

Step 1 : Copy link of **LTspice website** from **Assignments**.

Assignments



LTspice_practice_01

Download and install LTspice simulation program on your personal computer.

Once the LTspice program is successfully installed, DO NOT update it to a newer version at the update prompt notice of the program during the semester.

If you are already done with the procedure in the last semester with CET243, you do not need to redo it; just launch and screen capture and submit as follows:

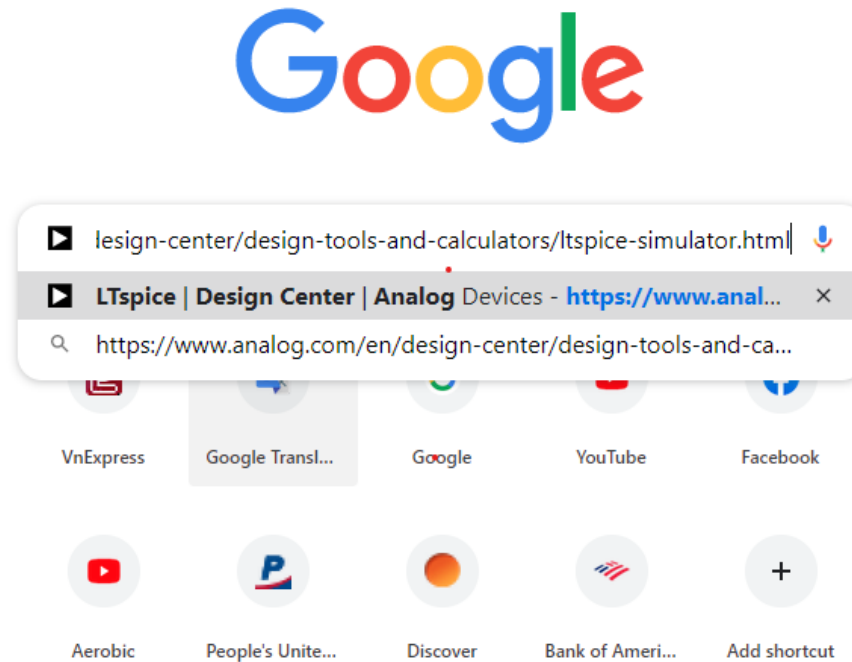
LTspice website

<https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>

From the above URL, you will download and install the proper version (Mac or Windows) LTspice program on your own computer at home.



Step 2 : Paste into **Google** search engine and press Enter.



Step 3 : The **LTspice** screen appears

AnalogDialogueEngineerZoneWiki

AHEAD OF WHAT'S POSSIBLE™

Search

COMPANYMYANALOGPRODUCTSAPPLICATIONSDSIGN CENTEREDUCATIONSUPPORTMY HISTORY

Design Center > Circuit Design Tools & Calculators > LTspice
Printmy.analog

Search

Simulation Models

Reference Designs

Evaluation Hardware & Software

Packaging, Quality, Symbols & Footprints

Circuit Design Tools & Calculators

- Amplifier & Linear
- Clock & Timing
- Data Converter
- LTspice**
- Power Management
- RF & Synthesis

Processors & DSP

LTspice

LTspice® is a high performance SPICE simulation software, schematic capture and waveform viewer with enhancements and models for easing the simulation of analog circuits. Included in the download of LTspice are macromodels for a majority of Analog Devices switching regulators, amplifiers, as well as a library of devices for general circuit simulation.

Contact Technical Support for assistance

Benefits of using LTspice

Our enhancements to SPICE have made simulating switching regulators extremely fast compared to normal SPICE simulators, allowing the user to view waveforms for most switching regulators in just a few minutes. This video provides an overview of the advantages of using LTspice in an analog circuit design and how easy it is to get started.

