Project06: SPICE Simulation 3

繳交期限：2022年4月28日23:59

以下翻译结果来自-有道神经网络翻译(NMT)

题目描述：

在这个项目中，你将使用ngspice软件在你的系统上模拟SPICE电路网表。我们将对带通滤波器、带阻滤波器和RLC电路网络进行SPICE模拟。

请对
(1)第十讲幻灯片文件中绘制的两个电路:L10:P10，
(2)第十讲幻灯片文件中绘制的电路:L10:P11，
(3)第十讲幻灯片文件中绘制的RLC电路:L10:P15
进行瞬态SPICE模拟。由于没有指定电压源的类型和电阻、电容、电感的值，请在模拟时指定，并观察负载电阻的电压瞬态行为。

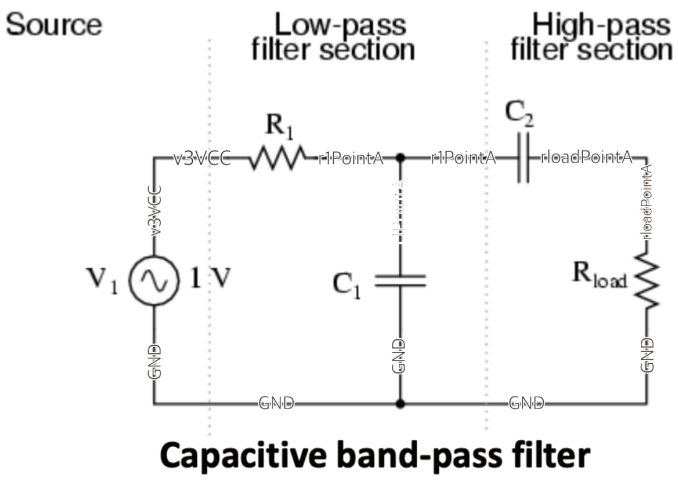
请按照以下规则提交学习报告:

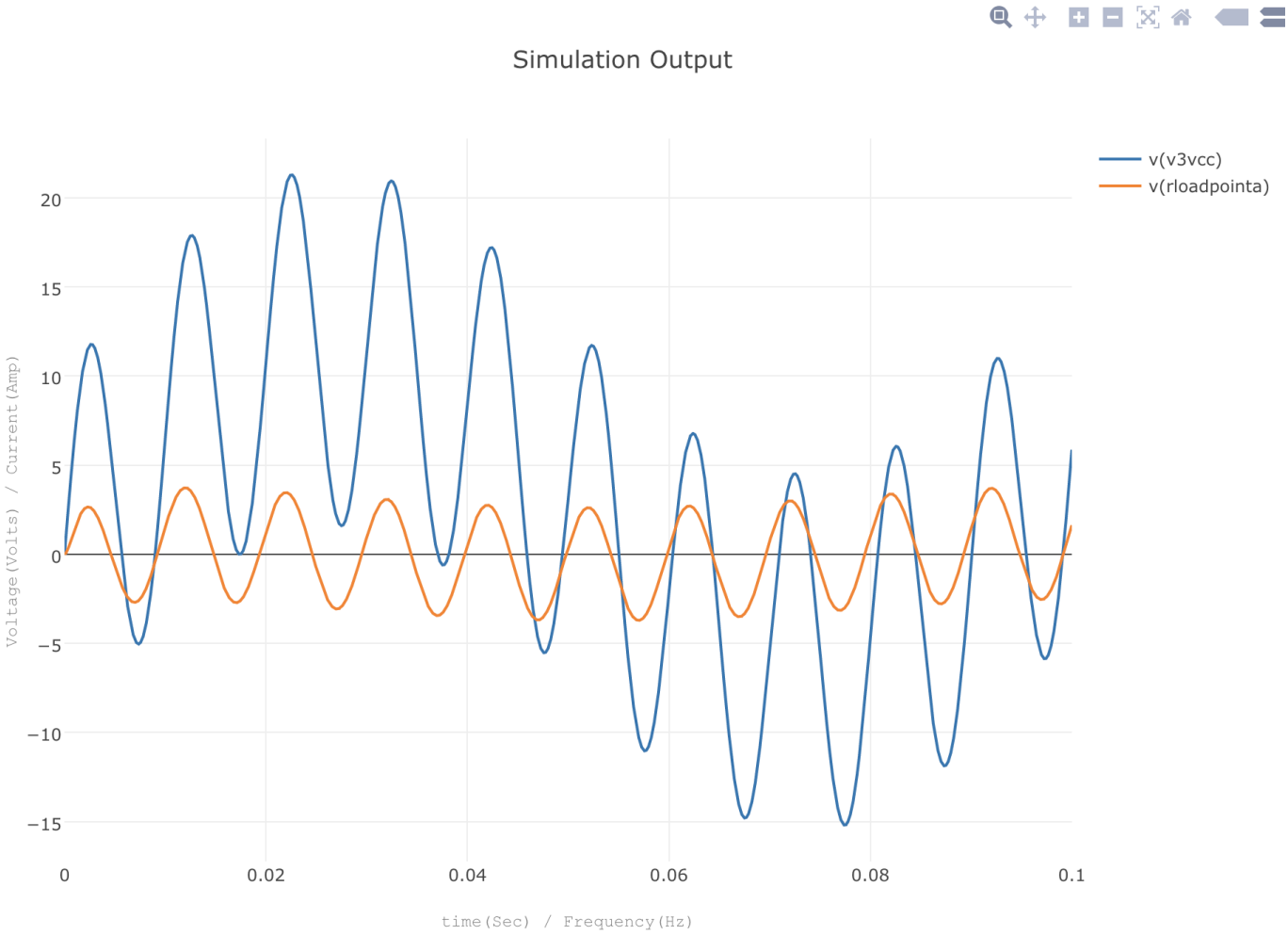
1- PDF格式的报告字体大小为12。
2- 文件名是你的学生证(例如，B12345678.pdf)。
3- 在模拟下发布SPICE电路的网络列表。
4- 绘制负载电阻的电压瞬态行为。
5- 分析改变R、L、C值时的瞬态行为。
6- 模拟结果的截图。

