Circuit Simulation Modeling

TABLE OF CONTENTS

Spice

❑ **SPICE**

(Simulation Program with Integrated Circuit Emphasis) is a widely used computer software tool for simulating and analyzing electronic circuits. It is a general-purpose circuit simulation program that allows engineers and designers to model and simulate the behavior of analog and digital circuits.

❑ **Our Tool provide the following capabilities:**

- Linear dc analysis
- AC Analysis
- Transient Analysis

Netlist Structure

- Passive Elements

Instance Name	Component Type	From Node	To Node	Value
R	resistor	-	-	(T | G | M | K | nothing | m)
C	capacitor	-	-	(f | p | n | u | m | nothing)
L	inductor	-	-	(f | p | n | u | m | nothing)

- Independent Sources

Instance Name	Component type	From Node	To Node	Type	Value
V	vsource	-	-	dc | ac | step	(m | nothing)
I	Isource	-	-	dc | ac	(m | nothing)
V	Vcos	-	-	cos | sin	(m | nothing)

Netlist Structure

- Dependent Sources:

Instance Name	Component type	K	k'	J	J'	Type	Value
I	vccs	-	-	-	-	dc | ac	(m | nothing)
V	vcvs	-	-	-	-	dc | ac	(m | nothing)
I	cccs	-	-	-	-	dc | ac	(m | nothing)
V	ccvs	-	-	-	-	dc | ac	(m | nothing)

- Analysis Types:

Analysis Name	type
dcOp	dc

Analysis Name	type	Start	stop	dec
AC Analysis	ac	-	-	-

Analysis Name	type	Time step	Stop time
Transient Analysis	tran	-	-

- Operational Amplifier (ideal):

Analysis Name	Component type	Negative terminal	Negative terminal	Output terminal
Operational_Amplifier	opamp	-	-	-

- Multiple Plot:

plot	V0	V1	V2	...	V#	I_V#

Netlist Structure

- Diode

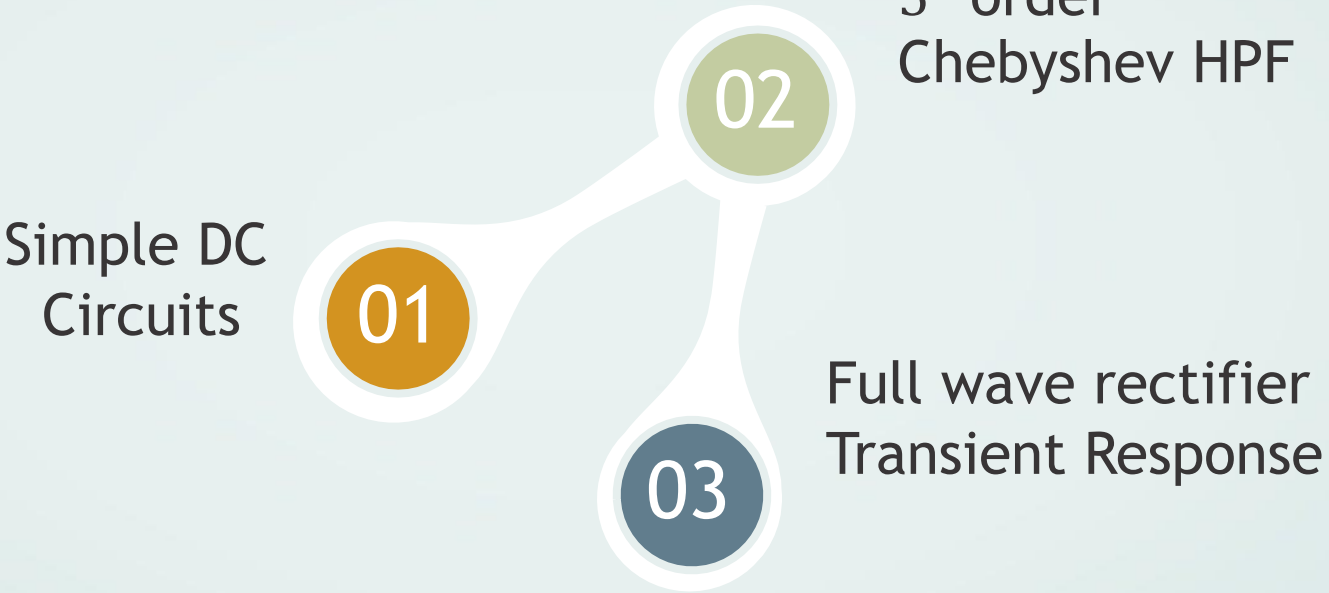
Instance Name	Component type	From Node	To Node
D	Diode	-	-