Download LTspice

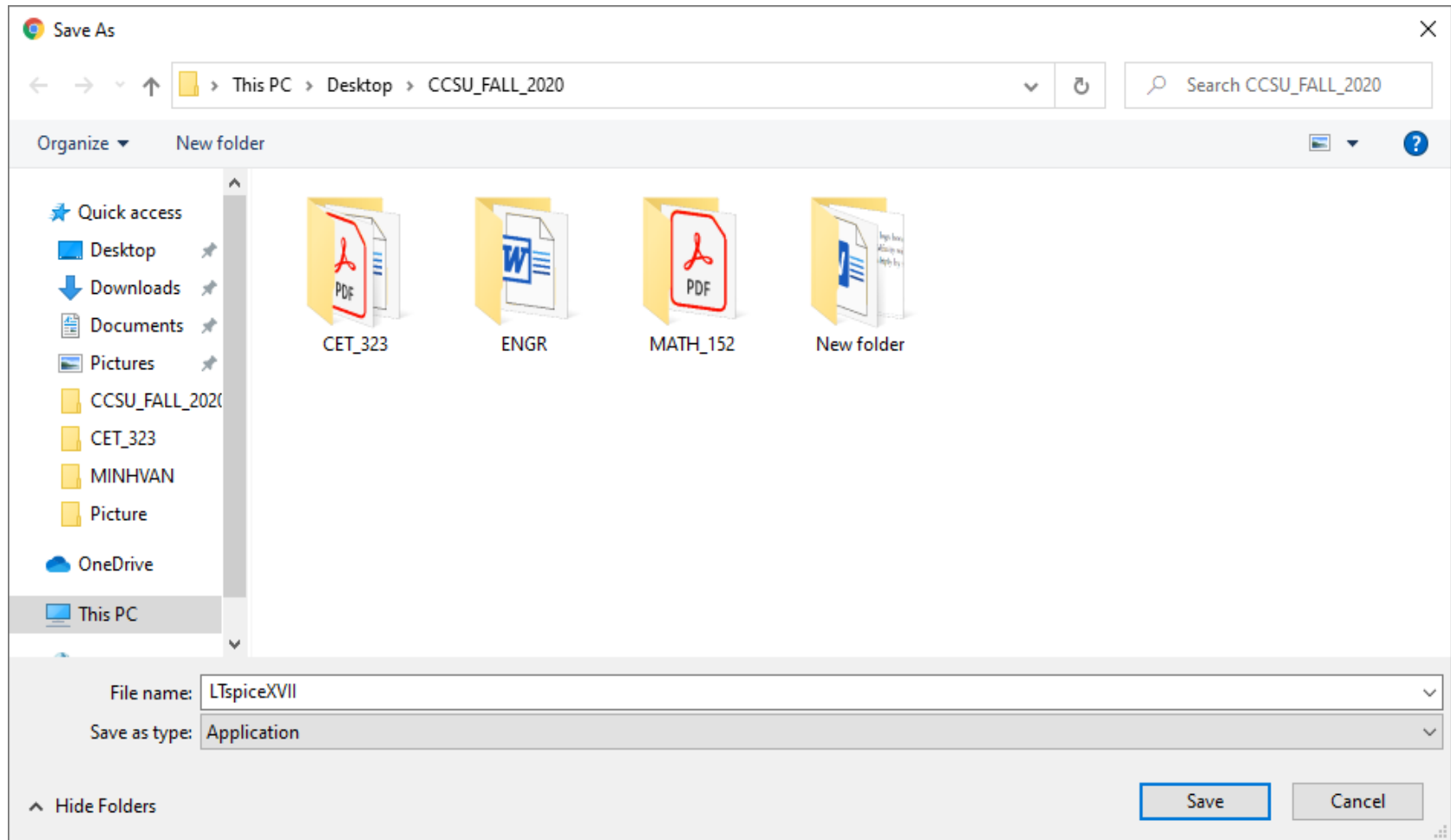
Download our LTspice simulation software for the following operating systems:

Operating System	Last Updated
Download for Windows 7, 8 and 10	Updated on Aug 22 2020 *
Download for Mac 10.9+	Updated on Aug 22 2020 *

[Download for Windows XP](#) (End of Support)

