



Model created in COMSOL Multiphysics 6.4

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 in order to 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 subcircuit 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 0.5 V whereas in the model a voltage step of 0.5 V 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 $10\mu\text{s}$ with data output every $0.05\mu\text{s}$. The internal dynamics of the op-amp interacts with the feedback network causing ringing in the output signal.

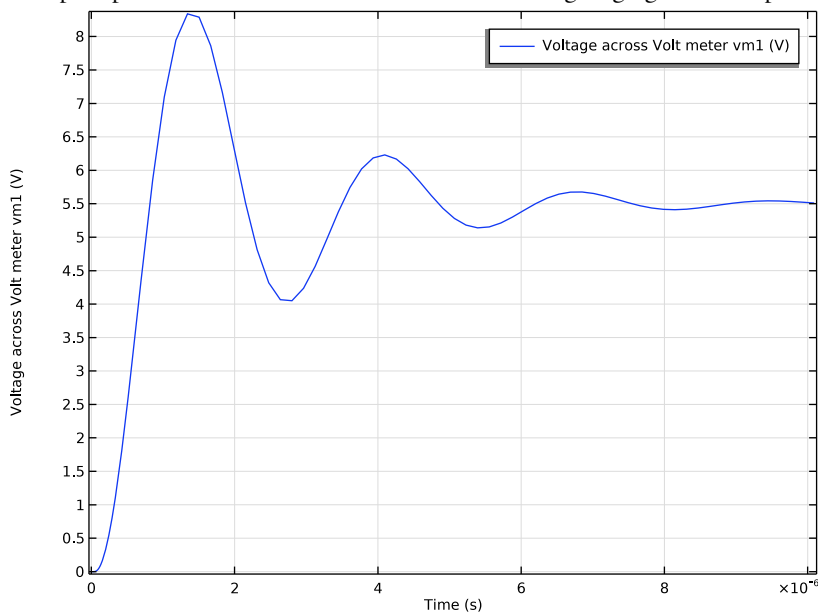


Figure 1: The output voltage of the op-amp as a function of time.

Reference


1. A. Vladimirescu et al., *SPICE Version 2G User's Guide*, University of California, Berkeley, 1981.

Application Library path: ACDC_Module/Electromagnetics_and_Circuits/
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 **Definitions** toolbar, click  **More Functions** and choose **Step**.

Variables 1

- 1 In the **Home** toolbar, click  **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	$.5[V]*\text{step1}((t-.05[\text{us}])/1[\text{us}])$	V	Input voltage

ELECTRICAL CIRCUIT (CIR)


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

OPAMP

- 1 In the **Electrical Circuit** toolbar, click  **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

5 Locate the **Voltage Measurement** section. In the table, enter the following settings:

Positive node	Negative node
p	n

6 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 $1 / (2 * \pi * \text{OPAMP_P} * 1[\text{nF}])$.

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 $1[\text{nF}]$.

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

- 5 Locate the **Voltage Measurement** section. In the table, enter the following settings:

Positive node	Negative node
2	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 op-amp.

- 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 **Subcircuit definition** 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, 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.

In the **Model Builder** window, under **Results** click **Probe Plot Group 1**.

