数字大规模集成电路设计

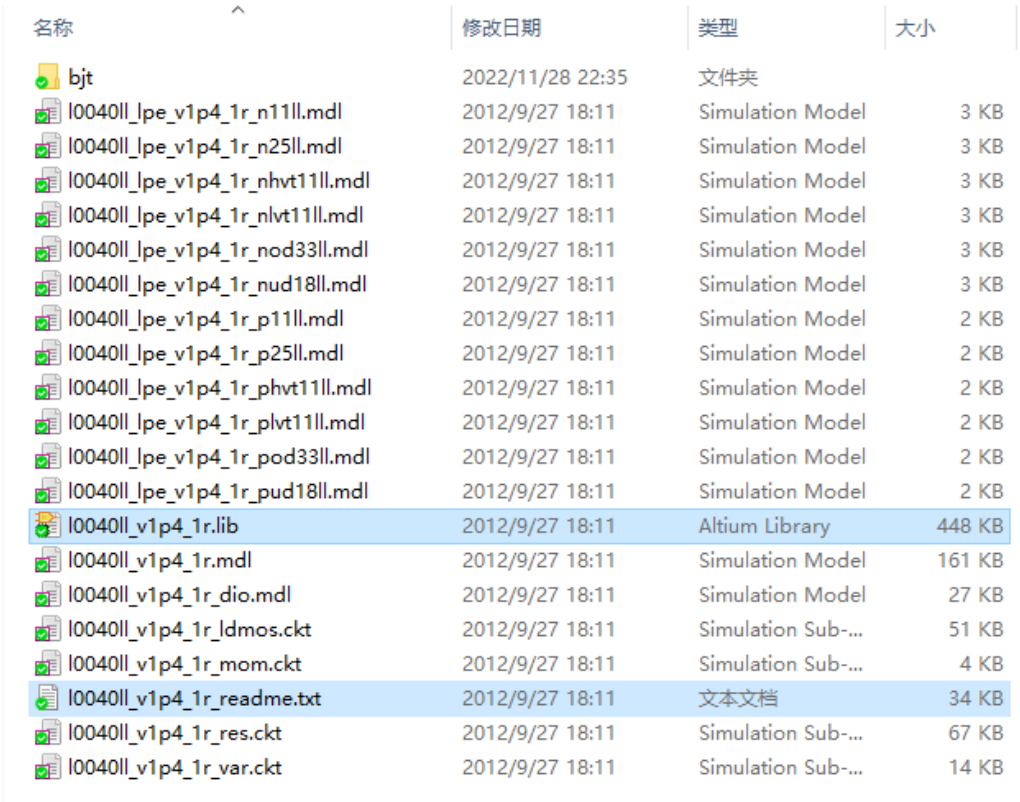
Hspice习题课

孙雅倩
syq19@mails.tsinghua.edu.cn

库文件

目录：/data2/class/lxy/lxy120/Documents/project2023/SMIC40NLL/models/hspice

l0040ll_v1p4_1r.lib	Hspice库文件
l0040ll_v1p4_1r_readme.txt	库文件使用说明
xxx.mdl	模型参数文件



名称	修改日期	类型	大小
bjt	2022/11/28 22:35	文件夹	
l0040ll_lpe_v1p4_1r_n11ll.mdl	2012/9/27 18:11	Simulation Model	3 KB
l0040ll_lpe_v1p4_1r_n25ll.mdl	2012/9/27 18:11	Simulation Model	3 KB
l0040ll_lpe_v1p4_1r_nhvt11ll.mdl	2012/9/27 18:11	Simulation Model	3 KB
l0040ll_lpe_v1p4_1r_nlvt11ll.mdl	2012/9/27 18:11	Simulation Model	3 KB
l0040ll_lpe_v1p4_1r_nod33ll.mdl	2012/9/27 18:11	Simulation Model	3 KB
l0040ll_lpe_v1p4_1r_nud18ll.mdl	2012/9/27 18:11	Simulation Model	3 KB
l0040ll_lpe_v1p4_1r_p11ll.mdl	2012/9/27 18:11	Simulation Model	2 KB
l0040ll_lpe_v1p4_1r_p25ll.mdl	2012/9/27 18:11	Simulation Model	2 KB
l0040ll_lpe_v1p4_1r_phvt11ll.mdl	2012/9/27 18:11	Simulation Model	2 KB
l0040ll_lpe_v1p4_1r_plvt11ll.mdl	2012/9/27 18:11	Simulation Model	2 KB
l0040ll_lpe_v1p4_1r_pod33ll.mdl	2012/9/27 18:11	Simulation Model	2 KB
l0040ll_lpe_v1p4_1r_pud18ll.mdl	2012/9/27 18:11	Simulation Model	2 KB
l0040ll_v1p4_1r.lib	2012/9/27 18:11	Altium Library	448 KB
l0040ll_v1p4_1r.mdl	2012/9/27 18:11	Simulation Model	161 KB
l0040ll_v1p4_1r_dio.mdl	2012/9/27 18:11	Simulation Model	27 KB
l0040ll_v1p4_1r_ldmos.ckt	2012/9/27 18:11	Simulation Sub-...	51 KB
l0040ll_v1p4_1r_mom.ckt	2012/9/27 18:11	Simulation Sub-...	4 KB
l0040ll_v1p4_1r_readme.txt	2012/9/27 18:11	文本文档	34 KB
l0040ll_v1p4_1r_res.ckt	2012/9/27 18:11	Simulation Sub-...	67 KB
l0040ll_v1p4_1r_var.ckt	2012/9/27 18:11	Simulation Sub-...	14 KB

Hspice代码示例

```
.title test
.options list node post=2 probe
.temp 25
```

标题与仿真设置

```
** add library section
.protect
.lib '/data2/class/lxy/lxy104/2022Project/
+lib/smic40ll_1125_2tm_oa_cds_1P5M_2012_10_11_v1.4/
+models/hspice/1004011_v1p4_1r.lib' TT
.unprotect
```