- Diode model
  - Static model $I_d = I_s * (e^{\frac{v_b}{V_T}} - 1)$
  - Dynamic model (large signal)

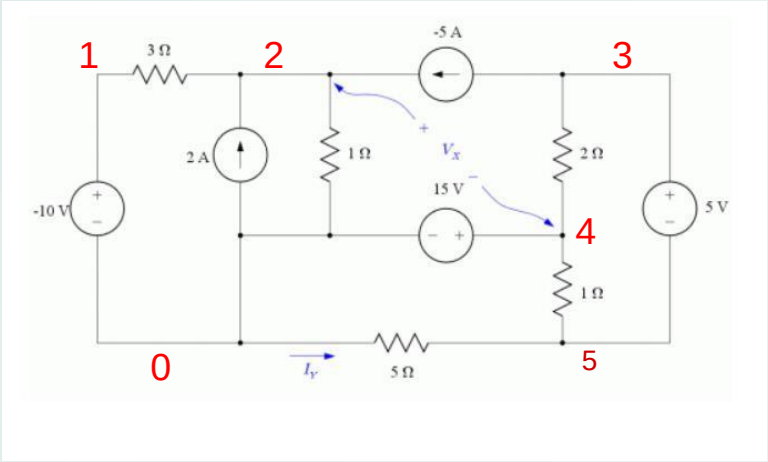
$$C_D = C_d + C_J = \begin{cases} \tau_d I_d + \dfrac{C_{j0}}{\left(1 - \dfrac{v_d}{V_j}\right)^m}, & v_d > FC * V_j \\ \tau_d I_d + \dfrac{C_{j0}}{F_2}\left(F_3 + \dfrac{m * v_d}{V_j}\right), & v_d > FC * V_j \end{cases}$$

$$FC = 0.5$$
$$F_2 = (1 - FC)^{1+m}$$
$$F_3 = 1 - FC(1 + m)$$

Examples


5^{th}order
Chebyshev HPF
02
Simple DC
Circuits
01
Full wave rectifier
Transient Response
03

Simple DC Circuit #1



Circuit Schematic

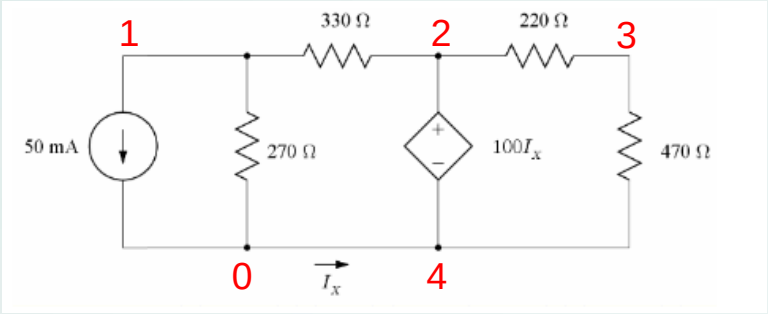
```
// Simple DC Circuit
R1 resistor 1 2 3
R2 resistor 2 0 1
R3 resistor 3 4 2
R4 resistor 4 5 1
R5 resistor 5 0 5
V1 vsource 0 1 dc 10
V2 vsource 4 0 dc 15
V3 vsource 3 5 dc 5
I1 isource 0 2 dc 2
I2 isource 2 3 dc 5
dcOp dc
```

Netlist

```
PS D:\Python Projects\Netlist> python.exe .\main.py
V1 = [-10.]
V2 = [-4.75]
V3 = [19.70588235]
V4 = [15.]
V5 = [14.70588235]
I_V1 = [-1.75]
I_V2 = [2.05882353]
I_V3 = [2.64705882]
```

Simulation Result

Simple DC Circuit #2



Circuit Schematic

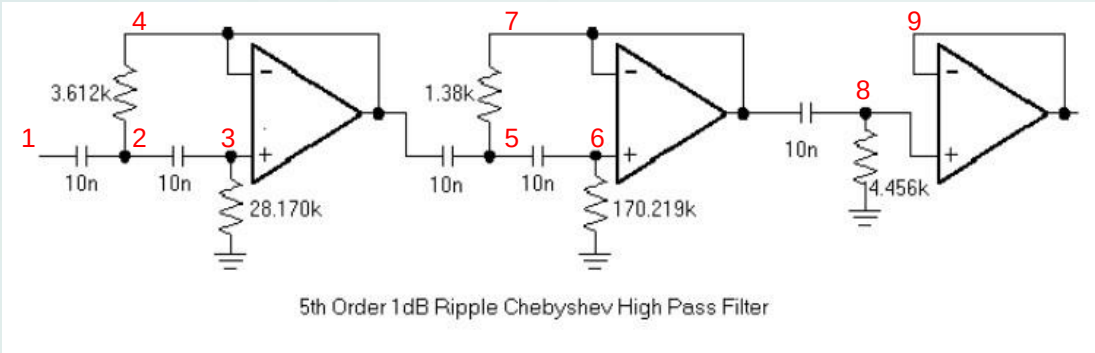
```
// Dependent Sources
R1 resistor 1 0 270
R2 resistor 1 2 330
R3 resistor 2 3 220
R4 resistor 3 4 470
I1 isource 1 0 dc 50m
Vdep ccvs 2 4 0 4 dc 100
dcOp dc
```