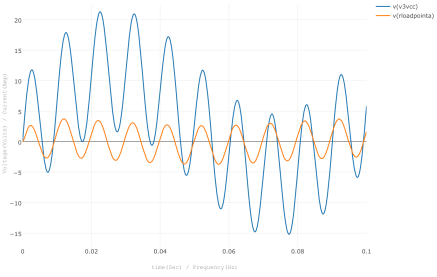
参考:

[1] Ngspice用户手册,
http://ngspice.sourceforge.net/docs/ngspice-manual.pdf

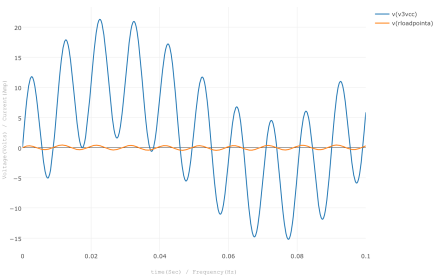
L10:P10（R C）



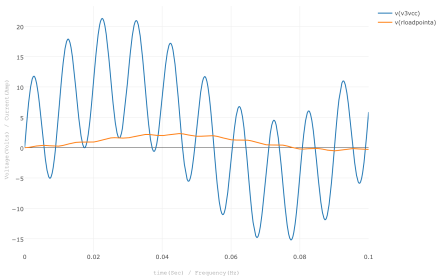




R減小（100 => 10）



C減小（100u => 10u）



C增大（100u => 10000u）

L10:P10（R L）

NGSPICE Online Simulator Home Author

Editor

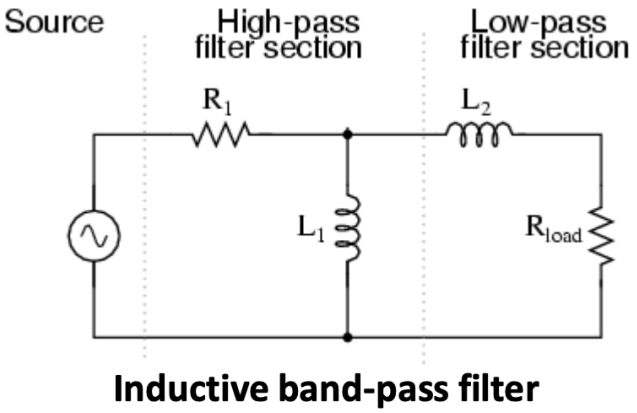