引用工艺库

```
.global vdd! gnd!
```

可定义全局节点

```
.include inv.sp
```

引用子模块文件

```
X_inv1 in nin INV_SYQ
X_inv2 nin out INV_SYQ f=4
```

调用子模块

inv.sp

```
.subckt INV_SYQ in out Wmin=120n Lmin=40n f=1
X_Mp out in vdd! vdd! p1111_ckt W='Wmin*2*f' L=Lmin
X_Mn out in gnd! gnd! n1111_ckt W='Wmin*f' L=Lmin
.ends INV_SYQ
```

```
** add output load
.param  cl=20f
Cload   out 0 c=cl
```

添加负载

```
** add power supply
.param  pwsp=1.0
Vddgl   vdd!    0   dc=pwsp
Vddgnd  gnd!    0       dc=0
```

定义电源/地

```
** add stimulition
.param  fclk = 200k tper='1/fclk'
.param  ts=10p
Vclk    in  0   pulse(0 pwsp 'tper-ts/2' ts ts 'tper/2-ts' tper)
```

定义输入信号

```
** add simulation command
.tran 1n  'tper*101'
.op
```

仿真命令

```
.probe tran v(vdd!) v(gnd!) v(in) v(nin) v(out)
.measure tran avg_power avg p(vddgl) from=0 to='tper*100'
```

选择输出

```
.alter
.param fclk=2Meg
.param pwsp=1.1V
```

可以更改参数
再进行一次仿真

仿真设置

```
.title test
.options list node post=2 probe
```

标题与仿真设置

```
** add library section
.protect
.lib '/data2/class/lxy/lxy104/2022Project/
+lib/smic40ll_1125_2tm_oa_cds_1P5M_2012_10_11_v1.4/
+models/hspice/1004011_v1p4_1r.lib' TT
.unprotect

.global vdd! gnd!

.include inv.sp
X_inv1 in nin INV_SYQ
X_inv2 nin out INV_SYQ f=4
```

参考Hspice_User_Guide
—— Chapter9 Simulation Options

POST = *x*

Stores simulation results for analysis, using the AvanWaves graphical interface or other methods.

- `POST = 1` saves the results in binary.
- `POST = 2` saves the results in ASCII format.
- `POST = 3` saves the results in New Wave binary format.

PROBE

Limits the post-analysis output to just the variables designated in `.PROBE`, `.PRINT`, `.PLOT`, and `.GRAPH` statements. By default, Star-Hspice outputs all voltages and power supply currents, in addition to variables listed in `.PROBE`/`.PRINT`/`.PLOT`/`.GRAPH` statements. `PROBE` significantly decreases the size of simulation output files.