Netlist

```
PS D:\Python Projects\Netlist> python.exe .\main.py
V1 = [-6.21]
V2 = [2.7]
V3 = [1.83913043]
V4 = [-0.]
I_Vdep_1 = [0.027]
I_Vdep_2 = [-0.03091304]
```

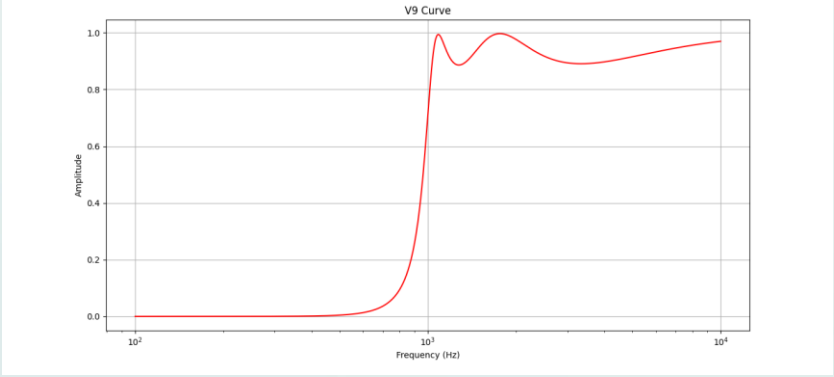
Simulation Result

AC Analysis



5th Order 1dB Ripple Chebyshev High Pass Filter

Circuit Schematic

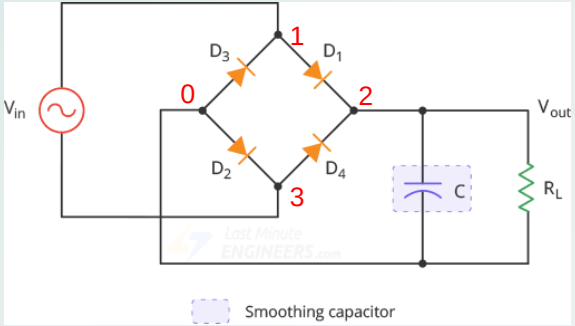


Simulation Result

```
// 5th order 1dB Ripple Chebycheve HPF
C1 capacitor 1 2 10n
C2 capacitor 2 3 10n
R1 resistor 2 4 3612
R2 resistor 3 0 28170
Opamp1 opamp 3 4 4

C3 capacitor 4 5 10n
C4 capacitor 5 6 10n
R3 resistor 5 7 1380
R4 resistor 6 0 170219
Opamp2 opamp 6 7 7

C5 capacitor 7 8 10n
R5 resistor 8 0 4456
Opamp1 opamp 8 9 9

V vsource 1 0 ac 1
ac ac 100 10K 100
plot V9
```

Netlist

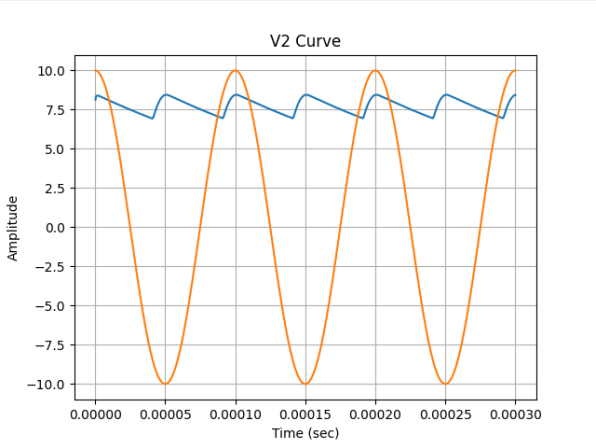
Non-linear Transient Analysis



Circuit Schematic

```
//full wave rectifier
D1 diode 1 2
D2 diode 0 3
D3 diode 3 2
D4 diode 0 1
V vcos 1 3 cos 10 10K
R1 resistor 2 0 10
C1 capacitor 2 0 20u

tran tran 300n 300u
plot V2
```

Netlist



Simulation Result