
附录：软件入门指导

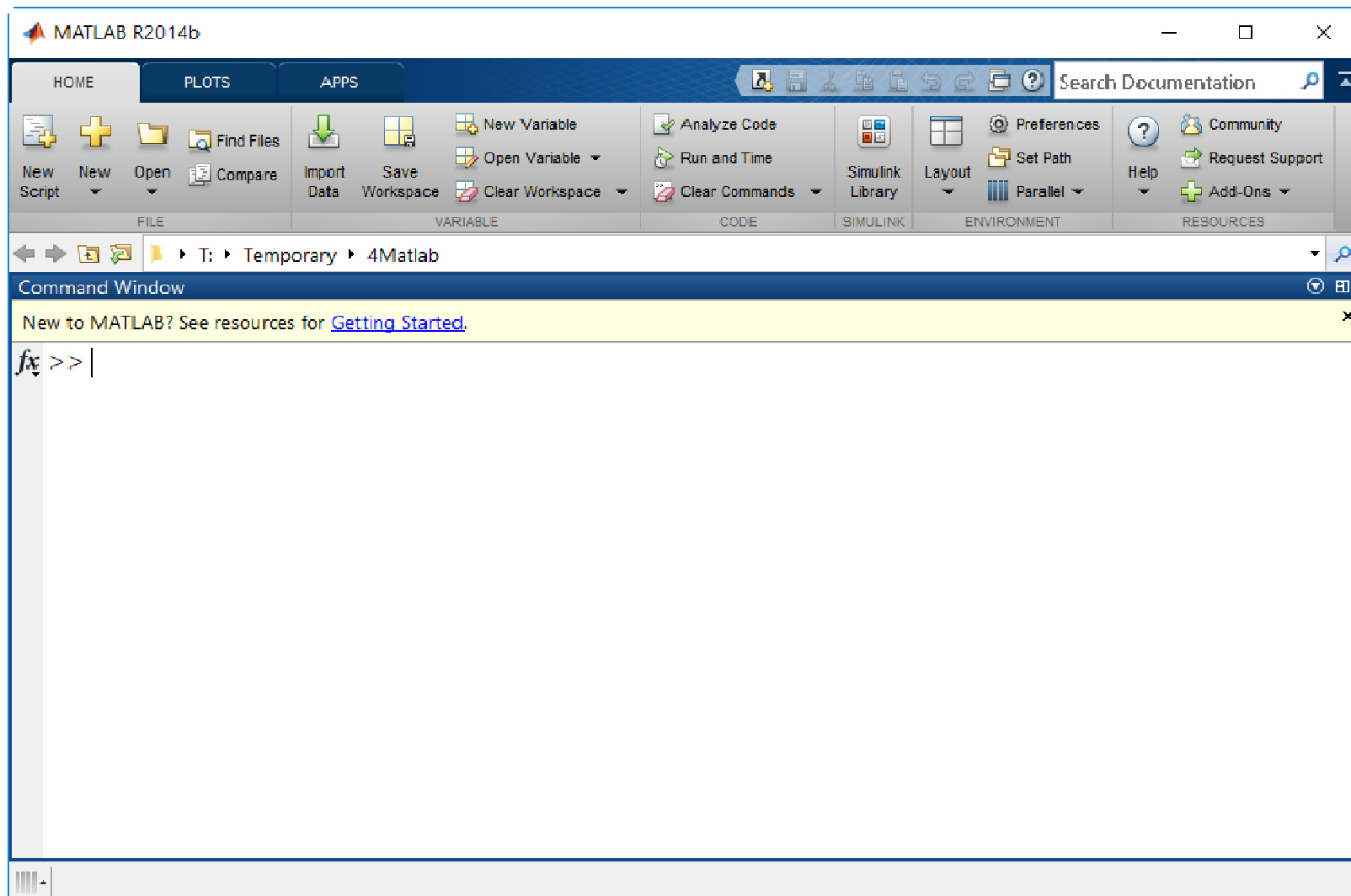
工具软件

- Matlab: Mathworks公司
- Multisim: NI公司
- HSPICE: Synopsys公司

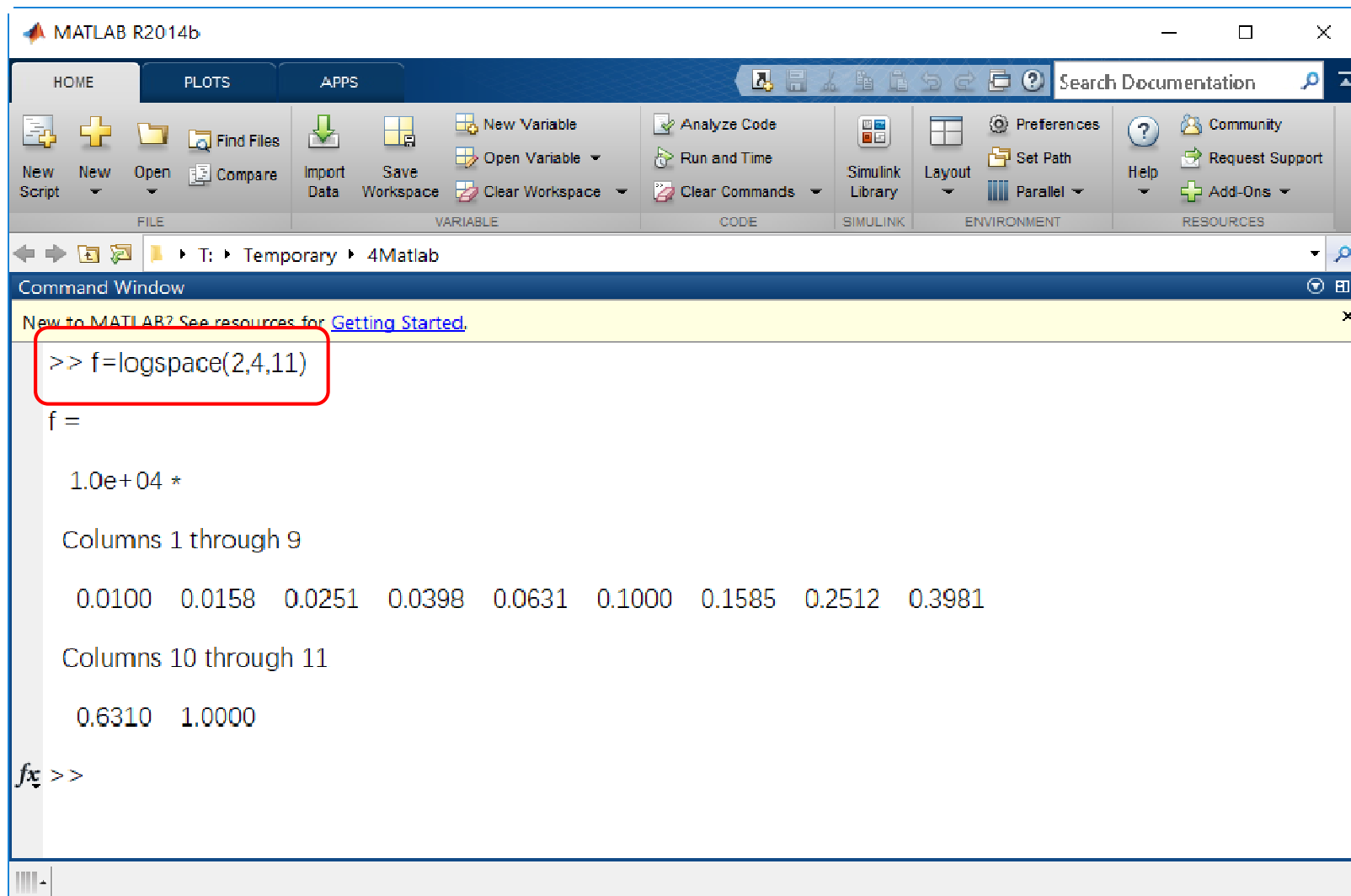
MATLAB

- 科学计算软件
- 丰富的内置函数
- 大量的官方或第三方的工具箱
- simulink仿真环境

命令行



代码输入



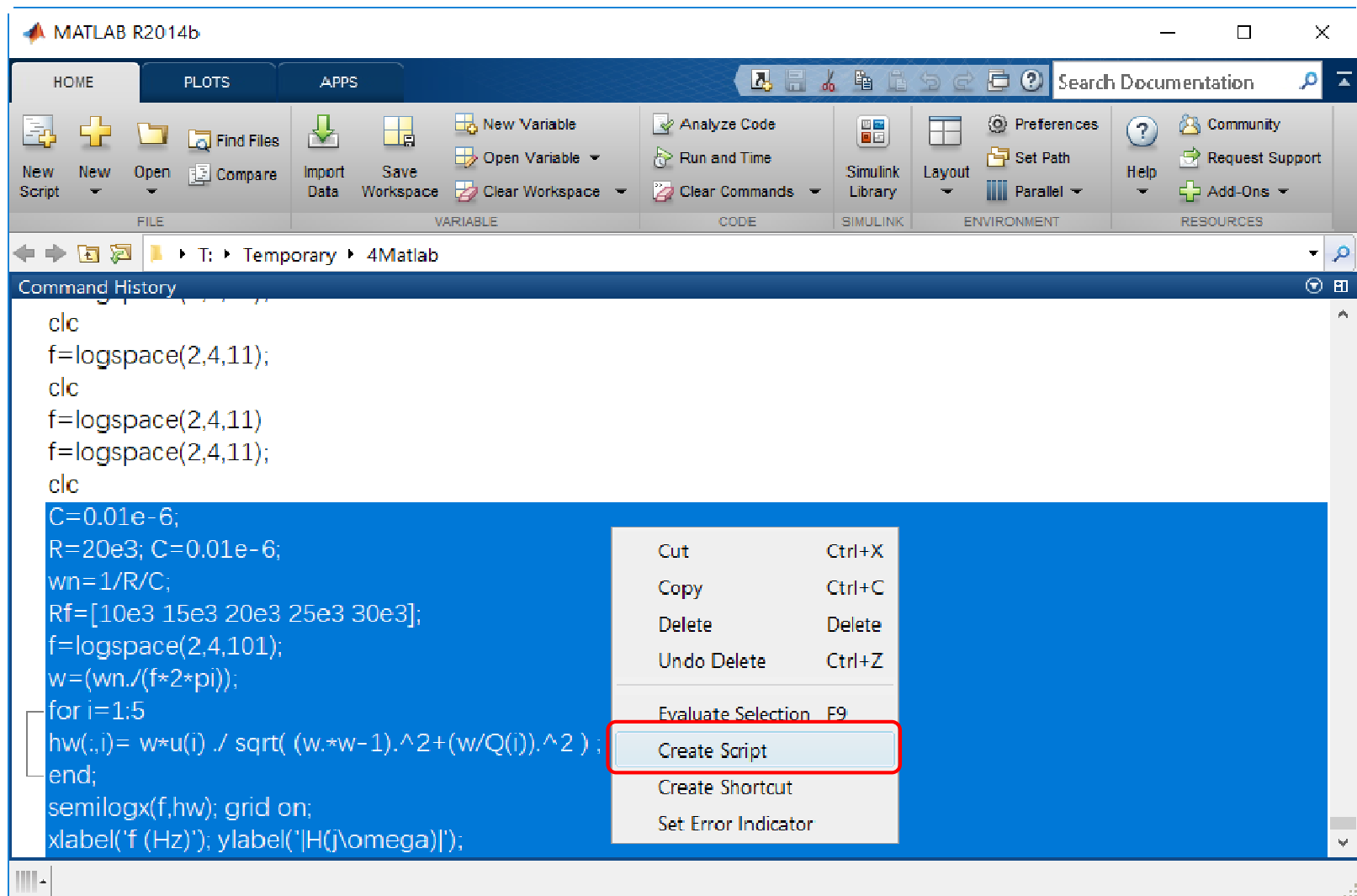
末尾分号

The image shows the MATLAB R2014b interface. The Command Window displays the following output for the command `f=logspace(2,4,11)`:

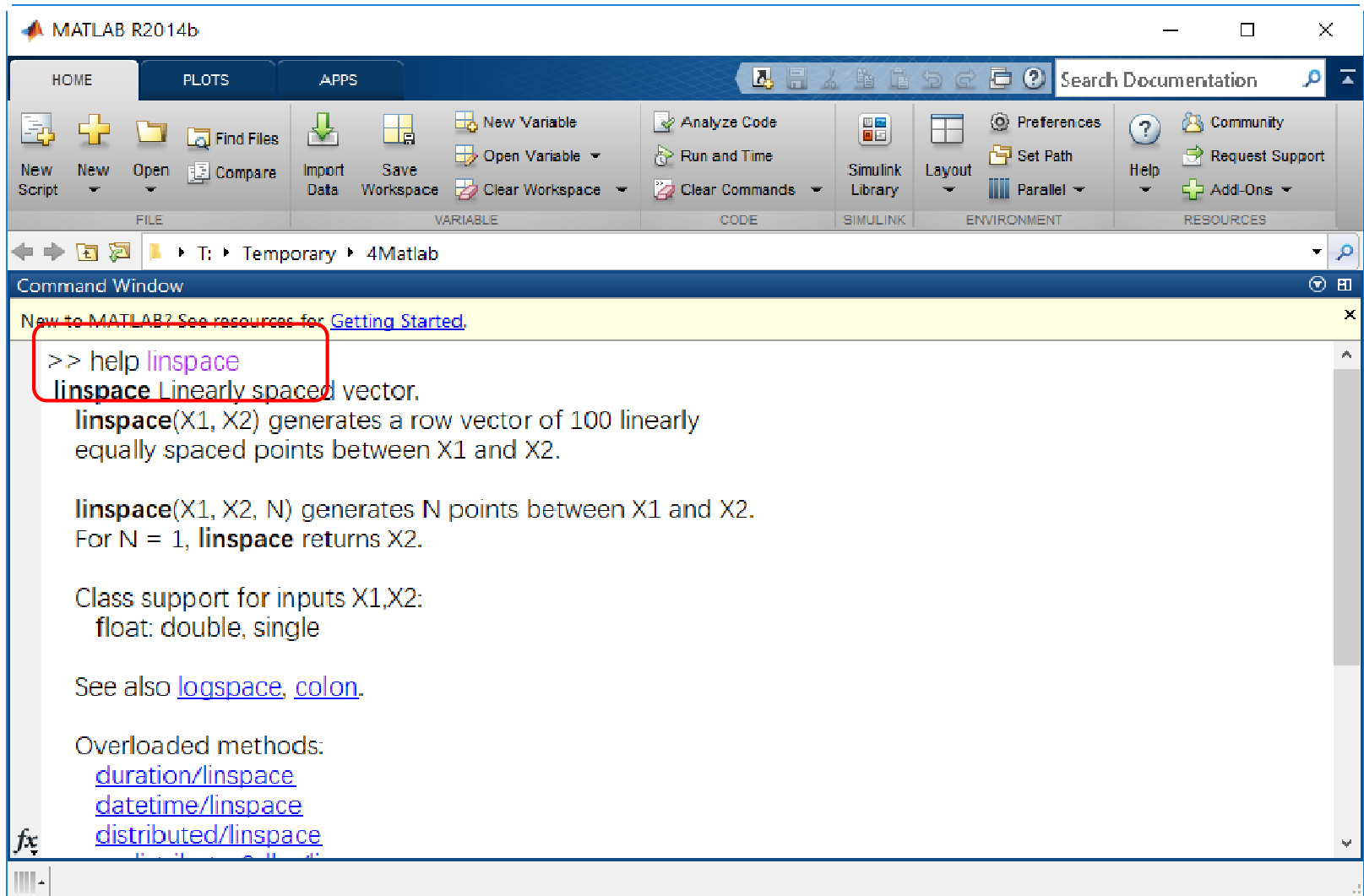
```
>> f=logspace(2,4,11)
f =
    1.0e+04 *
    Columns 1 through 9
    0.0100    0.0158    0.0251    0.0398    0.0631    0.1000    0.1585    0.2512    0.3981
    Columns 10 through 11
    0.6310    1.0000
```

Below this, the command `>> f=logspace(2,4,11);` is entered and highlighted with a red box. The output for this command is not shown, but it is implied to be suppressed.