仿真设置

```
.title test
.options list node post=2 probe
```

标题与仿真设置

```
** add library section
.protect
.lib '/data2/class/lxy/lxy104/2022Project/
+lib/smic40ll_1125_2tm_oa_cds_1P5M_2012_10_11_v1.4/
+models/hspice/l004oll_v1p4_1r.lib' TT
.unprotect

.global vdd! gnd!

.include inv.sp
X_inv1 in nin INV_SYQ
X_inv2 nin out INV_SYQ f=4
```

参考Hspice_User_Guide
—— Chapter9 Simulation Options

语名	功能
.option post	输出高分辨率图形
.option post probe	只用高分辨率输出 probe 语句指明的变量
.option postlvl=0..6	将几级子电路的内部节点输出到高分辨率图形
.option brief	输出文本省略某些内容
.option list	输出元器件列表
.option nomod	不输出模型库相关信息
.option dccap	DC仿真时计算 C-V 特性

语名	功能
.option runlvl=0..6	仿真精度：6 精度最高，0 速度最快
.option dcic=0|1	控制瞬态仿真时是否使用.IC语句指定的初值
.option scale=xxx	设置全局器件参数的比例值
.option defl=0.18u	设置略省MOSFET沟道长度

引用库文件

```
.title test
.options list node post=2 probe

** add library section
.protect
.lib '/data2/class/lxy/lxy104/2022Project/
+lib/smic40ll_1125_2tm_oa_cds_1P5M_2012_10_11_v1.4/
+models/hspice/l004oll_v1p4_1r.lib' TT
.unprotect

.global vdd! gnd!

.include inv.sp
X_inv1 in nin INV_SYQ
X_inv2 nin out INV_SYQ f=4
```

引用工艺库

- 用加号（+）表示续行，此时加号应该是新续之行的第一个非数字、非空格字符；
- 星号（*）和美圆符号（$）可以引出注释行，但*必须是每行第一个字母，而$一般跟在一个语句后，并与语句有至少一个空格。

子电路

```
.title test
.options list node post=2 probe

** add library section
.protect
.lib '/data2/class/lxy/lxy104/2022Project/
+lib/smic40ll_1125_2tm_oa_cds_1P5M_2012_10_11_v1.4/
+models/hspice/l004oll_v1p4_1r.lib' TT
.unprotect

.global vdd! gnd!
```

可定义全局节点

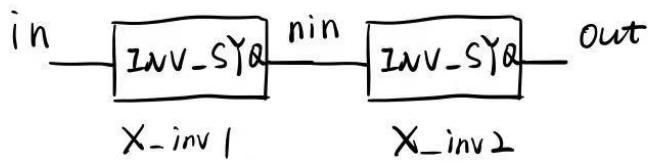
```
.include inv.sp
X_inv1 in nin INV_SYQ
X_inv2 nin out INV_SYQ f=4
```

引用子模块文件

调用子模块

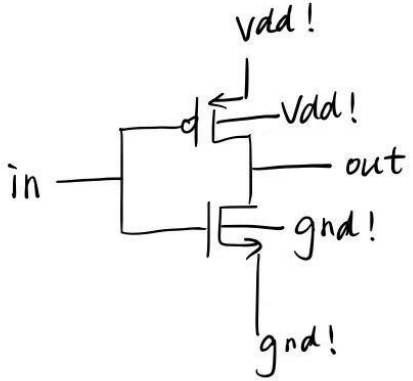
子模块文件inv.sp
定义了一个反相器，名字是INV_SYQ

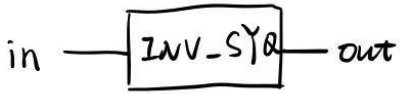
顶层代码中，引用了INV_SYQ，搭建了一个buffer



inv.sp

```
.subckt INV_SYQ in out Wmin=120n Lmin=40n f=1
X_Mp out in vdd! vdd! p1lll_ckt W='Wmin*2*f' L=Lmin
X_Mn out in gnd! gnd! n1lll_ckt W='Wmin*f' L=Lmin
.ends INV_SYQ
```