Please write your ngspice netlist in the below editor.

```
*L10:P10 RC Band-pass Filter

v1 v1VCC GND sin(0 10 1 0 0)
v2 v2VCC v1VCC sin(0 10 10 0 0)
v3 v3VCC v2VCC sin(0 10 100 0 0)

r1 v3VCC r1PointA 10
l1 r1PointA GND 10m
l2 r1PointA rloadPointA 10m
rload rloadPointA GND 10

.tran 1u 0.1 uic
*Please do not remove this line
.control
      run
.endc
.end
```

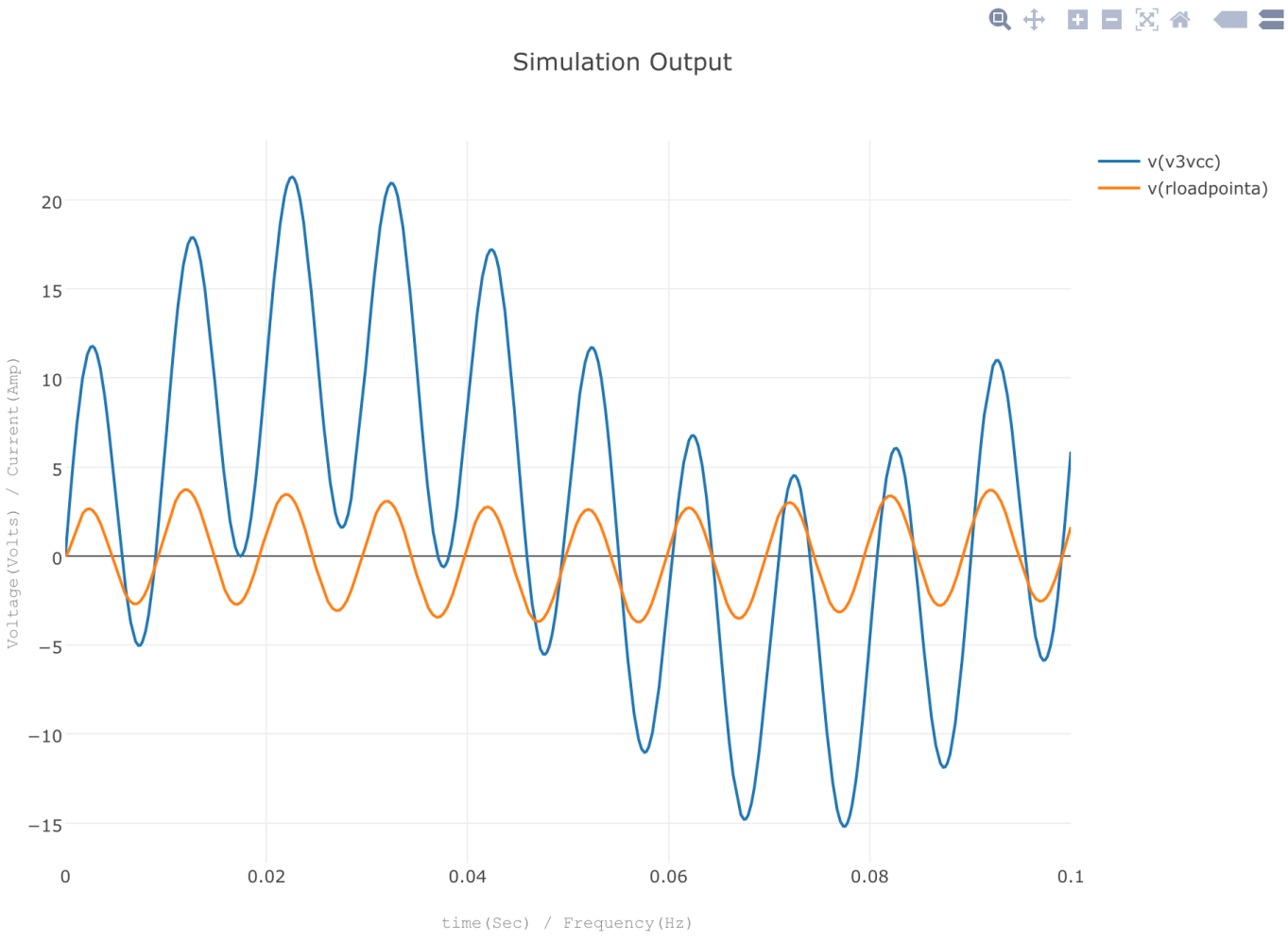
Plots:

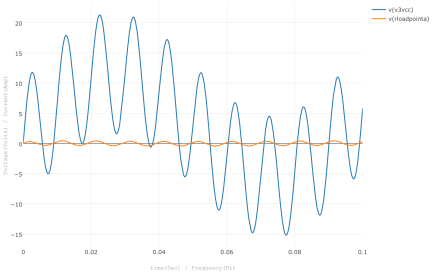
```
v(v3VCC) v(rloadPointA)
```

Save Netlist | Submit

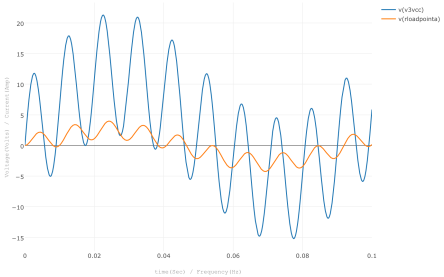


Inductive band-pass filter

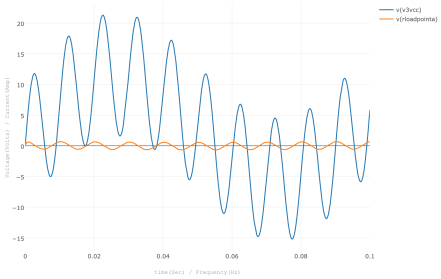




R增加(10 => 1000)



L增加（10m => 100m）



L减少(10m => 1m)

L10:P11

NGSPICE Online Simulator Home Author

Editor

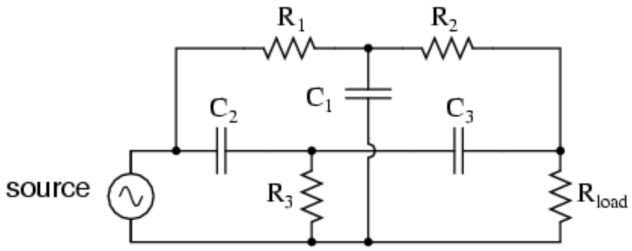
Please write your ngspice netlist in the below editor.

```
*L10:P11 RL Band-pass Filter

v1 v1VCC GND sin(0 10 1 0 0)
v2 v2VCC v1VCC sin(0 10 10 0 0)
v3 v3VCC v2VCC sin(0 10 100 0 0)

r1 v3VCC r1PointB 100
c1 r1PointB GND 10u
r2 r1PointB rloadPointA 100

c2 v3VCC c2PointB 10u
c3 c2PointB rloadPointA 10u

rload rloadPointA GND 1k
r3 c2PointB GND 100

.tran 1u 0.1 uic
*Please do not remove this line
.control
        run
.endc
.end
```

