QUICK START TUTORIAL

CREATING A PROJECT STEP-BY-STEP

Welcome! We assume that you have successfully installed Pintar ***V*irtuaLab**$_{TM}$ Electronics on your computer. You are now ready to create your very first virtual electronics experiment using the powerful Pintar ***V*irtuaLab**$_{TM}$ Electronics. Creating a virtual experiment with Pintar ***V*irtuaLab**$_{TM}$ Electronics is very similar to doing it in a real laboratory – linking together components into a circuit and testing its validity. But doing it in a virtual laboratory is even more fun because you can smoke the experiment without fear of anyone chasing after you. Let's see you do that in your school lab! So, let's get going…

INTRODUCTION

Diodes have the ability to conduct current in one direction and block current in the other direction. They are used in circuits called rectifiers that convert AC voltage into DC voltage. Rectifiers are found in all AC power supplies that operate from an AC voltage source. Power supplies are an essential part of all electronics systems from the simplest to the most complex.

In this tutorial, the goal is to put together a simple experiment that would simulate the conversion of an AC source to DC. For a novice user, the quickest way to become familiar with Pintar ***V*irtuaLab**$_{TM}$ Electronics is to follow this step-by-step tutorial. Throughout this tutorial, you will find quick references to the detail description, as indicated by the (?) symbol.

This tutorial also assumes that you possess a working knowledge of Windows 9x, NT, 2000, Me and XP. Certain words used to describe operations in this tutorial have specific meaning.

Click	Press once on the mouse button.
Double-click	Press twice on the mouse button in quick succession.
Select	Click once on a specific object.
Drag	Press the mouse button and drag, holding the mouse button down as you do so. Let go to the mouse at the intended location.
Type	Press on a specific key on the keyboard.
Choose	Select a menu item.

Launching Pintar *V*irtuaLab$_{TM}$ Electronics

In the Electronics folder, double-click on the application icon named TRON.EXE. When the Pintar ***V*irtuaLab**$_{TM}$ Electronics information panel appears, click on the information panel and it will go away and showing a new untitled Workbench.

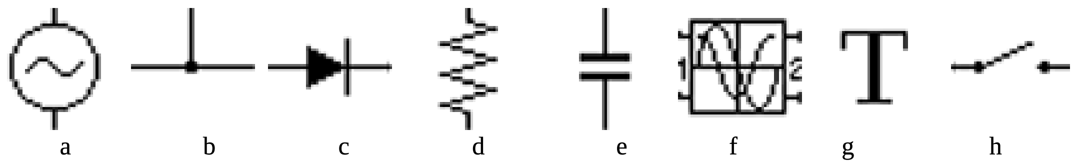
EXPERIMENT

OBJECTIVE:

To convert an AC voltage to a DC voltage (full-wave rectification).

COMPONENTS USED:

For this experiment, you would need the following components:



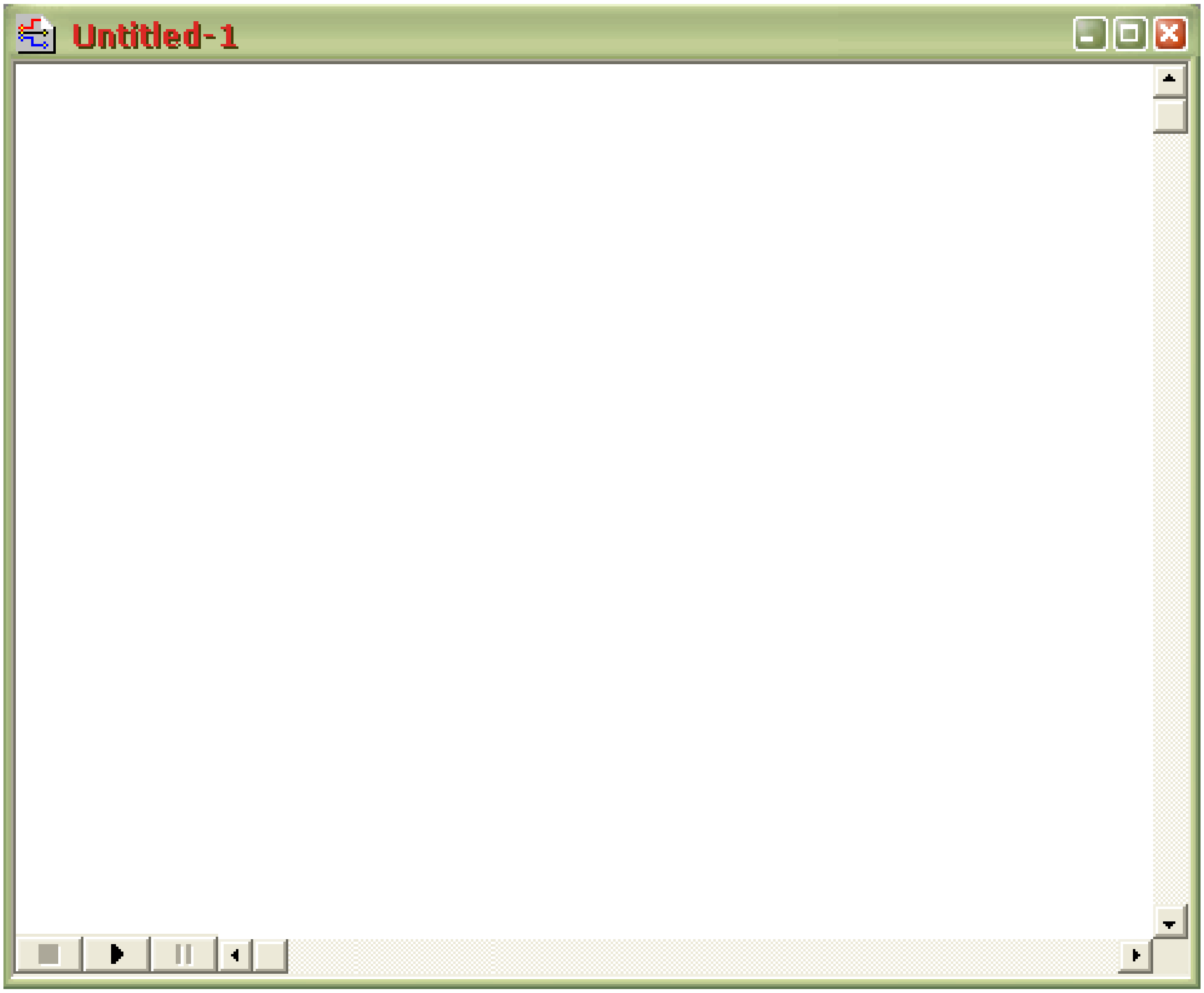
a. sine wave generator
b. connector
c. rectifier
d. resistor
e. capacitor
f. oscilloscope
g. text
h. switch

METHOD:

1. Starting a new project

To start on a new experiment, choose 'New...' from the File menu. A new empty Workbench with the tittle 'Untitled-1' will open.

? • **Starting a new experiment**



2. Placing a component on the Workbench

All components are kept in the Components manu. Select the components you need (see 'Components Used' above) for your experiment and place them on the Workbench.

a. Click on the Components menu. A categorical list of components appears.
b. Slide your cursor down the list; and for each category of components, a second tier menu listing the individual components appears.
c. Click on the components you seek. The cursor changes into the shape of the component.
d. Move the cursor to the spot on the workbench where you want to place the component and click. The chosen component is transferred onto the Workbench.
e. Repeat Step 2 until you have placed all the required components onto the Workbench.