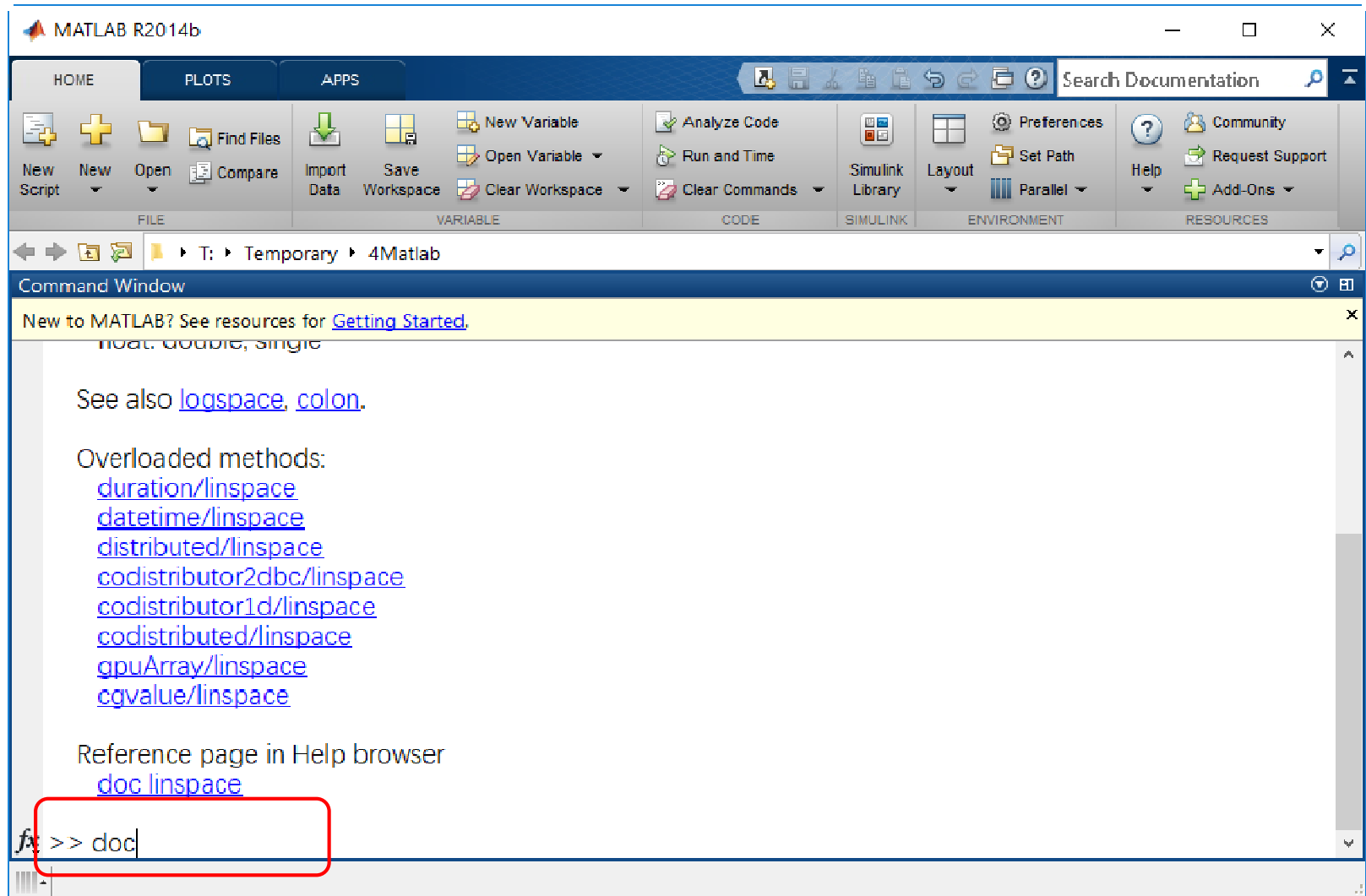
创建脚本



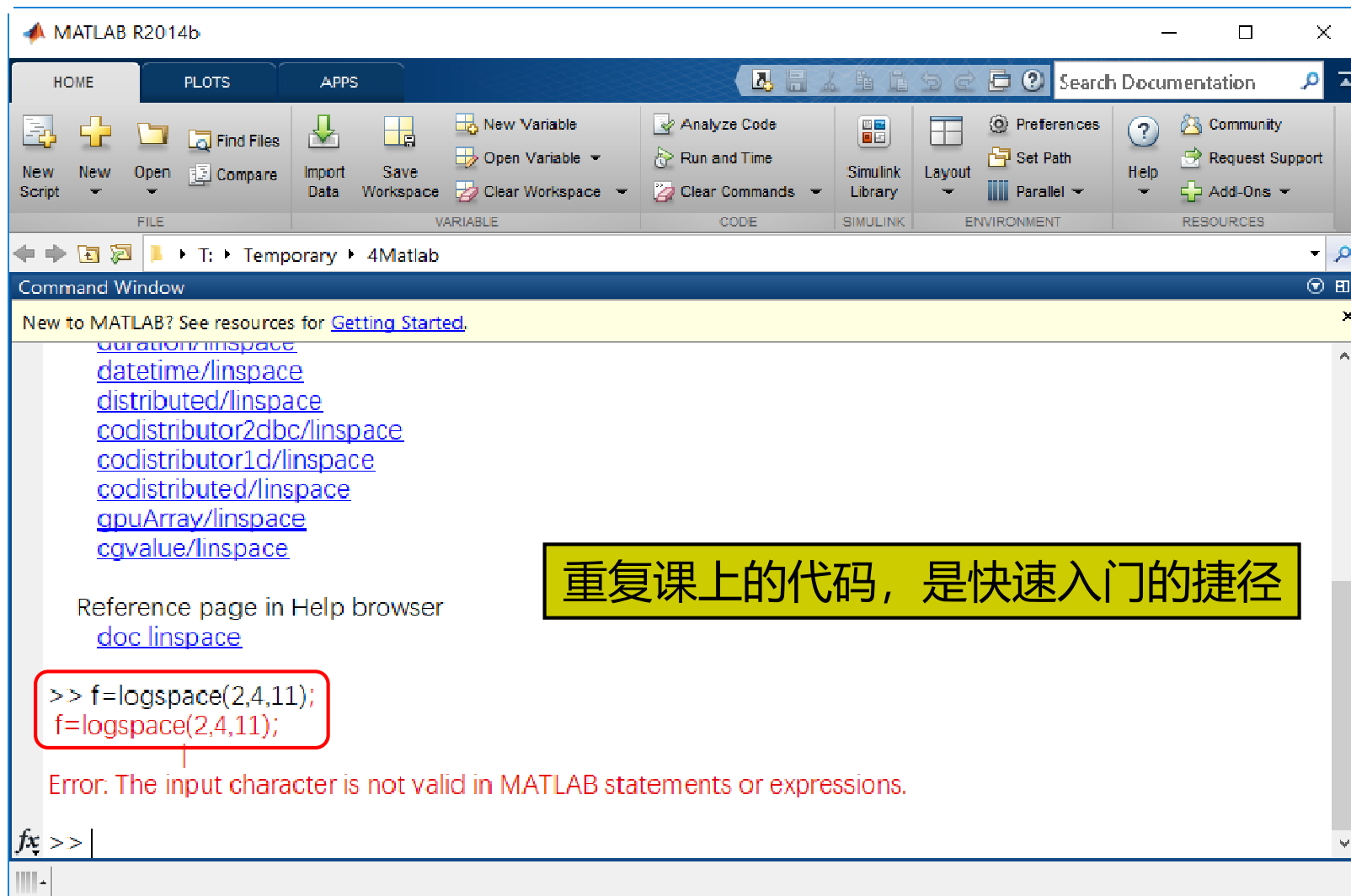
help命令



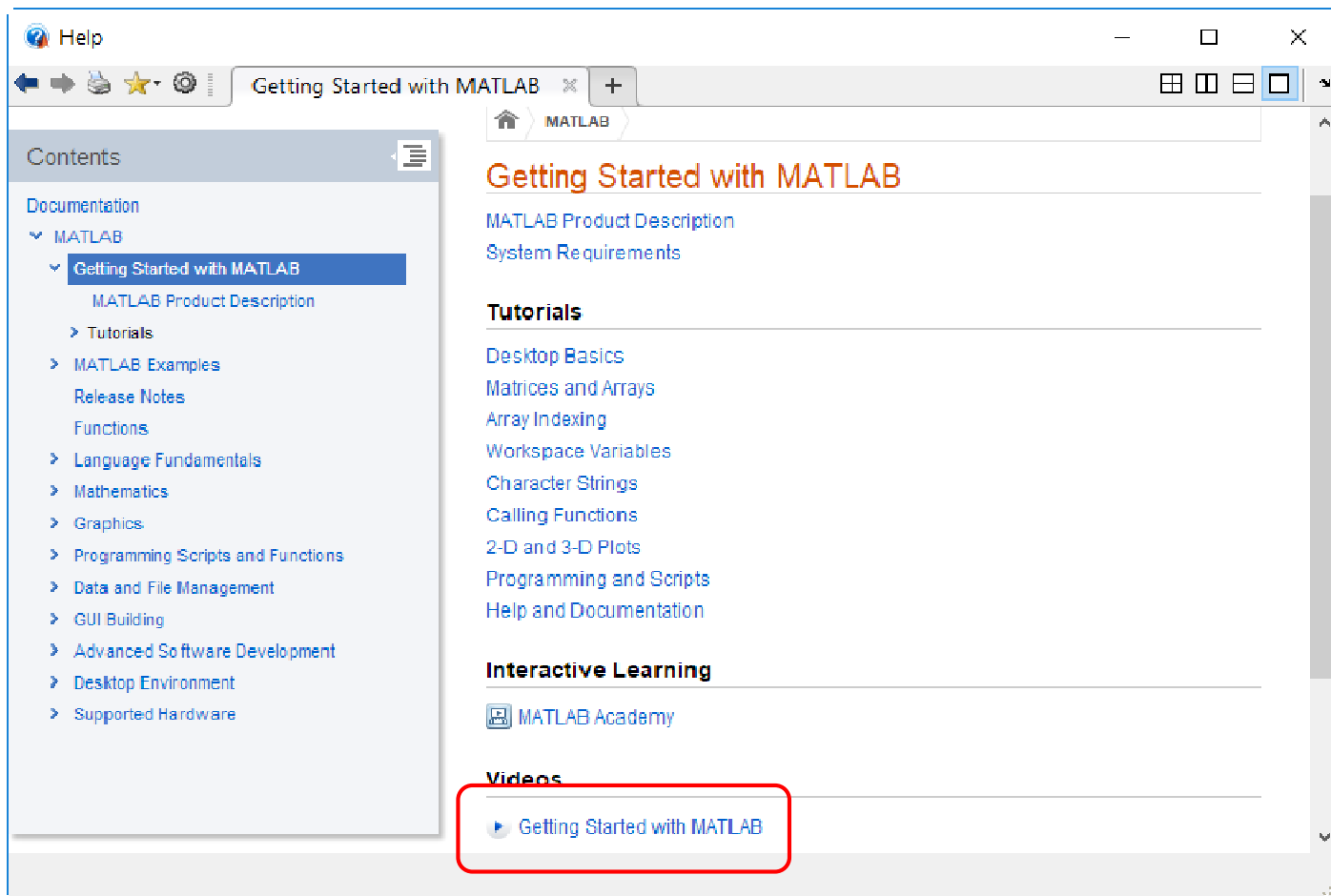
doc命令



代码出错示例



快速入门视频



SPICE

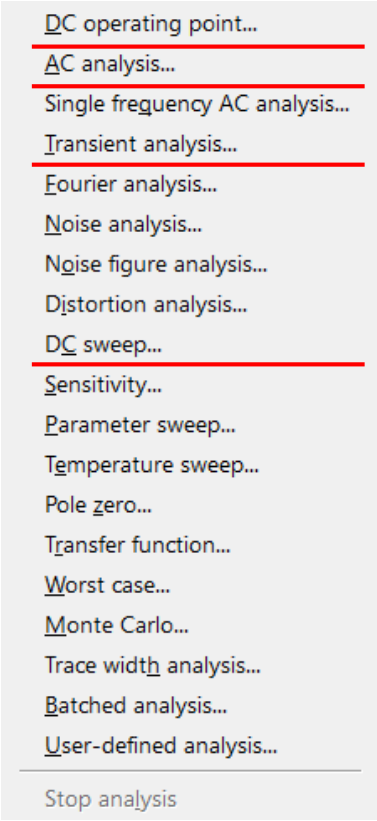
- Simulation Program with Integrated Circuit Emphasis
