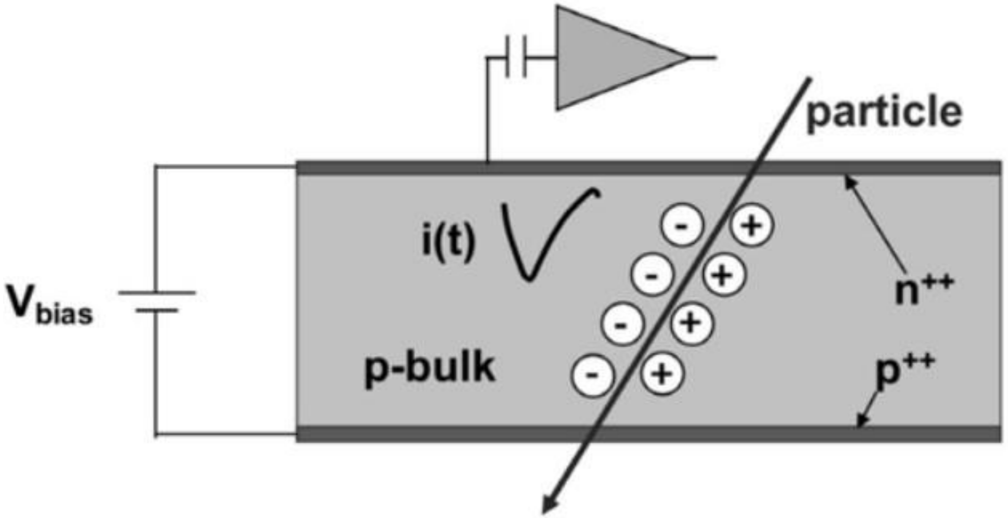
Electronics Simulation

Reporter: 李雁鹏(Jilin University)

OUTLINE:

- Why we need electronics?
- What circuit are we using?
- How to simulate the circuit using RASER?
- Practice

Why we need electronics?



We need electronics to amplify and convert the current signal from the detector into a voltage signal.