元件引用

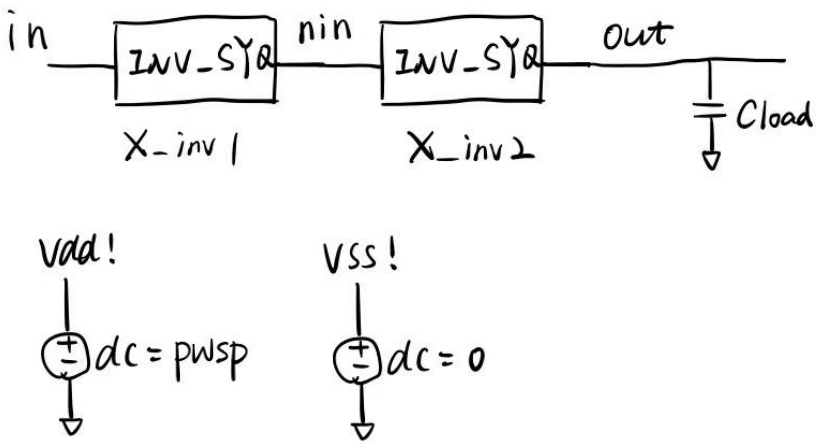
```
** add output load
.param  cl=20f
Cload   out 0 c=cl
```

添加负载

```
** add power supply
.param  pwsp=1.0
Vddgl   vdd!    0   dc=pwsp
Vddgnd  gnd!    0       dc=0
```

定义电源/地

```
** add stimulition
.param  fclk = 200k tper='1/fclk'
.param  ts=10p
Vclk    in  0   pulse(0 pwsp 'tper-ts/2' ts ts 'tper/2-ts' tper)

** add simulation command
.tran 1n  'tper*101'
.op


.probe tran v(vdd!) v(gnd!) v(in) v(nin) v(out)
.measure tran avg_power avg p(vddgl) from=0 to='tper*100'

.alter
.param fclk=2Meg
.param pwsp=1.1V
```



参考Hspice_User_Guide
—— Chapter4 Elements

电阻：Rxxx n1 n2 <mname> <R = >resistance
电容：Cxxx n1 n2 <mname> <C=>capacitance
BJT：Qxxx nc nb ne <ns> mname <area> <OFF>
MOS：Mxxx nd ng ns <nb> mname <<L = >length> <<W = >width>
子电路：Xxxx <node1 node2 ···> SUBNAME

注意，本次课程设计使用的工艺库中，MOS管被定义成了子电路，因此调用MOS管要用X开头而不是M开头

元件引用

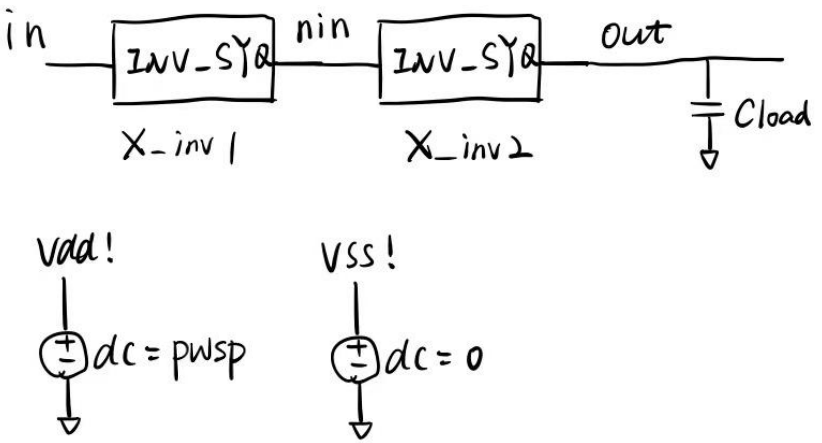
```
** add output load
.param  cl=20f
Cload   out 0 c=cl
```

添加负载

```
** add power supply
.param  pwsp=1.0
Vddgl   vdd!    0   dc=pwsp
Vddgnd  gnd!    0       dc=0
```