- 通用、开源的电路仿真器
- 既支持PCB板级电路、也支持集成电路仿真
- Multisim和HSPICE, 都属于基于SPICE的商用软件

电路仿真前的准备工作

- 测试哪些指标?
- 采用什么仿真?
- 具体测试电路?

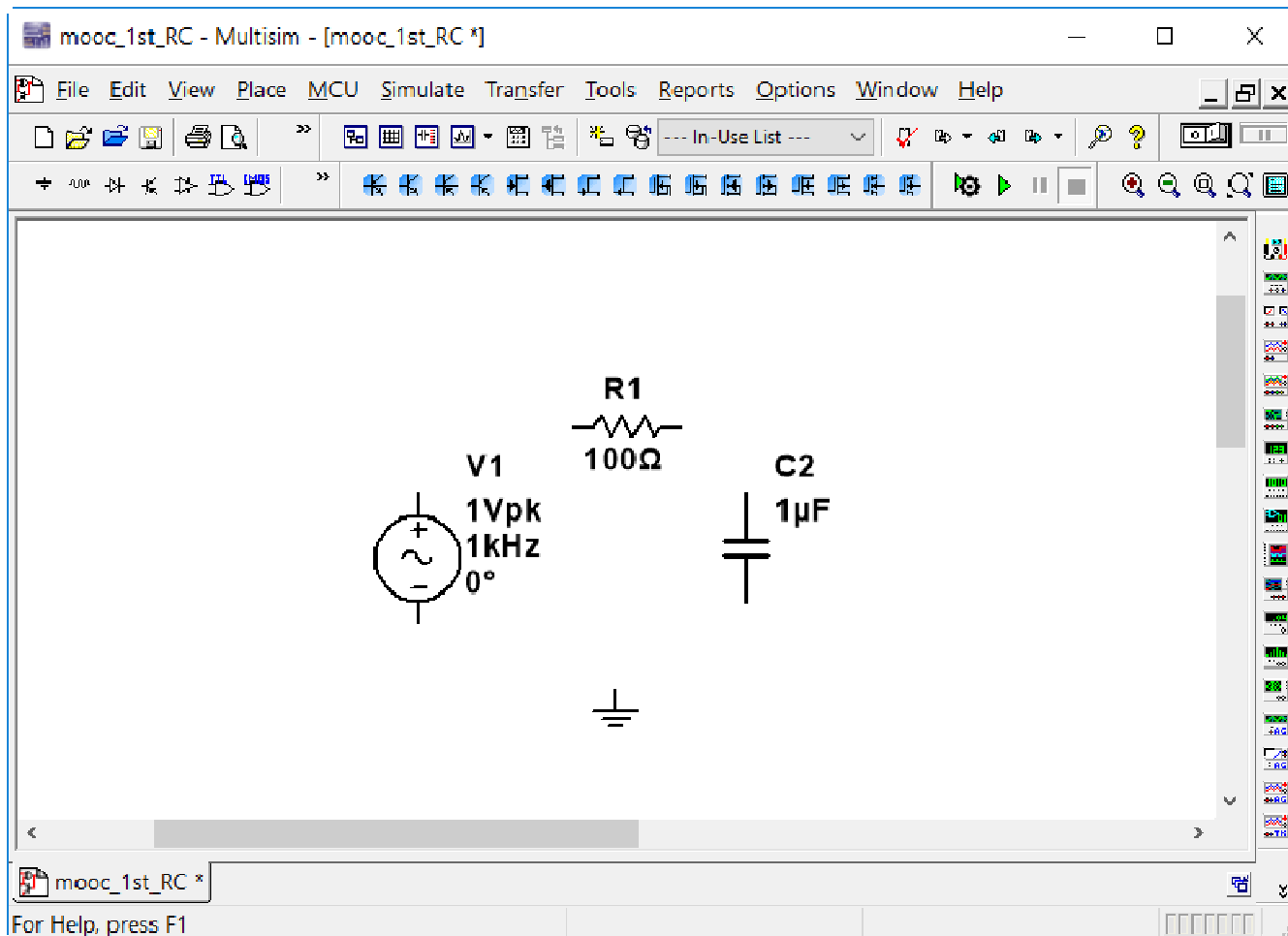
主要仿真类型及原理

- 直流仿真
 - 直流特性
 - 可参考 “线性电阻电路分析”
- 交流小信号仿真
 - 频域特性
 - 可参考 “CMOS共源放大电路I”
- 瞬态仿真
 - 时域特性
 - 可参考 “动态电路时域近似求解”