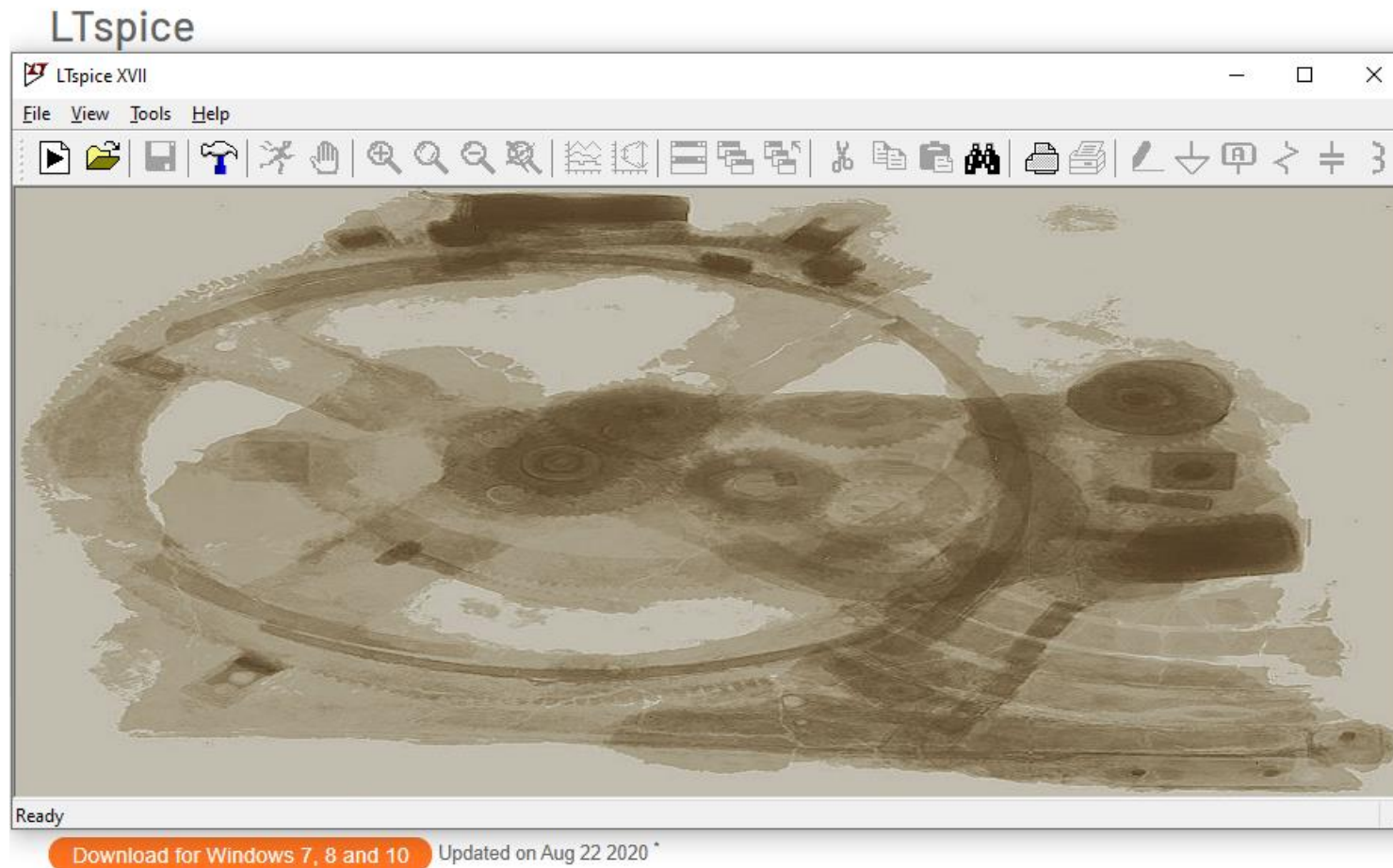
*date displayed reflects the most recent upload date

Step 4 : Choose Folder to **Save as ...** the file The LTspice.