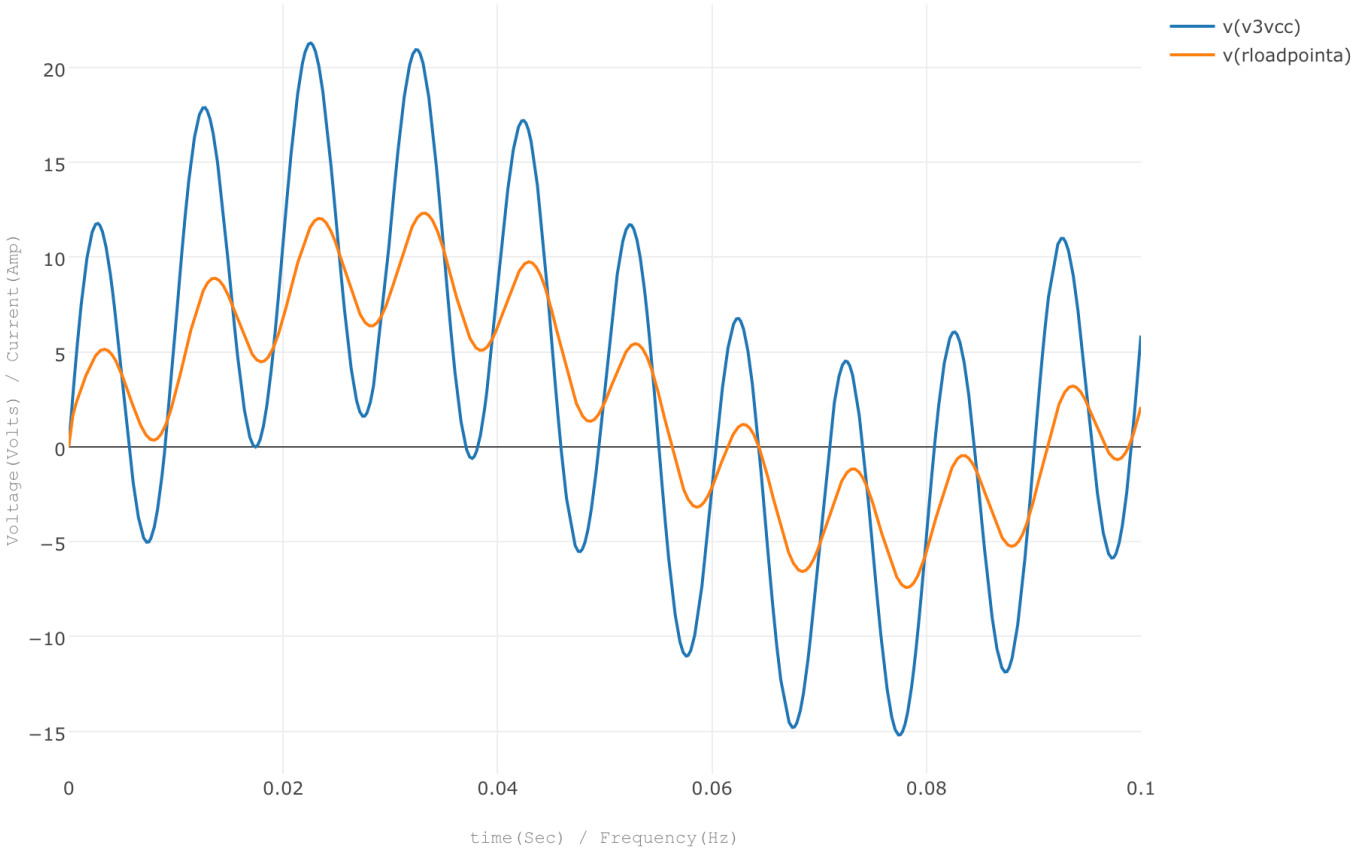
Plots:

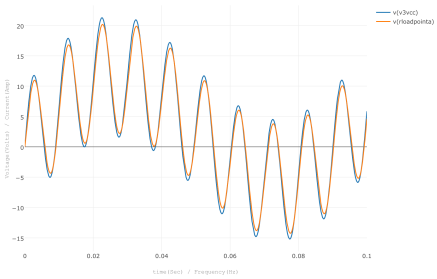
```
v(v3VCC) v(rloadPointA)
```

Save Netlist | Submit

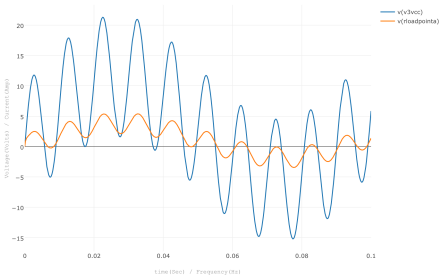


“Twin-T” band-stop filter

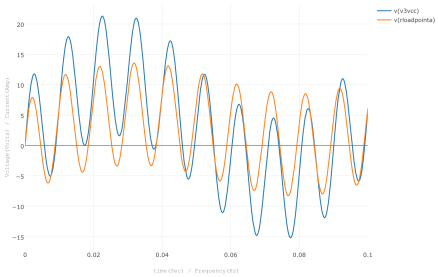




R减少（100 => 10）



R增加(100 => 1k)



C增加(10u => 100u)

L10:P15

NGSPICE Online Simulator Home Author

Editor

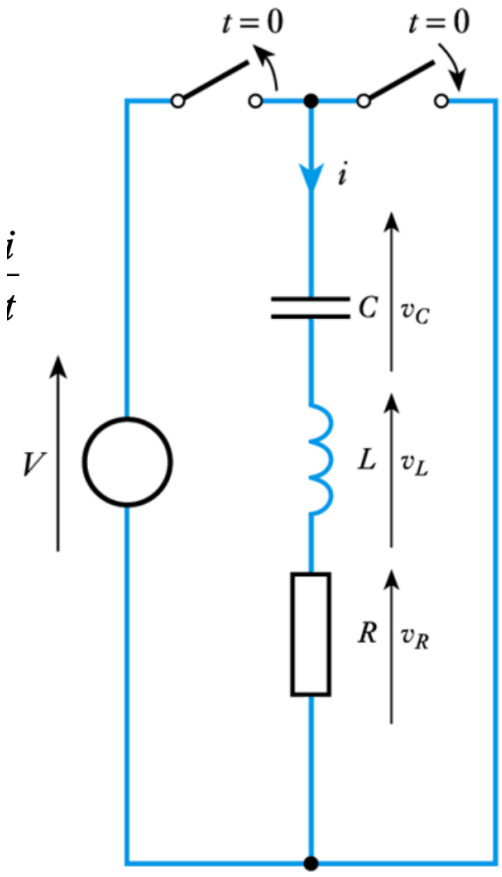
Please write your ngspice netlist in the below editor.