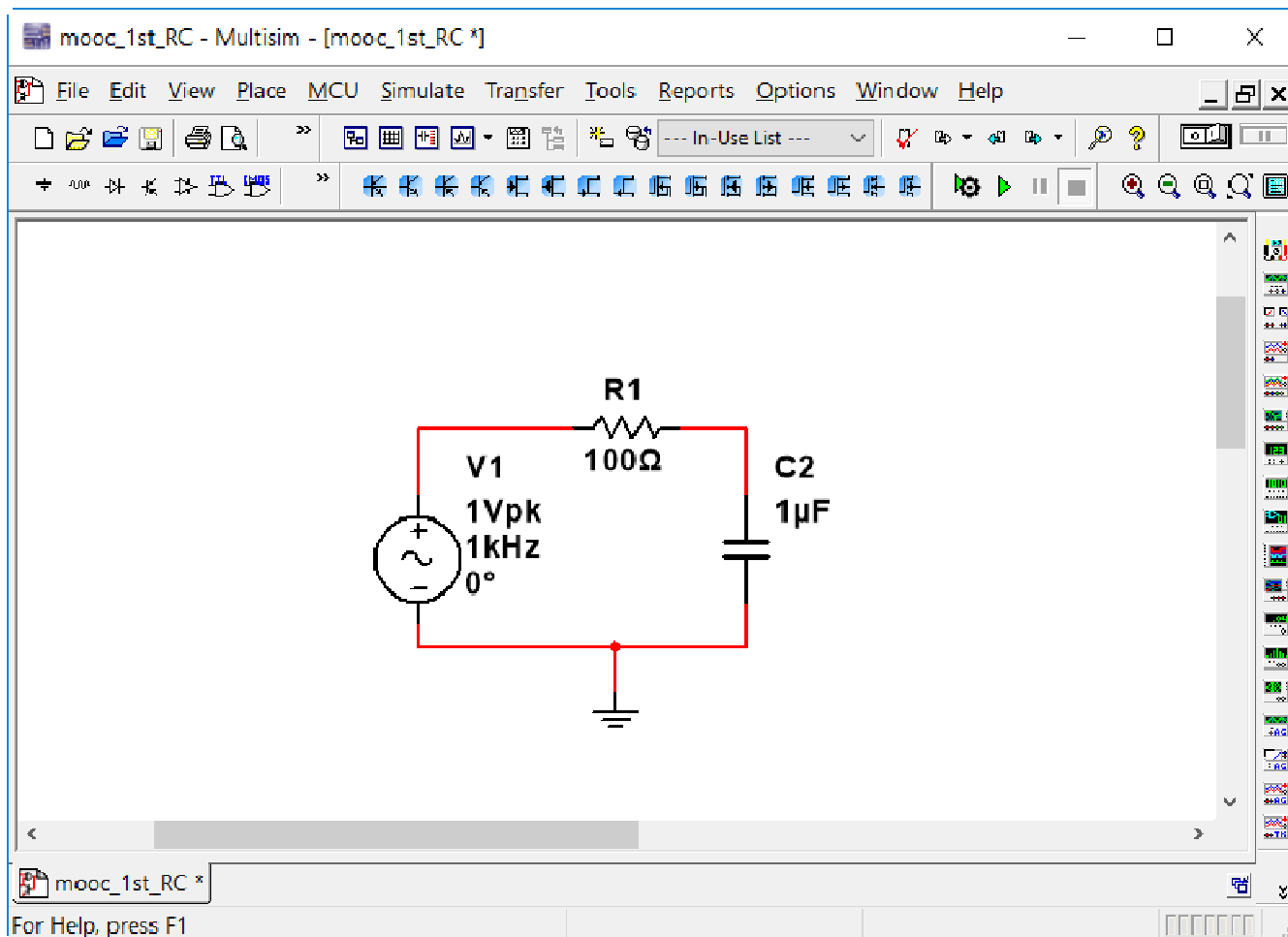


DC operating point...
AC analysis...
Single frequency AC analysis...
Transient analysis...
Fourier analysis...
Noise analysis...
Noise figure analysis...
Distortion analysis...
DC sweep...
Sensitivity...
Parameter sweep...
Temperature sweep...
Pole zero...
Transfer function...
Worst case...
Monte Carlo...
Trace width analysis...
Batched analysis...
User-defined analysis...
Stop analysis