定义电源/地

```
** add stimulition
.param  fclk = 200k tper='1/fclk'
.param  ts=10p
Vclk    in  0   pulse(0 pwsp 'tper-ts/2' ts ts 'tper/2-ts' tper)

** add simulation command
.tran 1n  'tper*101'
.op


.probe tran v(vdd!) v(gnd!) v(in) v(nin) v(out)
.measure tran avg_power avg p(vddgl) from=0 to='tper*100'

.alter
.param fclk=2Meg
.param pwsp=1.1V
```

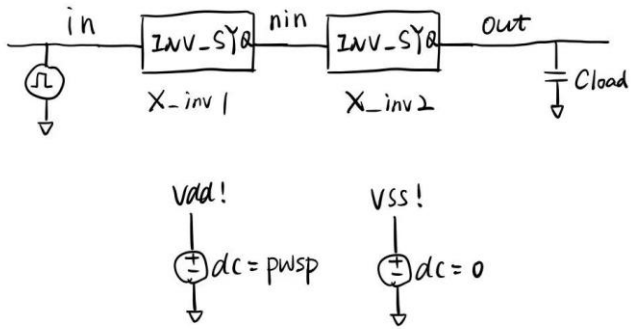


参考Hspice_User_Guide
—— Chapter4 Elements

关键字母	元件类形
R	电阻
C	电容
L	电感
M	MOSFET
Q	BJT
J	JFET or MESFET
D	Diode
X	子电路调用

激励源

```
** add output load
.param  cl=20f
Cload   out 0 c=cl

** add power supply
.param  pwsp=1.0
Vddgl   vdd!    0   dc=pwsp
Vddgnd  gnd!    0       dc=0
```




```
** add stimulition
.param  fclk = 200k tper='1/fclk'
.param  ts=10p
Vclk    in  0   pulse(0 pwsp 'tper-ts/2' ts ts 'tper/2-ts' tper)
```

定义输入信号

```
** add simulation command
.tran 1n  'tper*101'
.op


.probe tran v(vdd!) v(gnd!) v(in) v(nin) v(out)
.measure tran avg_power avg p(vddgl) from=0 to='tper*100'

.alter
.param fclk=2Meg
.param pwsp=1.1V
```

参考Hspice_User_Guide
—— Chapter5 Using Sources and Stimuli

直流电压源

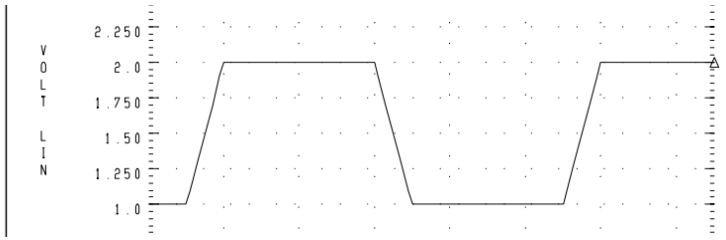
```
Vxxx n+ n- <<DC=> dcval> <tranfun> <AC=acmag>
+ <acphase>>
```

直流电流源

```
Iyyy n+ n- <<DC=> dcval> <tranfun> <AC=acmag>
+ <acphase>> <M=val>
```

脉冲源

```
Vxxx n+ n- PU<LSE> <(>v1 v2 <td <tr <tf <pw
+ <per>>>>> <)>
```

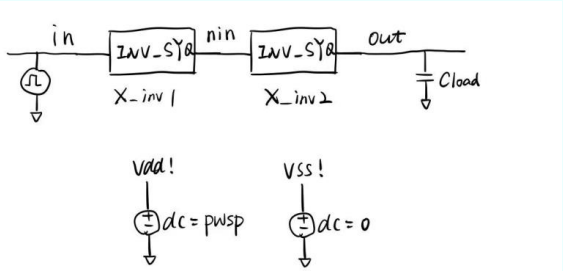


关键字母	元件类形
V	独立电压源
I	独立电流源
E	压控电压源
F	流控电流源
G	压控电流源
H	流控电压源

仿真类型

```
** add output load
.param   cl=20f
Cload    out 0 c=cl

** add power supply
.param   pwsp=1.0
Vddgl    vdd!    0    dc=pwsp
Vddgnd   gnd!    0         dc=0


** add stimulition
.param   fclk = 200k tper='1/fclk'
.param   ts=10p
Vclk     in   0    pulse(0 pwsp 'tper-ts/2' ts ts 'tper/2-ts' tper)

** add simulation command
.tran 1n   'tper*101'
.op


.probe tran v(vdd!) v(gnd!) v(in) v(nin) v(out)
.measure tran avg_power avg p(vddgl) from=0 to='tper*100'

.alter
.param fclk=2Meg
.param pwsp=1.1V
```

