Created in COMSOL Multiphysics 6.0

Operational Amplifier with Capacitive Load

Introduction

An Operational Amplifier (op-amp) is a differential voltage amplifier with a wide range of applications in analog electronics. This example shows how to model an operational amplifier connected to a feedback loop and a capacitive load and calculate the transient step response of the entire system. The basic op-amp model used here is implemented as an equivalent linear subcircuit in the Electrical Circuit interface in COMSOL Multiphysics. The latter is partially based on the SPICE format originally developed at Berkeley University (Ref. 1).

Model Definition

The op-amp subcircuit is described by the following lines in a SPICE netlist:

```
.SUBCKT OPAMP p n out gnd
RIN p n 100000.0
EGAIN 1 gnd p n 100000.0
RP 1 2 1591549.4309189531
CP 2 gnd 1.0E-9
EBUFFER 3 gnd 2 gnd 1.0
ROUT 3 out 100.0
.ENDS OPAMP
```

The different stages are:

- An input with high impedance: `RIN`.
- A high gain differential amplifier: `EGAIN`.
- A single-pole low-pass filter: `RP` and `CP`.
- An output buffer with unity gain: `EBUFFER` and `ROUT`.

The op-amp subcrcuit instance is then inserted into the main circuit:

```
VIN 1 0 DC 0.5
XOPAMP 1 2 3 0 OPAMP
R1 2 0 470.0
R2 2 3 4700.0
CLOAD 3 0 1.0E-8
```

Here the voltage source is indicated as being constant at 0.5V whereas in the model a voltage step of 0.5V is applied at t=0. A resistive feedback loop is provided through the resistors `R1` and `R2` and the amplifier output is terminated to ground via a capacitive load `CLOAD`.

Results and Discussion

The model is simulated for 10µs with data output every 0.05µs. The internal dynamics of the op-amp interacts with the feedback network causing ringing in the output signal.

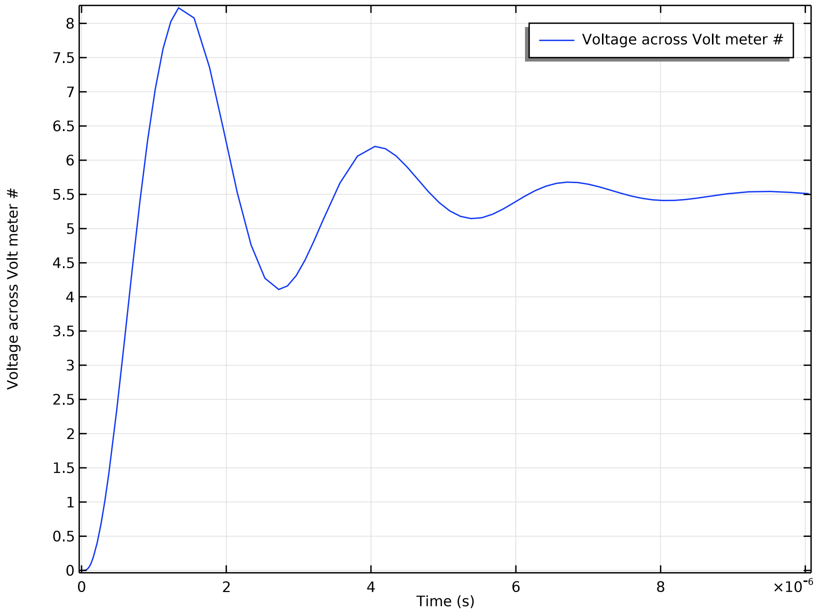


Figure 1: The output voltage of the op-amp as a function of time.

Reference

1. The SPICE home page, http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE.

Application Library path: `ACDC_Module/Tutorials/opamp_capacitive_load`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **0D**.
2 In the **Select Physics** tree, select **AC/DC>Electrical Circuit (cir)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Time Dependent**.
6 Click **Done**.