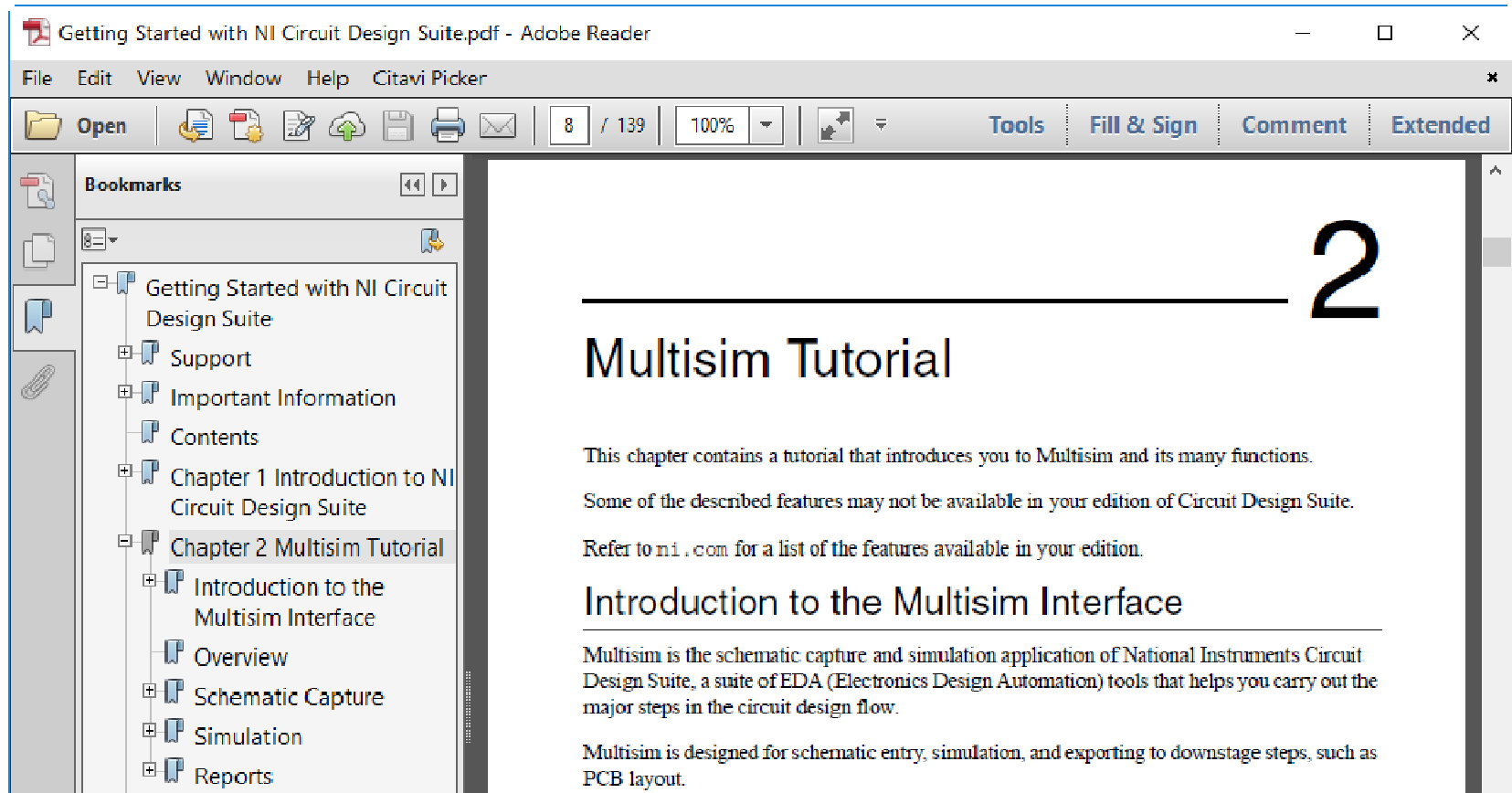
原理图绘制①：放置元器件



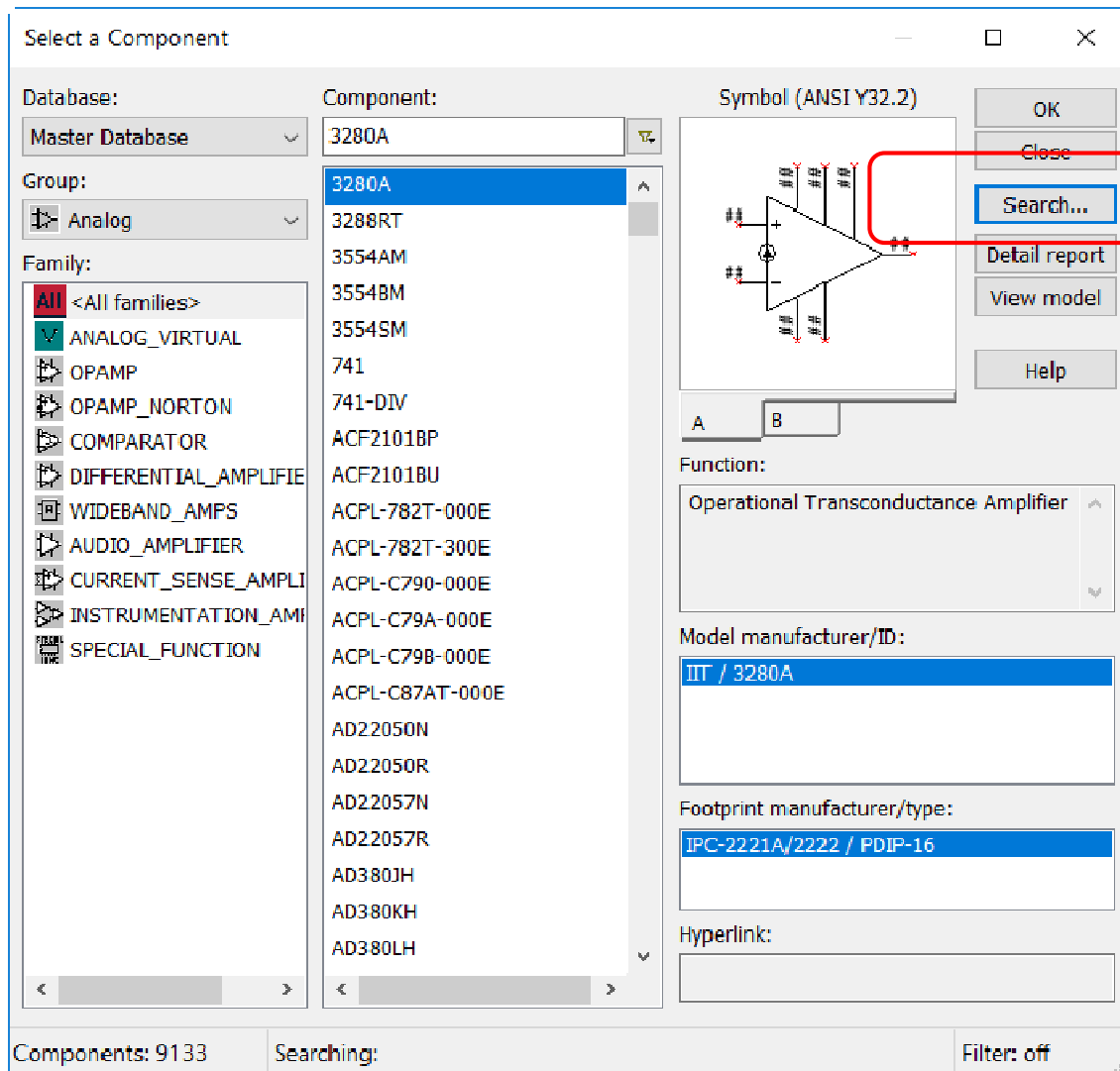
原理图绘制②：连线



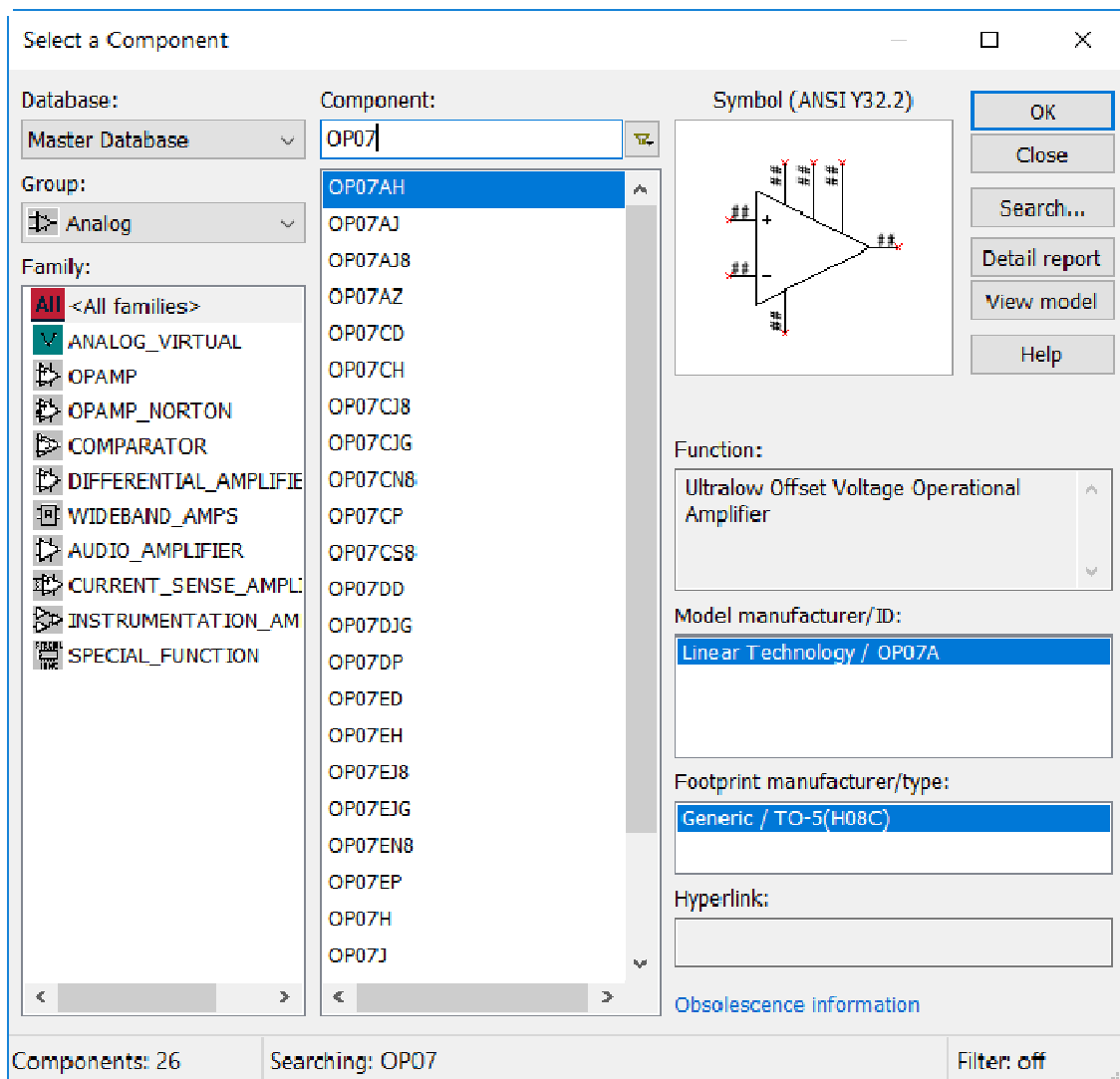
Multisim快速入门文档



搜索功能



搜索功能



①不同封装
②不同厂商

交流小信号仿真设置

AC Analysis

Frequency parameters | Output | Analysis options | Summary

Start frequency (FSTART): 1 Hz

Stop frequency (FSTOP): 10 GHz

Sweep type: Decade

Number of points per decade: 10

Vertical scale: Decibel

Reset to default

Simulate OK Cancel Help

瞬态仿真设置

Transient Analysis

Analysis parameters Output Analysis options Summary

Initial conditions: Determine automatically

Start time (TSTART): 0 s

End time (TSTOP): 0.001 s

☐ Maximum time step (TMAX): Determine automatically s

Setting a small TMAX value will improve accuracy, however the simulation time will increase.

☐ Initial time step (TSTEP): Determine automatically s

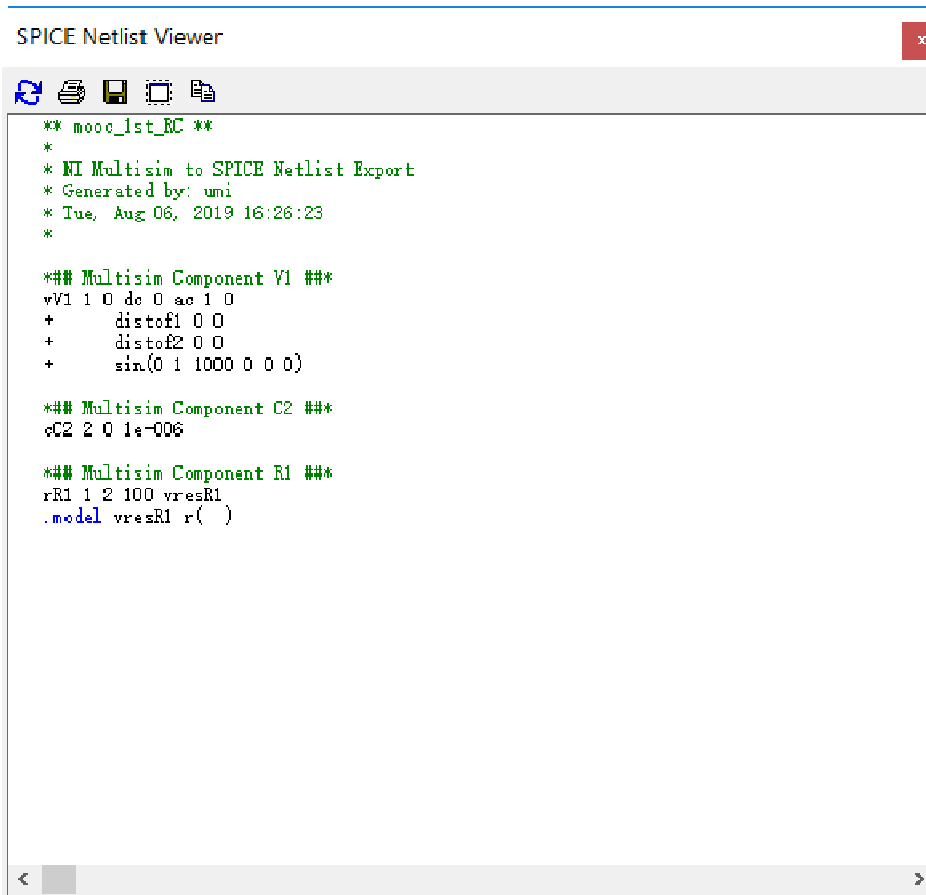
Reset to default

Simulate OK Cancel Help

网表

- Multisim

- HSPICE



```
SPICE Netlist Viewer
** moco_1st_RC **
*
* NI Multisim to SPICE Netlist Export
* Generated by: uni
* Tue, Aug 06, 2019 16:26:23
*
*** Multisim Component V1 ***
vV1 1 0 dc 0 ac 1 0
+   distof1 0 0
+   distof2 0 0
+   sin(0 1 1000 0 0 0)

*** Multisim Component C2 ***
cC2 2 0 1e-006

*** Multisim Component R1 ***
rR1 1 2 100 vresR1
.model vresR1 r()
```