仿真命令



参考Hspice_User_Guide
—— Chapter10 Initializing DC/Operating Point Analysis
—— Chapter11 Transient Analysis
—— Chapter12 AC Sweep and Small Signal Analysis

瞬态仿真语法

```
.TRAN tincr1 tstop1 <tincr2 tstop2 ...tincrN tstopN>
+ <START = val> <UIC>
```

.TRAN Examples

1. The following example performs and prints the transient analysis, every 1 ns, for 100 ns.

   ```
   .TRAN 1NS 100NS
   ```

2. The following example performs the calculation every 0.1 ns, for the first 25 ns; and then every 1 ns, until 40 ns. Printing and plotting begin at 10 ns.

   ```
   .TRAN .1NS 25NS 1NS 40NS START = 10NS
   ```

3. The following example performs the calculation every 10 ns, for 1 µs. This example bypasses the initial DC operating point calculation. It uses the nodal voltages, specified in the .IC statement (or by IC parameters in element statements), to calculate the initial conditions.

   ```
   .TRAN 10NS 1US UIC
   ```

4. The following example increases the temperature by 10 °C, through the range -55 °C to 75 °C. It also performs transient analysis, for each temperature.

   ```
   .TRAN 10NS 1US UIC SWEEP TEMP -55 75 10
   ```

结果的输出

参考Hspice_User_Guide
—— Chapter8 Simulation Output

```
** add output load
.param  cl=20f
Cload   out 0 c=cl

** add power supply
.param  pwsp=1.0
Vddgl   vdd!    0   dc=pwsp
Vddgnd  gnd!    0       dc=0


** add stimulition
.param  fclk = 200k tper='1/fclk'
.param  ts=10p
Vclk    in  0   pulse(0 pwsp 'tper-ts/2' ts ts 'tper/2-ts' tper)

** add simulation command
.tran 1n  'tper*101'
.op


.probe tran v(vdd!) v(gnd!) v(in) v(nin) v(out)
.measure tran avg_power avg p(vddgl) from=0 to='tper*100'

.alter
.param fclk=2Meg
.param pwsp=1.1V
```

选择输出

.PROBE	Outputs data to post-processor output files, but not to the output listing (used with .OPTION PROBE, to limit output).
.MEASURE	Prints the results of specific user-defined analyses (and post-processor data, if you specify .OPTION POST), to the output listing file. You can use the .MEASURE statement in Star-Hspice.

Fundamental measurement modes in Star-Hspice are:

- Rise, fall, and delay
- Find-when
- Equation evaluation
- Average, RMS, min, max, and peak-to-peak
- Integral evaluation
- Derivative evaluation
- Relative error

Alter语句

```
** add output load
.param  cl=20f
Cload   out 0 c=cl

** add power supply
.param  pwsp=1.0
Vddgl   vdd!    0   dc=pwsp
Vddgnd  gnd!    0       dc=0


** add stimulition
.param  fclk = 200k tper='1/fclk'
.param  ts=10p
Vclk    in  0   pulse(0 pwsp 'tper-ts/2' ts ts 'tper/2-ts' tper)

** add simulation command
.tran 1n  'tper*101'
.op


.probe tran v(vdd!) v(gnd!) v(in) v(nin) v(out)
.measure tran avg_power avg p(vddgl) from=0 to='tper*100'

.alter
.param fclk=2Meg
.param pwsp=1.1V
```

可以更改参数，重复仿真

参考Hspice_User_Guide
——Chapter 3 Simulation Input and Controls
—— Input Netlist File Composition
—— ALTER Statement

参数的使用

参考Hspice_User_Guide
—— Chapter7 Parameters and Functions

You can assign the following types of values to parameters:

- A constant real number.
- An algebraic expression of real values.
- A predefined function.
- A function that you define.
- A circuit value.
- A model value.

```
.PARAM Pi           = '355/113'
.PARAM Pi2          = '2*Pi'

.PARAM npRatio      = 2.1
.PARAM nWidth       = 3u
.PARAM pWidth       = 'nWidth * npRatio'
Mp1                 ... <pModelName> W = pWidth
Mn1                 ... <nModelName> W = nWidth
...

.PRINT DC v(3) gain = PAR('v(3)/v(2)')
```