GLOBAL DEFINITIONS

Start by defining the parameters to be used in the model.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
OPAMP_RIN	100[kohm]	1E5 Ω	Op-amp input resistance
OPAMP_GAIN	1e5	1E5	Op-amp gain
CLOAD	10[nF]	1E-8 F	Capacitive load
R1	470[ohm]	470 Ω	Feedback resistance 1
R2	4700[ohm]	4700 Ω	Feedback resistance 2
OPAMP_P	100[Hz]	100 Hz	Op-amp pole frequency
OPAMP_ROUT	100[ohm]	100 Ω	Op-amp output resistance

DEFINITIONS

Add a step function and a variable defining the voltage step used to drive the model.

Step 1 (step1)

In the **Home** toolbar, click f(x) **Functions** and choose **Local>Step**.

Variables 1

1 In the **Home** toolbar, click a= **Variables** and choose **Local Variables**.
2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
VIN	.5[V]*step1((t-.05[us])/1[us])	V	Input voltage

ELECTRICAL CIRCUIT (CIR)

Now, define the circuit. Start by defining the subcircuit for the op-amp.

OPAMP

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electrical Circuit (cir)** and choose **Subcircuit Definition**.

2 In the **Settings** window for **Subcircuit Definition**, type OPAMP in the **Label** text field.

3 Locate the **Node Connections** section. Click + **Add**.

4 Click + **Add**.

5 In the table, enter the following settings:

Node names
p
n
out
gnd

Resistor RIN

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, type Resistor RIN in the **Label** text field.

3 In the **Name** text field, type RIN.

4 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	p
n	n

5 Locate the **Device Parameters** section. In the R text field, type OPAMP_RIN.

Voltage-Controlled Voltage Source EGAIN

1 In the **Electrical Circuit** toolbar, click **Voltage-Controlled Voltage Source**.

2 In the **Settings** window for **Voltage-Controlled Voltage Source**, type Voltage-Controlled Voltage Source EGAIN in the **Label** text field.

3 In the **Name** text field, type `EGAIN`.

4 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	1
n	gnd
measure (+)	p
measure (-)	n

5 Locate the **Device Parameters** section. In the **Gain** text field, type `OPAMP_GAIN`.

Resistor RP

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, type `Resistor RP` in the **Label** text field.

3 In the **Name** text field, type `RP`.

4 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	1
n	2

5 Locate the **Device Parameters** section. In the R text field, type `1/(2*pi*OPAMP_P*1[nF])`.

Capacitor CP

1 In the **Electrical Circuit** toolbar, click **Capacitor**.

2 In the **Settings** window for **Capacitor**, type `Capacitor CP` in the **Label** text field.

3 In the **Name** text field, type `CP`.

4 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	2
n	gnd

5 Locate the **Device Parameters** section. In the C text field, type `1[nF]`.

Voltage-Controlled Voltage Source EBUFFER

1 In the **Electrical Circuit** toolbar, click **Voltage-Controlled Voltage Source**.

2 In the **Settings** window for **Voltage-Controlled Voltage Source**, type `Voltage-Controlled Voltage Source EBUFFER` in the **Label** text field.

3 In the **Name** text field, type `EBUFFER`.

4 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	3
n	gnd
measure (+)	2
measure (-)	gnd

Resistor ROUT

1 In the **Electrical Circuit** toolbar, click **Resistor**.

2 In the **Settings** window for **Resistor**, type `Resistor ROUT` in the **Label** text field.

3 In the **Name** text field, type `ROUT`.

4 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	3
n	out

5 Locate the **Device Parameters** section. In the R text field, type `OPAMP_ROUT`.

Voltage Source VIN

Proceed to add the main circuit, start by adding the voltage source.

1 In the **Electrical Circuit** toolbar, click **Voltage Source**.

2 In the **Settings** window for **Voltage Source**, type `Voltage Source VIN` in the **Label** text field.

3 In the **Name** text field, type `VIN`.

4 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	1
n	0

5 Locate the **Device Parameters** section. In the v_{src} text field, type `VIN`.

Subcircuit Instance XOPAMP

Add the subcircuit instance for the opamp.