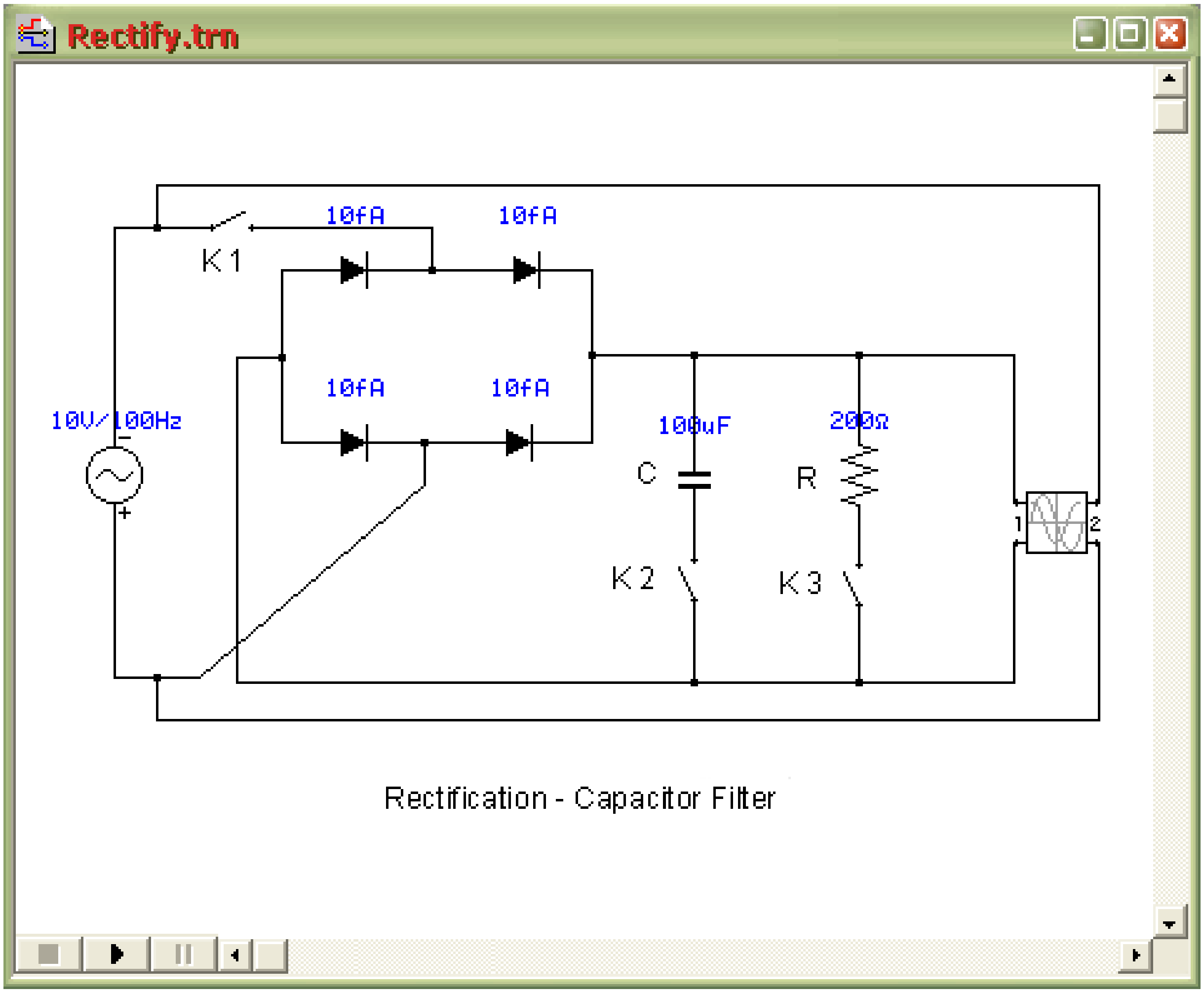
? • **The Workbench Environment**

? • **The Components menu**

? • **Transferring a component from the Components menu onto the Workbench**

3. Connect the components to create a circuit

After transferring the required components onto the Workbench, you can then proceed to connect the components, so that your circuit resembles that shown in the diagram below (*Fig 2*):



? • [Connecting components together]

? • [Rotating a component on the Workbench]

? • [Creating a bend in a connection]

Note: The sine wave generator is used to represent an AC power source, and its signal is fed directly into channel 2 of the oscilloscope. Using connectors, the same signal from the sine wave generator is passed through a group of four rectifiers, whose task is to change the full sine wave into one-way sine signal (containing only positive values). Finally, the function of RC-filter is to flatten the one-way sine signal.

4. Set appropriate parameters for components

Set the properties for the components:
Sine wave generator – peak: 10V; frequency: 100Hz
Resistor: 200W

Capacitor: 100uF

Oscilloscope:

a. Select both channels to measure in volts (V)
b. Select the checkbox ‘Visible’, which will open the display panel of the oscilloscope

? • **Specifying the parameters of a components**

5. Annotate your experiment

It is always a good practice to annotate or label your experiment. Label the diodes, fixed resistor, capacitor, switches and the oscilloscope; and give a caption to the experiment.

>The oscilloscope already has a default name. If you wish, you may want to change the name to something more appropriate.

a. Open the oscilloscope’s properties panel.
b. Retype the default name with a new name.
c. Click OK.