.title RC 1st LPF

R1 1 2 100

C2 2 0 1u

V1 1 0 DC=0 AC=1

.AC dec 10 10 1g

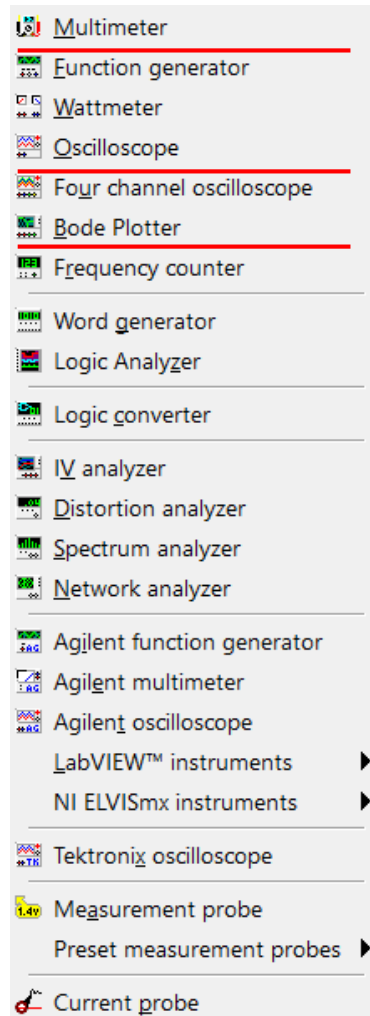
.probe vdb(2) vp(2)

.option post probe

.end

Multisim虚拟仪表

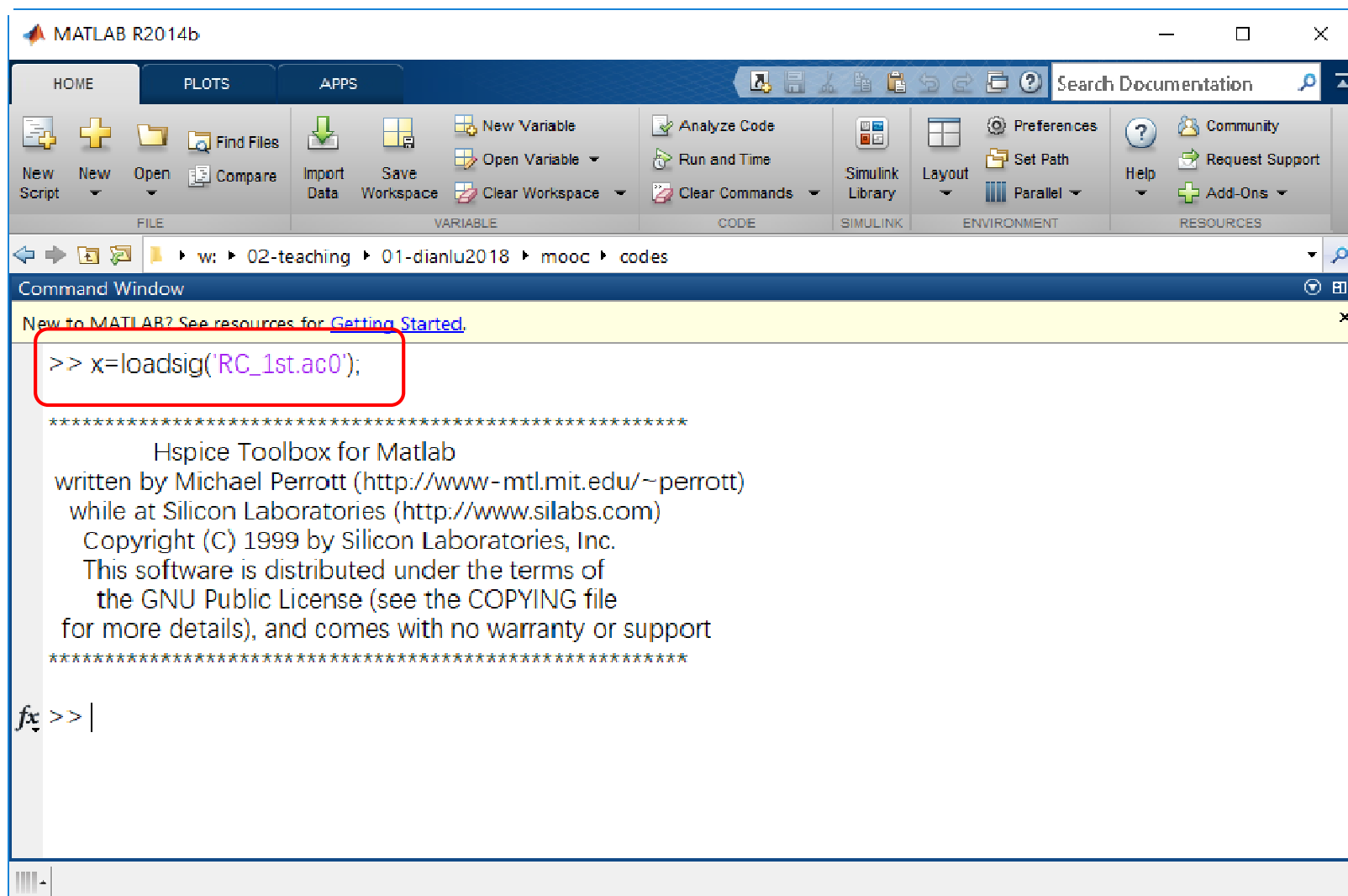
- 万用表
 - 直流电压/电流
- 波特图仪
 - 频域特性
- 示波器
 - 时域波形