1 In the **Electrical Circuit** toolbar, click **Subcircuit Instance**.
2 In the **Settings** window for **Subcircuit Instance**, type `Subcircuit Instance XOPAMP` in the **Label** text field.
3 In the **Name** text field, type `XOPAMP`.
4 Locate the **Node Connections** section. From the **Name of subcircuit link** list, choose **OPAMP (sub1)**.
5 In the table, enter the following settings:

Local node names	Node names
p	1
n	2
out	3
gnd	0

Resistor 1 (R1)

1 In the **Electrical Circuit** toolbar, click **Resistor**.
2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.
3 In the table, enter the following settings:

Label	Node names
p	2
n	0

4 Locate the **Device Parameters** section. In the R text field, type `R1`.

Resistor 2 (R2)

1 In the **Electrical Circuit** toolbar, click **Resistor**.
2 In the **Settings** window for **Resistor**, locate the **Node Connections** section.
3 In the table, enter the following settings:

Label	Node names
p	2
n	3

4 Locate the **Device Parameters** section. In the R text field, type `R2`.

Capacitor CLOAD

1 In the **Electrical Circuit** toolbar, click **Capacitor**.

2 In the **Settings** window for **Capacitor**, type `Capacitor CLOAD` in the **Label** text field.

3 In the **Name** text field, type `CLOAD`.

4 Locate the **Node Connections** section. In the table, enter the following settings:

Label	Node names
p	3
n	0

5 Locate the **Device Parameters** section. In the *C* text field, type `CLOAD`.

Volt Meter 1 (vm1)

In order to see the output voltage, a voltmeter is added.

1 In the **Electrical Circuit** toolbar, click **Volt Meter**.

2 In the **Settings** window for **Volt Meter**, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	3
n	0

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 Click **Range**.

4 In the **Range** dialog box, type `0.05[us]` in the **Step** text field.

5 In the **Stop** text field, type `10[us]`.

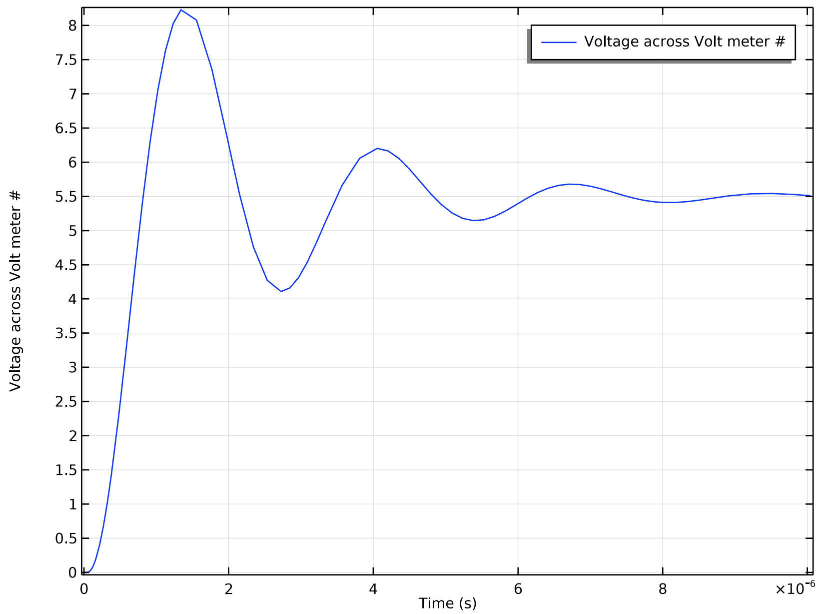
6 Click **Replace**.

7 In the **Home** toolbar, click **Compute**.

RESULTS

Probe Plot Group 1

The output voltage appears as a probe plot.



ROOT

Finally add a model thumbnail image.

1. In the **Model Builder** window, click the root node.
2. In the root node's **Settings** window, locate the **Presentation** section.
3. Find the **Thumbnail** subsection. Click **Set from Graphics Window**.