Hspice的单位

符号	数量级
F(f)	1e-15
P(p)	1e-12
N(n)	1e-9
U(u)	1e-6
M(m)	1e-3
K(k)	1e+3
MEG(meg)	1e+6
G(g)	1e+9
T(t)	1e+12
DB(db)	$20\log_{10}$

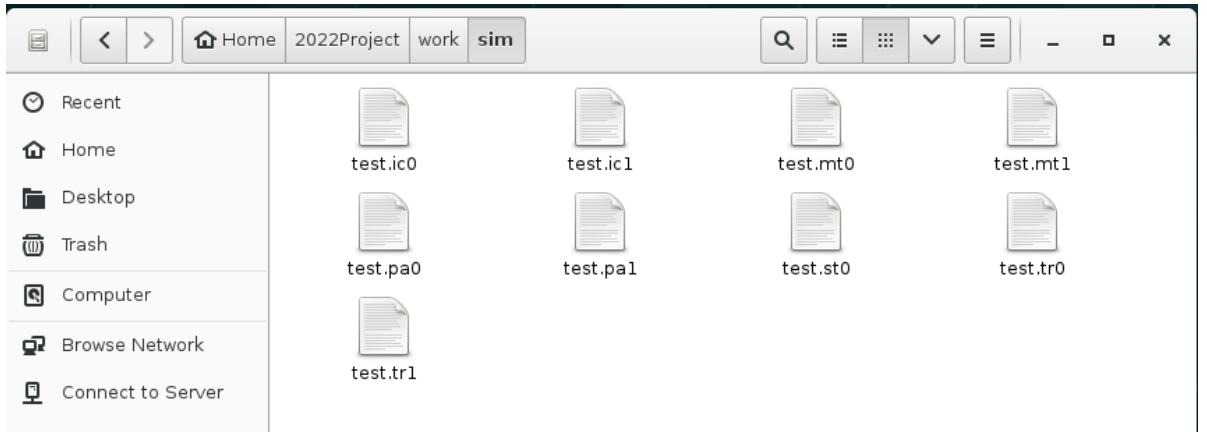
注：SPICE不区分大小写字母

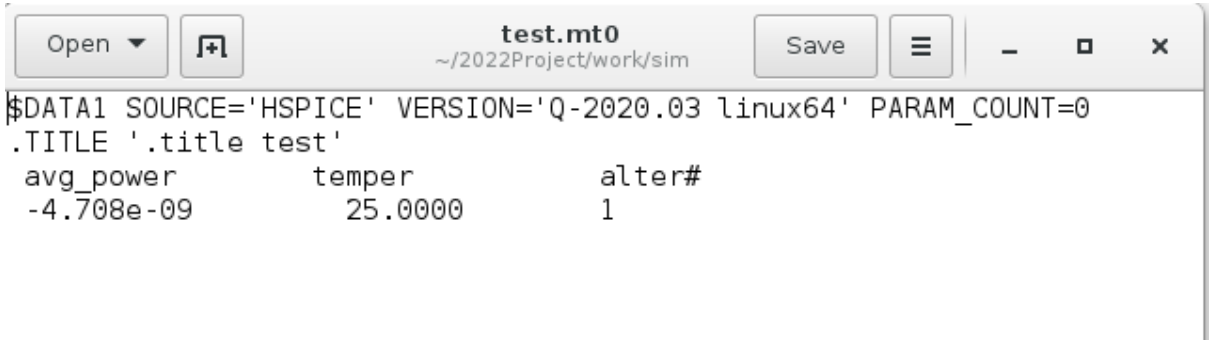
Hspice代码运行

运行命令：hspice …/…/xxx.sp

输入网表文件	**.sp**
输出列表文件	**.lis**
瞬态分析文件	**.tr#**
直流分析文件	**.sw#**
交流分析文件	**.ac#**
初始状态文件	.ic#
子电路路径表	.pa#
测量结果文件	.mt#

图象输出





波形查看器 WaveViewer

运行命令：wv