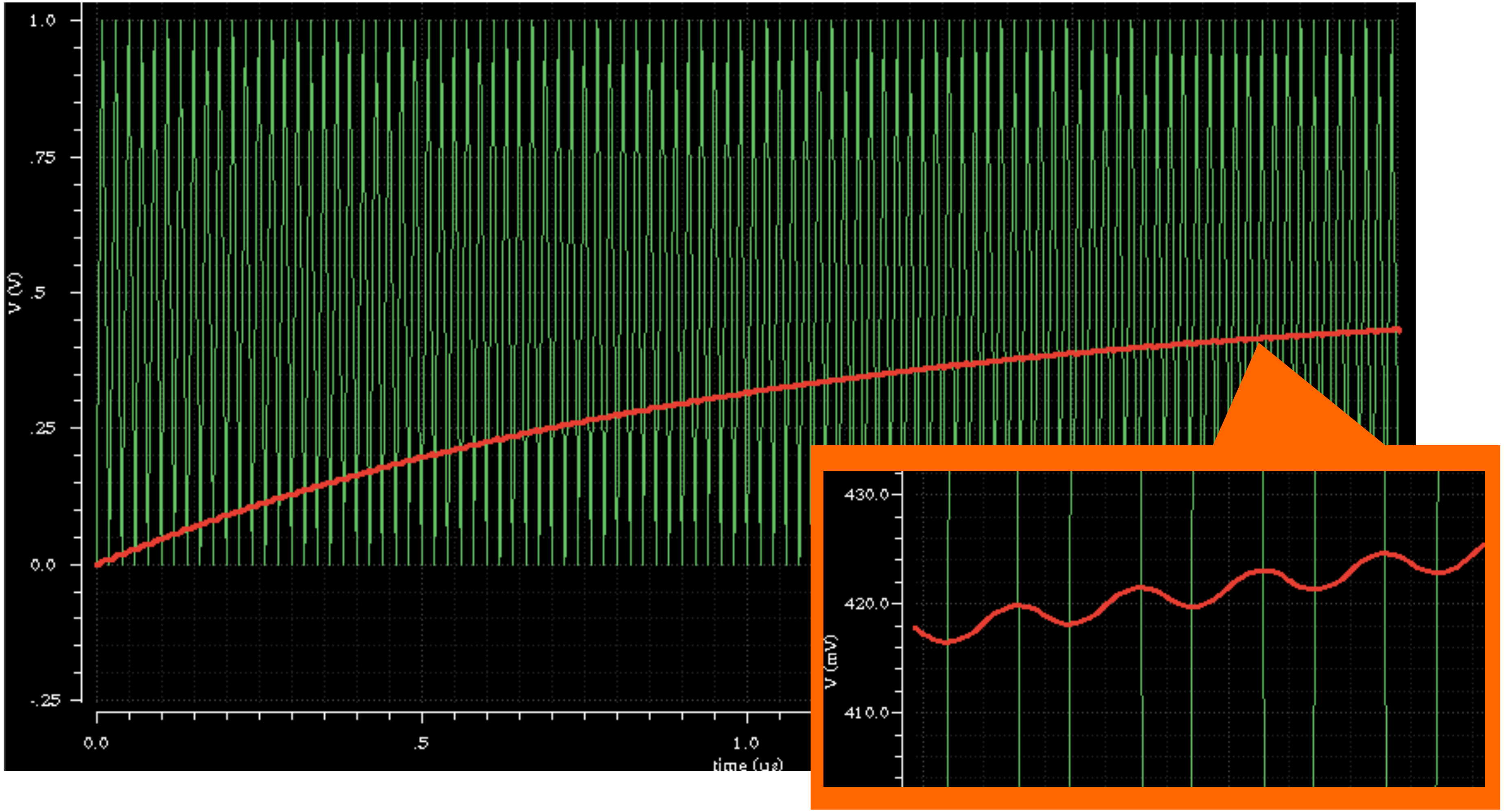
- For TR=TF=1µs:

Note initial condition



Transient Simulation: Example

- The same at high input frequency (TR = TF = 10ns):



- Do you understand what happens?

AC Analysis

- This LINEAR analysis works in the frequency domain
- It assumes ALL components linear

1. First the operation point is determined
   - How much current flows in a diode
   - What is the voltage at each node,…
2. R,C,L of all components at this operation point is determined
   - C of a capacitor can depend on voltage
   - internal resistance of a diode depends on current,…
3. ONLY these values are kept, i.e. the circuit is **linearized**
4. The transfer functions (to all nodes) are **calculated**

- AC Magnitude and Phase are used in sources
  - It make no difference weather you use 1 V or 1 mV, best use 1 V!

AC Analysis: Example