传感器 → 主放 → 数字化 → FPGA

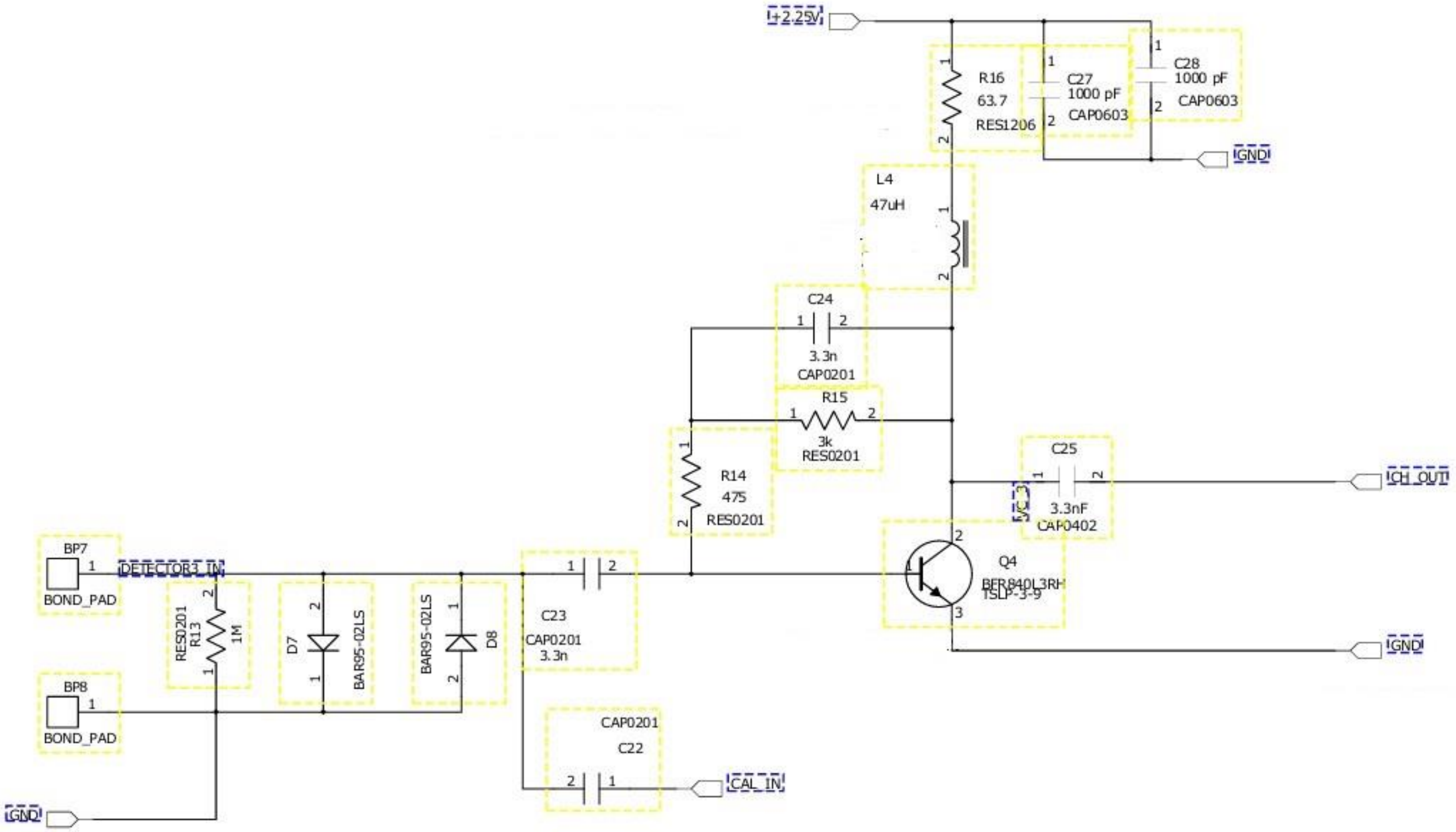
一般有一个前放

模拟信号→数字信号

数据处理

What circuit are we using?

UCSC read out board



How to simulate the circuit using RASER?

SPICE Introduction: What is SPICE?

SPICE是Simulation Program with Integrated Circuits Emphasis的缩写，由美国加利福尼亚大学伯克利(Berkeley)分校的电工和计算机科学系开发，骨干是Ron Rohrer和Larry Nagel，开始是使用FORTRAN语言设计的仿真软件，用于快速可靠地验证集成电路中的电路设计以及预测电路的性能。第一个版本SPICE1于1971年推出，通过围绕晶体管建立电流和电压变量来仿真电路的行为，称为模拟仿真或电路级仿真，且只能模拟100个晶体管的电路。
1975年SPICE2发布，开始正式实用化，1983年发布的SPICE2G.6在很长时间内都是工业标准，它包含超过15000条FORTRON语句，运行于多种中小型计算机上。1985年SPICE3推出，转为用C语言开发，易于运行于UNIX工作站，还增加了图形后处理工具和原理图工具，提供了更多的器件模型和分析功能。在1988年SPICE被定为美国国家标准。

Spice仿真器采用修改的节点分析法来建立电路方程组，提供非线性直流分析，非线性瞬态分析（实域分析）和线性小信号分析（频域分析）等。其中瞬态分析是最费时的验证方法，通常是利用数值积分法把非线性微分方程变成一组代数方程组，然后用高斯消去法来求解，因为这些线性方程仅仅在积分时刻点是有效的，而随着仿真器进展到下一个积分步长，积分方法必须重复来得到新的线性方程组，如果信号变化得特别快，积分步长应该取得非常小以便积分方法能收敛到正确的解，因此瞬态分析需要大量的数学操作。

https://blog.csdn.net/wxh0000mm/article/details/81329098

The NGspice is built into the RASER software.

How to simulate the circuit using RASER?

Step1: Create a '.cir' file

```
I1 2 0 pulse(0 -10u 0 0.1n 1n 0.00000001n 20n 0)
* input current source
VCC 6 0 dc 2.25
* VCC, DC source
Rin 2 0 1MEG
* resistance of the sensor
C6 2 0 20p
* capacitance of the sensor
C1 2 3 3.3n
x1 5 3 0 BFR840L3RHESD
* the amplifier BFR840L3RHESD
R1 3 4 4700
* feedback resistance
R2 4 5 3k
C2 4 5 3.3n
C5 5 out 3.3n
R4 out 0 50
L1 5 6 47U
```

```
.subckt BFR840L3RHESD 1 2 3
*
Rcx 15 1 1.57895
Rbx 25 2 1.92983
Rex 35 3 0.0800447
*
CBEPAR 22 33 1.9449E-013
CBCPAR 22 11 3.44161E-014
CCEPAR 11 33 2.24848E-013
LB     22 20 3.04259E-010
LC    11 10  2.88058E-010
CBEPCK 20 30  1E-014
CBCPCK 20 10  1.5502E-014
CCEPCK 10 30  1E-014
LBX     20 25 9.04631E-011
LEX    30 35 3.71422E-011
LCX    10 15  9.15043E-011
*
R_CS_npn 55 5 500
*
```

```
.control
tran 0.1p 20n
* <step> <stopping time>
wrdata ./t1.raw v(out)
* save raw data
.endc

.end
```

Example:/sicar/paras/T1.cir

BFR840L3RHESD: BFR840L3RHESD - Infineon Technologies

How to simulate the circuit using RASER?