As for the resistor, capacitor and the caption, label according to *Fig 2*,

a. Select the Text component from the Component menu.
b. Move the cursor to the Workbench and click at the location where you wish to place the text. A text object with the default word "Text" appears at the position where.
c. Next, double-click on the text object and the text properties dialog panel appears.
d. In the scrolling field enter the appropriate text that you wish to display.
e. Choosing from the options available, set alignment and font.
f. When satisfield with the setting, click ‘OK’.

? • **The Workbench environment**

6. Save your experiment

At this point, it would be a good idea to save the experiment that you have created thus far.

Note : If you are using a free trial version of Pintar **VirtuaLab**$_{TM}$ Electronics, you will not be able to save your experiment because this feature has been disabled. Continue with Step 7.

a. Selecting ‘Save As...’ from the File menu.
b. Name the project, "FirstExp". You may want to save your project in the examples folder.

? • **Saving an experiment**

7. Play your experiment

Finally, you are now ready to run your experiment.

a. Click on the 'Run' button in the Control Panel at the bottom right corner of the Workbench.

Observe the curves generated in the display of the oscilloscope. There is no output from Ch1, and the output from Ch2 equals the input directly from the sine wave generator (*Fig. 3*). Pay particular attention to the red curve (Ch1) which represents the resultant DC waveform. The blue curve (Ch2) represents the untreated waveform generated by the sine wave generator.

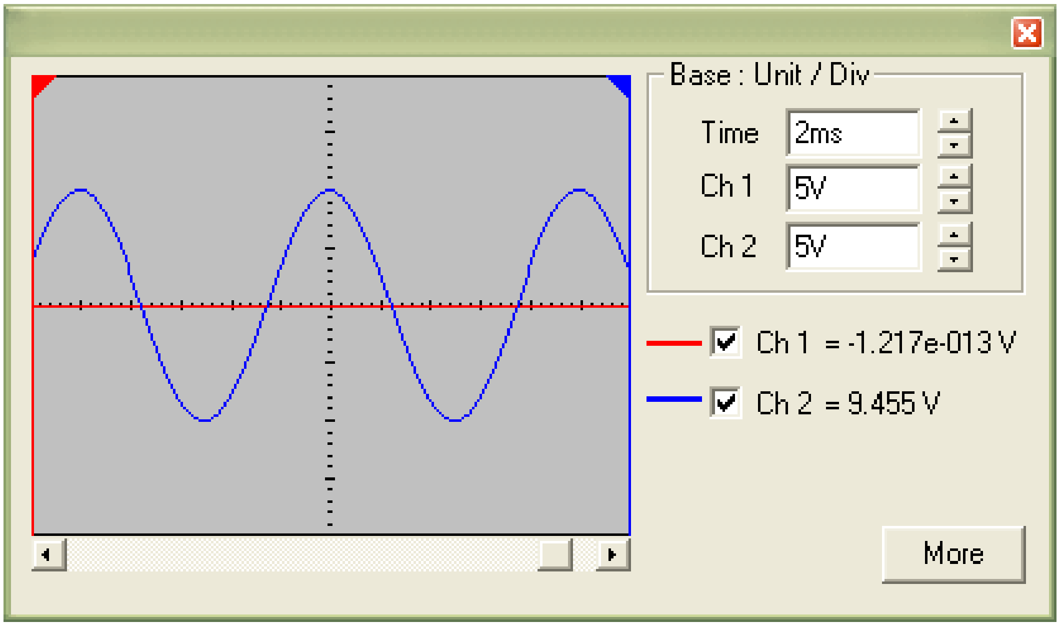


Fig 3

b. Close the portion of the circuit leading to the group of rectifiers by clicking on the switch K1. Observe again the curves generated in the display of the oscilloscope. Wait for the waveforms to stabilize.

Ch1 = the result of the four diodes (rectifiers), it is called rectification. At the point prior to rectification, the sine wave value is from –10 to +10. After rectification, the output is positive (ideally, Ch1=0 to +10), but is still not a DC.

When the input cycle is positive, diodes D1 and D2 are forward-biased and conduct current. Diodes D3 and D4 are reverse-biased. During negative half-cycle, diodes D3 and D4 become forward-biased and conduct current, while diodes D1 and D2 have become reverse-biased.