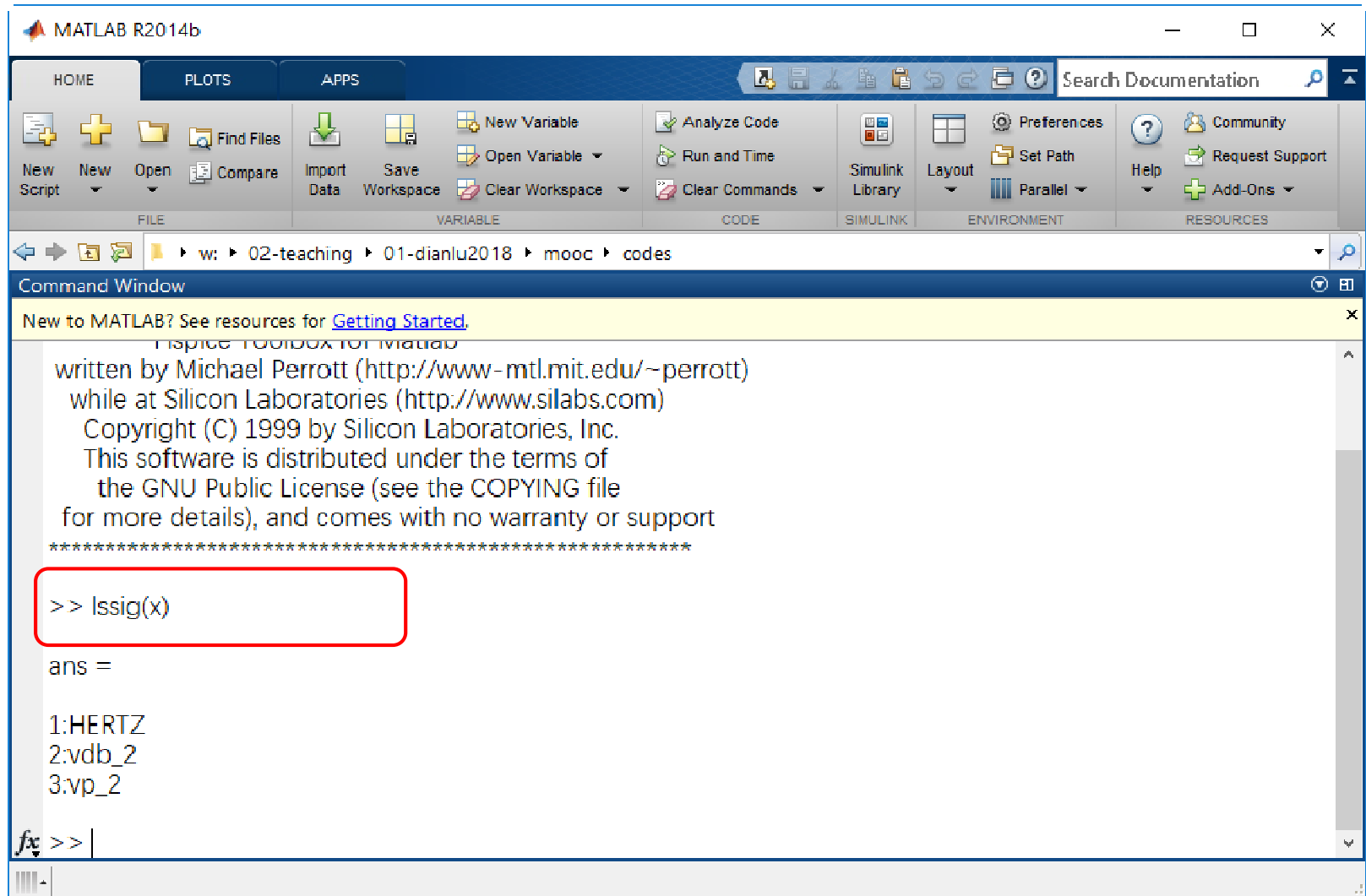
HSPICE输出文件

- .lis文件
 - 直流工作点信息等
- .tr#文件
 - 瞬态仿真结果
- .ac#文件
 - AC仿真结果

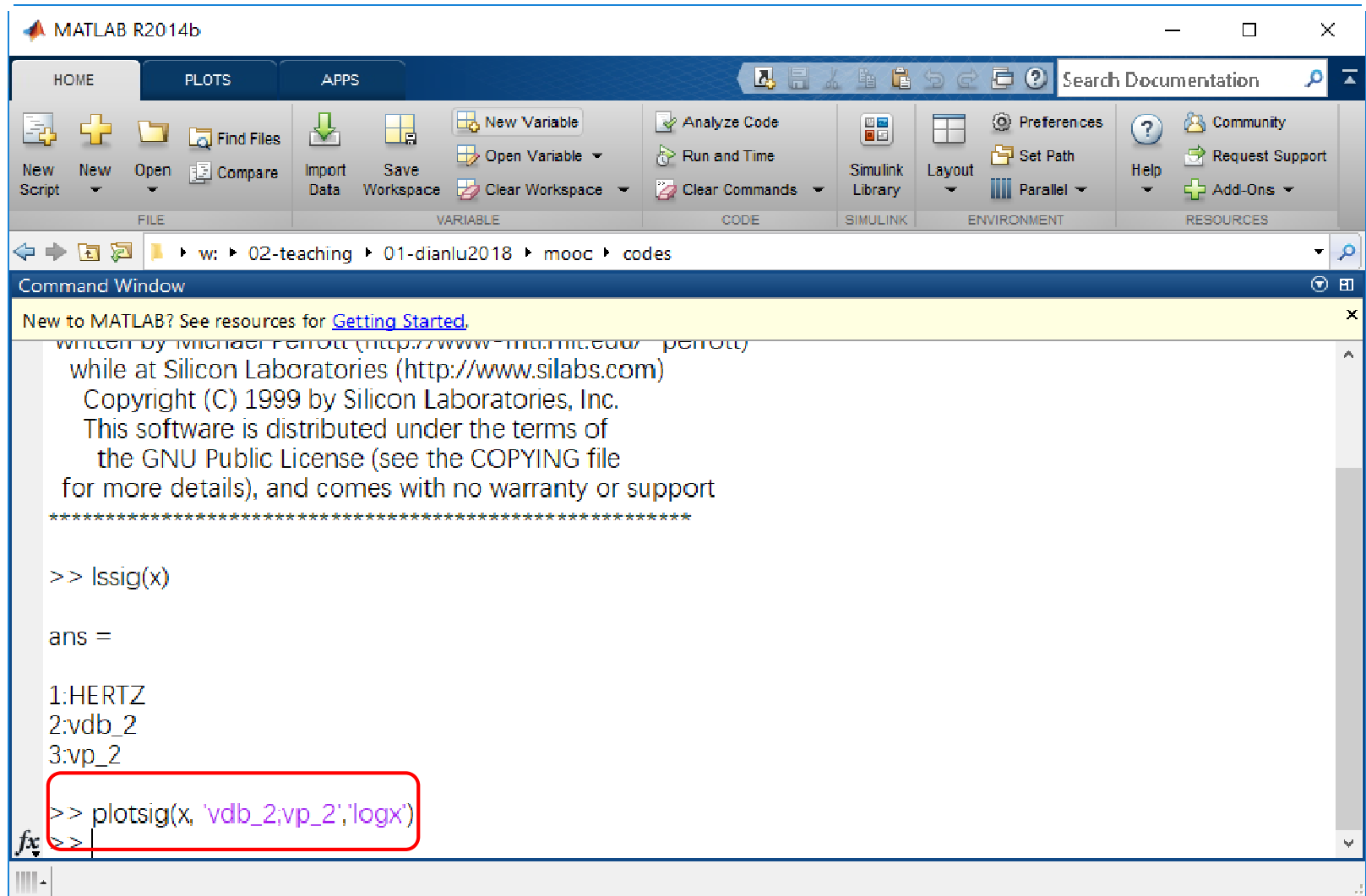
HSPICE Toolbox



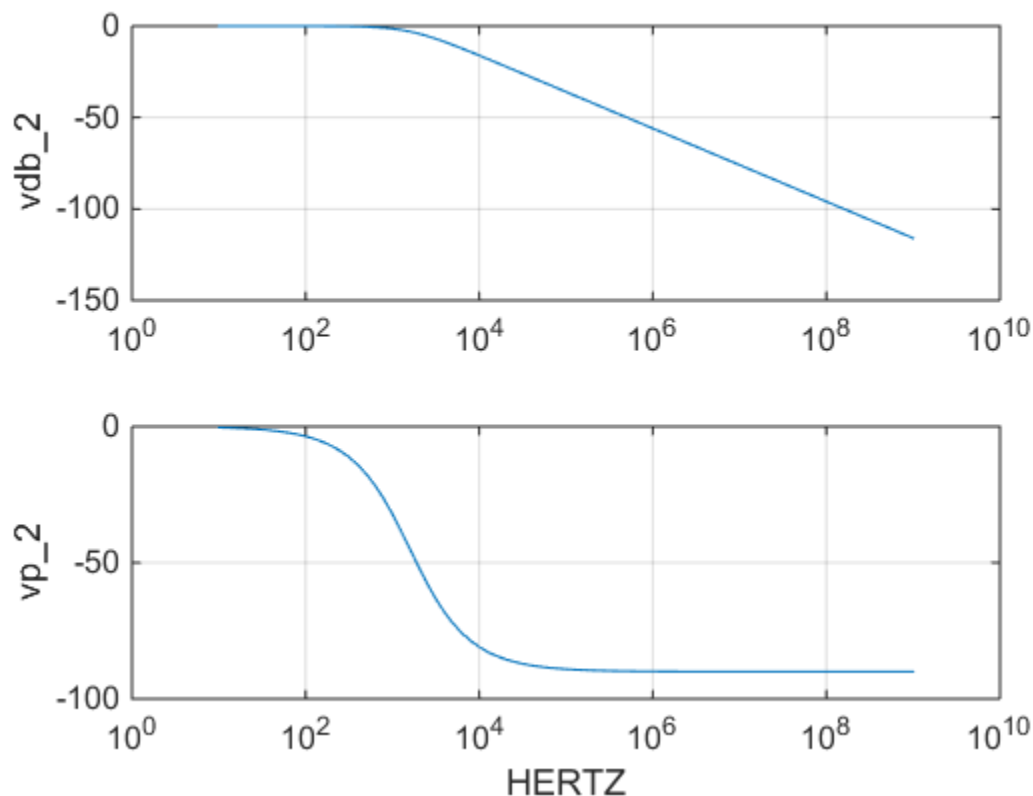
HSPICE Toolbox



HSPICE Toolbox



HSPICE Toolbox



重复课上的仿真，是快速入门的捷径

几点建议

- 了解软件的安装目录结构
- 不要忽略软件给出的任何提示
- 善于利用软件本身的帮助系统
- 学会过滤搜索引擎给出的劣质资源

软件只是工具

- Garbage in, garbage out
- 软件永远无法代替你思考，学会判断软件结果是否正确
- 软件终究只是工具，真正有意义的是你的设计
- 善用工具，优化设计