Step2: Call ngspice for circuit simulation

In the sicar folder:

mkdir -p output/fig

```
bash-4.2$ mkdir -p optput/fig
bash-4.2$ ls
bjt_dd_0_SiC.msh  bjt_dd_0_Si.msh  cfg  ext  LICENSE  Makefile  optput  paras  raser  README.md  setting  tests
```

source cfg/setup.sh

raser-shell

```
bash-4.2$ source cfg/setup.sh
Setting up raser ...
bash-4.2$ raser-shell
Apptainer>
```

How to simulate the circuit using RASER?

Step2: Call ngspice for circuit simulation

Apptainer>ngspice

```
Apptainer> ngspice
******
** ngspice-34 : Circuit level simulation program
** The U. C. Berkeley CAD Group
** Copyright 1985-1994, Regents of the University of California.
** Copyright 2001-2020, The ngspice team.
** Please get your ngspice manual from http://ngspice.sourceforge.net/docs.html
** Please file your bug-reports at http://ngspice.sourceforge.net/bugrep.html
** Creation Date: Sun Jan 31 11:43:30 UTC 2021
******
```

ngspice 1 -> set ngbehavior=nomc
ngspice2 >source /scratchfs/atlas/liyanpeng/sicar/paras/T1·cir

```
ngspice 1 -> set ngbehavior=nomc
ngspice 2 -> source /scratchfs/atlas/liyanpeng/sicar/paras/T1.cir

Circuit: t1 circuit
```

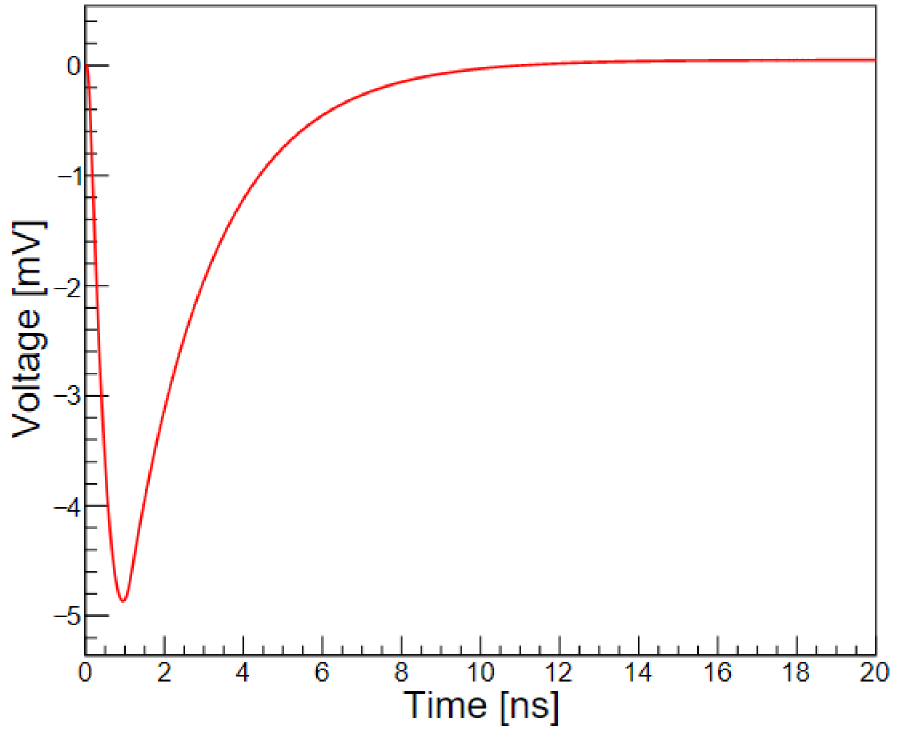
ngspice3->exit

How to simulate the circuit using RASER?

Step3: Plot

Apptainer> python raser/elec/ngspice_get_fig.py t1.raw

```
bash-4.2$ raser-shell
Apptainer> python raser/elec/ngspice_get_fig.py t1.raw
sh: 1: sed: not found
Info in <TCanvas::Print>: pdf file ./output/fig/t1.pdf has been created
figure  has been saved in  ./output/fig/t1.pdf
```



Apptainer> exit

hand-on practice

Follow the steps on slides 7 to 9 of the PPT to obtain the output signal of T1 by yourself.

- cd ~/tutorial
- 把T1.cir从分享的网盘里下载到本地再复制到虚拟机的~/tutorial/paras/下