Pintar **VirtuaLab**$_{TM}$ Electronics simulates real world conditions and as such the waveforms you see in your experiments would probably differ from the idealized waveforms shown in school textbooks. The ideal wave should be between 0--+10V.

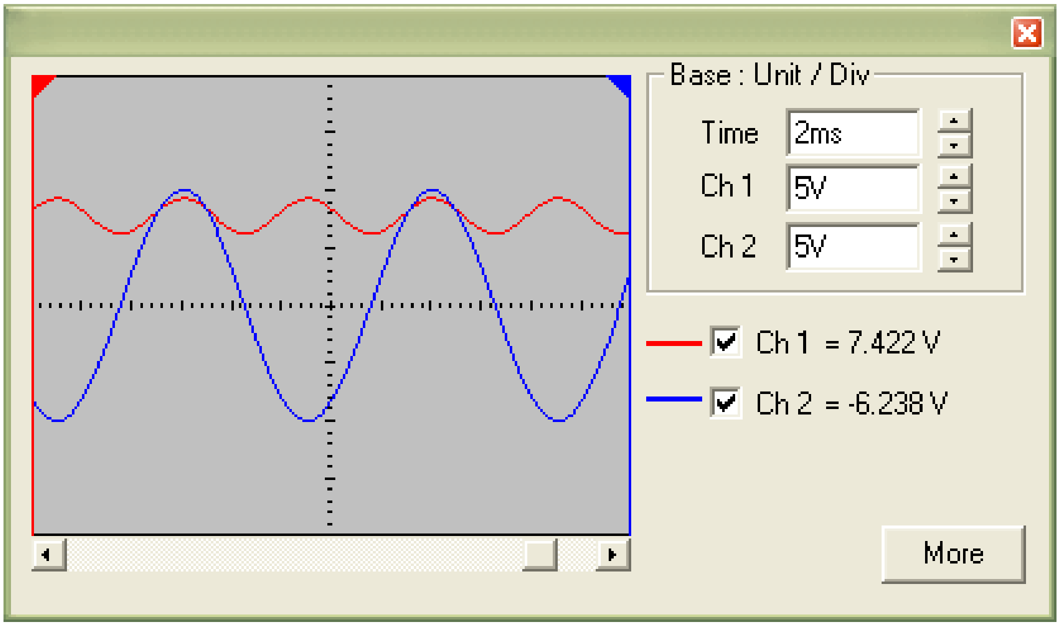


Fig 4

c. In this step, the goal is to add a filter to reduce the fluctuations and produce a nearly constant-level DC output. This filtering can be achieved through the addition of a capacitor.

Close the switch K3 (switch K1 remains closed). The function of the capacitor C now comes into play. The red waveform changes, becoming more like DC power.