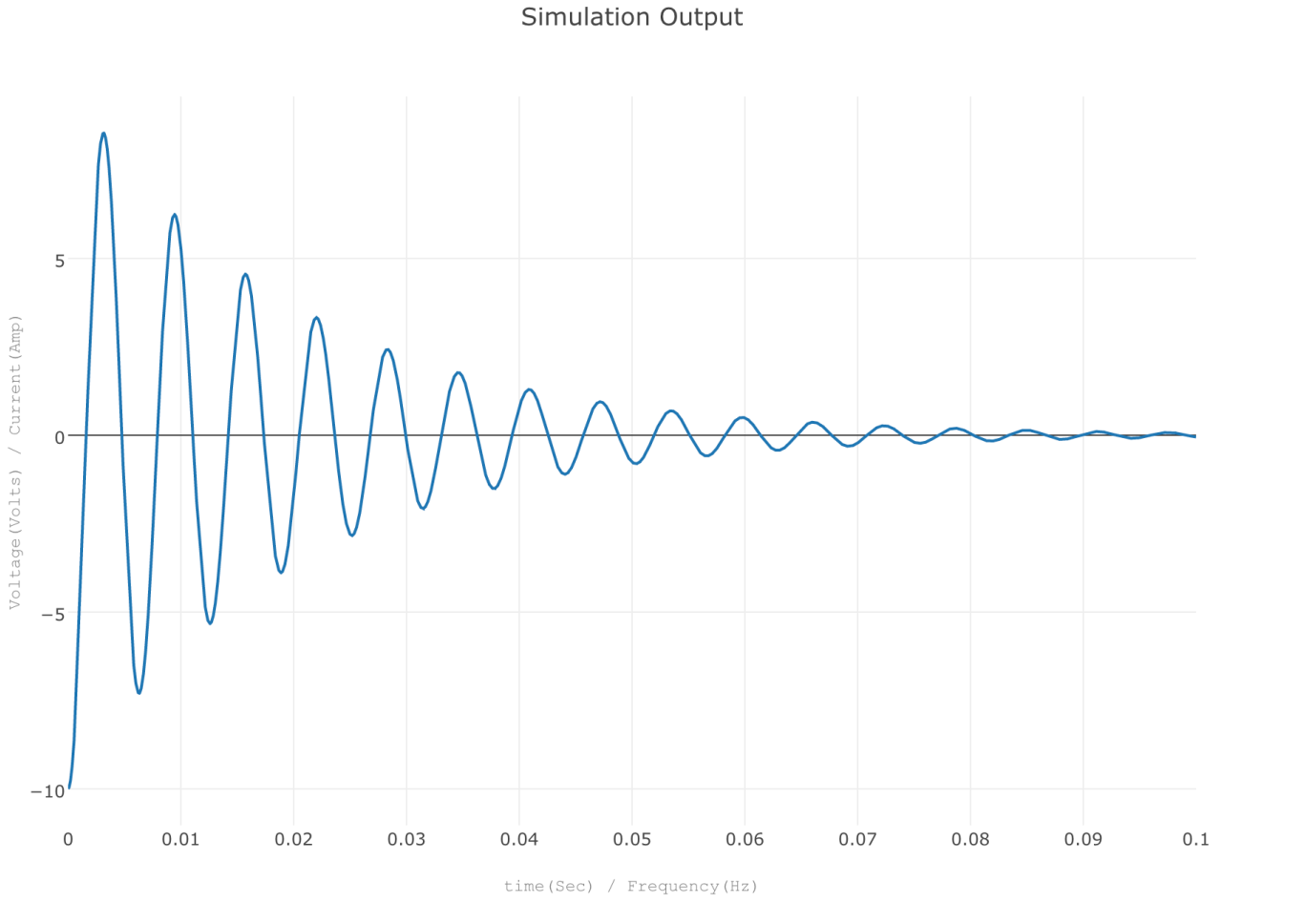
```
*L10:P15 RLC Circuit Under Damped

r1 r1PointA GND 100

l1 l1PointA r1PointA 1

c1 GND l1PointA 1u ic=10

.tran 1u 0.1 uic
*Please do not remove this line
.control
      run
.endc
.end
```

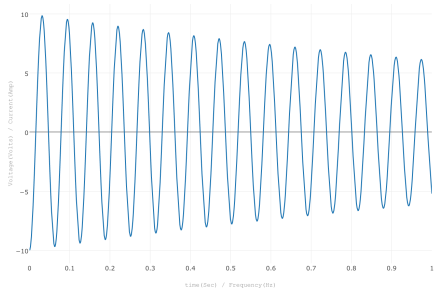
Plots:

```
v(l1PointA)
```

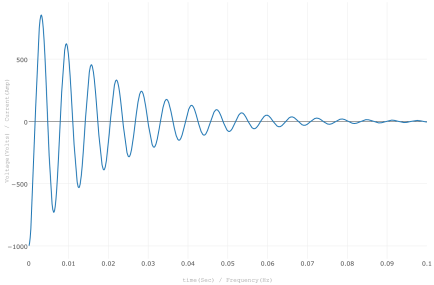
Save Netlist | Submit



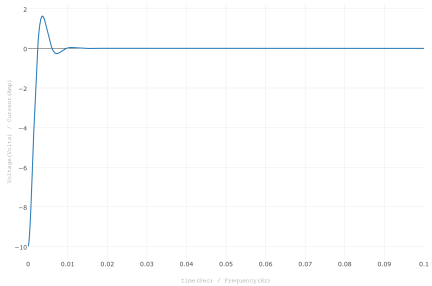




L增加(1 => 100，並增加tran的時間)



C增加(10 => 1000)



R增加(100 => 1k)

參考的平台資料

https://www.ceibs.edu/facultyCV/cvincent/2013_Simulation.pdf

https://ntust-spice.tk/