



# 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 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 0.5V 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  $10\mu\text{s}$  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.

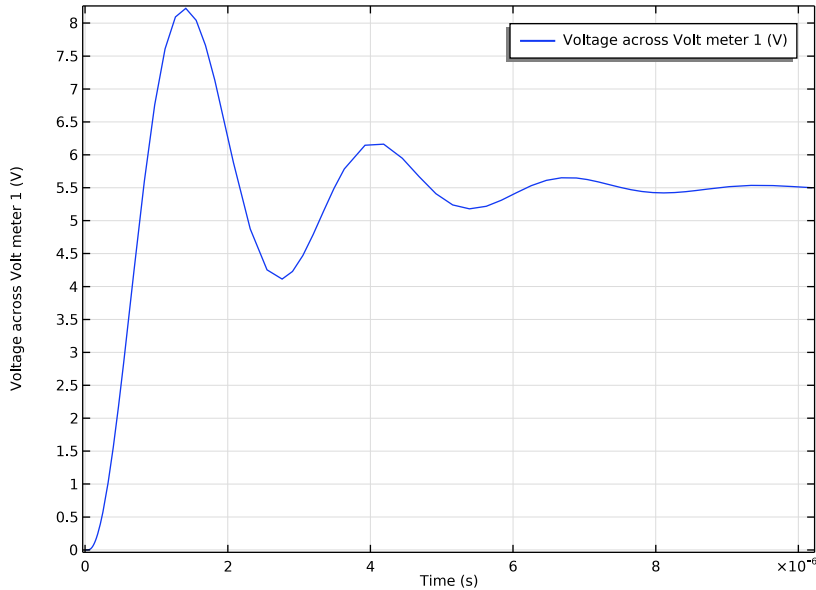


Figure 1: The output voltage of the op-amp as a function of time.

## Reference

---

1. 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 **Functions** and choose **Local>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	$.5[V] * \text{step1}((t - .05[\mu s]) / 1[\mu s])$	V	Input voltage

### ELECTRICAL CIRCUIT (CIR)

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

#### Subcircuit Definition I

- 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 R1

- 1 In the **Electrical Circuit** toolbar, click **Resistor**.
- 2 In the **Settings** window for **Resistor**, type Resistor\_RIN in the **Label** text field.
- 3 Locate the **Identifier** section. In the **R** text field, type IN.
- 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 E1

- 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 Locate the **Identifier** section. In the **E** text field, type GAIN.

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 R1*

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

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

3 Locate the **Identifier** section. In the **R** text field, type P.

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 C1*

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

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

3 Locate the **Identifier** section. In the **C** text field, type P.

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 E1*

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 Locate the **Identifier** section. In the E text field, type BUFFER.
- 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 R1

- 1 In the **Electrical Circuit** toolbar, click **Resistor**.
- 2 In the **Settings** window for **Resistor**, type Resistor ROUT in the **Label** text field.
- 3 Locate the **Identifier** section. In the R text field, type OUT.
- 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 V1

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 Locate the **Identifier** section. In the V text field, type IN.
- 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 $XI$

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 Locate the **Identifier** section. In the **X** text field, type OPAMP.
- 4 Locate the **Node Connections** section. From the **Name of subcircuit link** list, choose **OPAMP**.
- 5 In the table, enter the following settings:

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

### Resistor $R1$

- 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 $R2$

- 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 C1

- 1 In the **Electrical Circuit** toolbar, click **Capacitor**.
- 2 In the **Settings** window for **Capacitor**, type Capacitor CLOAD in the **Label** text field.
- 3 Locate the **Identifier** section. In the **C** text field, type LOAD.
- 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

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

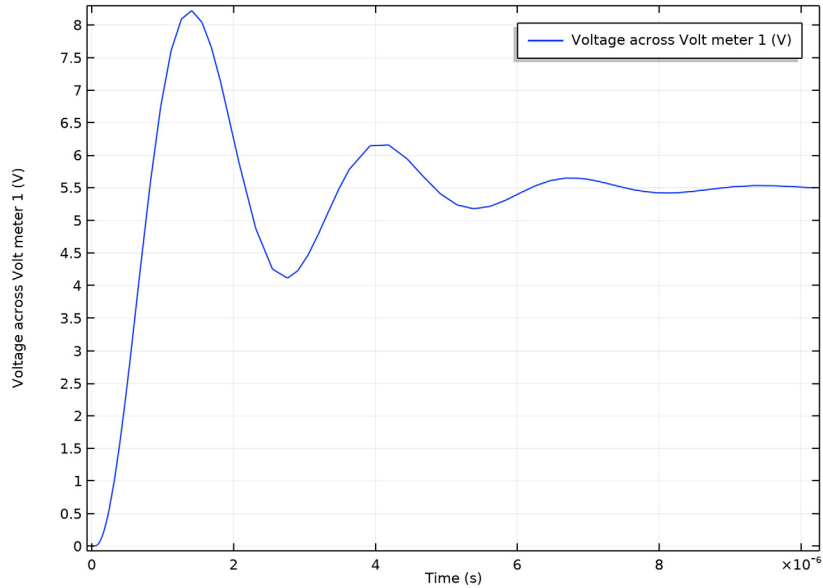
### Step 1: Time Dependent

- 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**.