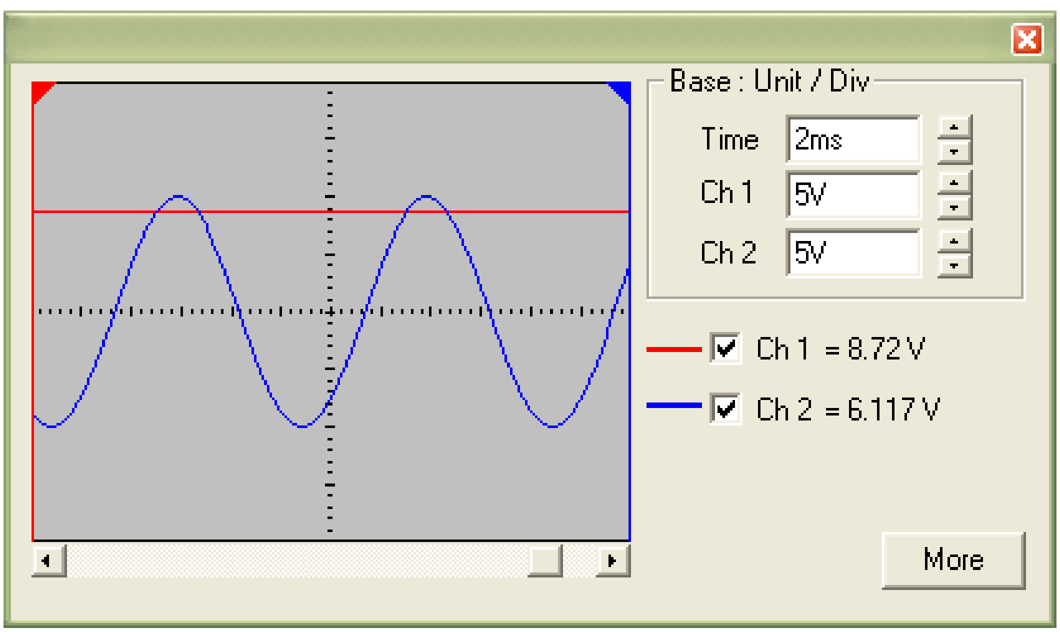


Fig. 5

d. Finally, to simulate the DC voltage in use, close the switch K2 (all three switches are closed). The addition of the resistor into the circuit is equivalent to connecting an electrical appliance to a DC power source, causing the ripples you now see in the red curve.

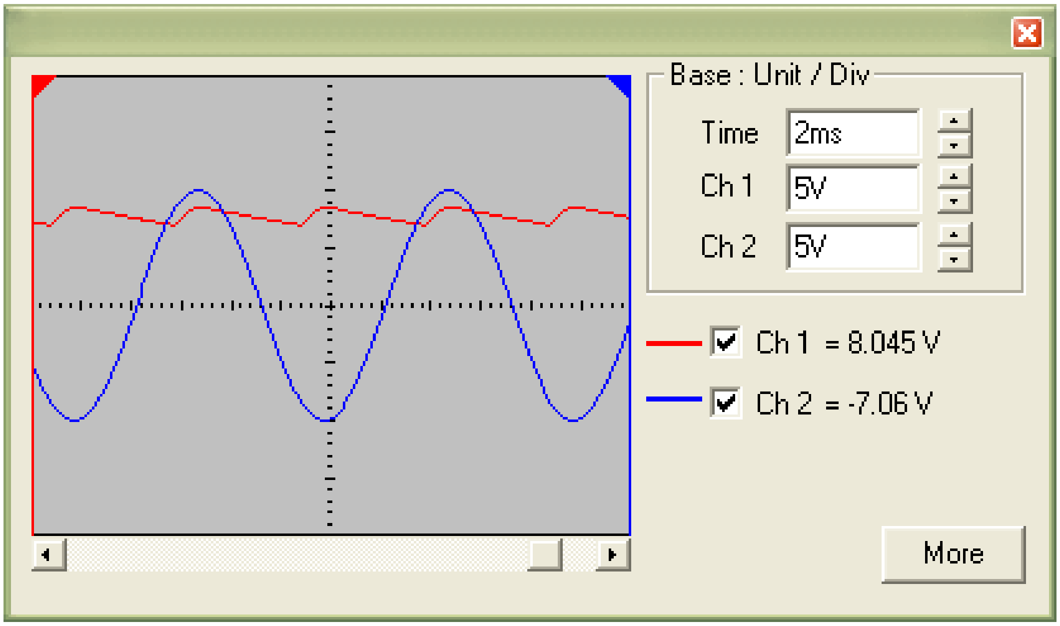


Fig. 6

? • **Playing and stopping your experiment**

8. Stop your experiment

Stop the project by clicking on the 'Construct' button.

9. Re-edit your project

Change the parameter of the fixed resistor to 10W.

10. One more time...

Change the resistance back to 200W but reduce the capacitance to 10uF. Repeat steps 7 and 8.

11. Yet another time...

Raise the resistance and capacitance to 500W and 200uF respectively. Repeat steps 7 and 8.

12. Ending your work session

Although this experiment is coming to an end, your learning adventure with the Pintar **VirtuaLab**$_{TM}$ Electronics is just beginning. Let your imagination loose and dream up experiments that escape the confines of earthly limitations! Select 'Quit' from the File menu.

CONCLUSION:

The ideal output would be a constant voltage to be used as a DC voltage source. But practically it is difficult to get such a fine result as can be observed in the waveform of channel 1 of the oscillator.

In the second part of the experiment (steps 9 to 11), it is observed that lowering either the resistance or capacitance would increase the magnitude of the ripples in the DC output while increasing the resistance and capacitance would minimize the magnitude. This rippling is an indication of the effectiveness of the filter. The effectiveness can be maximized by increasing the value of the filter capacitor.

go to top