Step 5 :

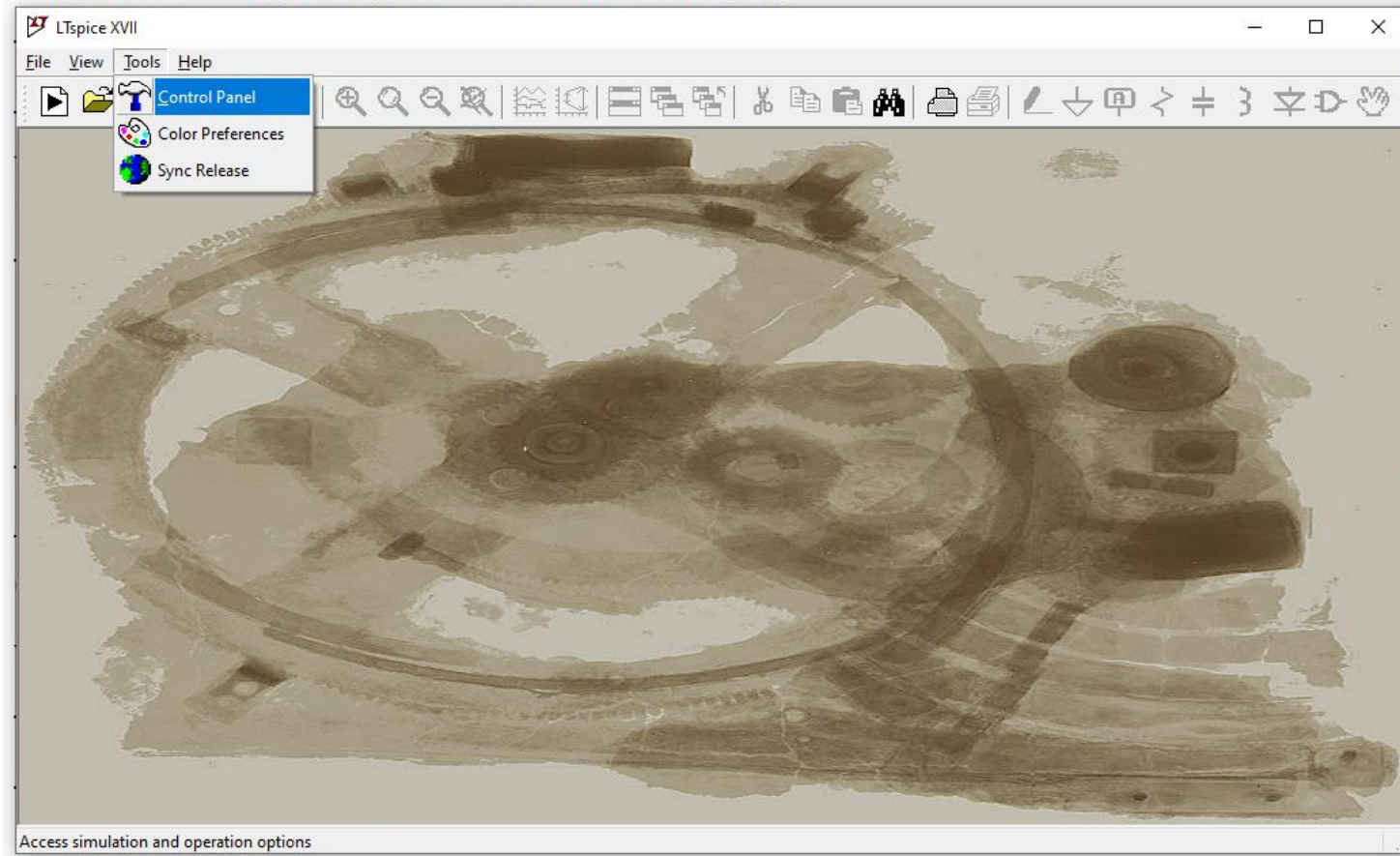
If appears on the screen, the **LTspice screen** contains the form. It means you have successfully downloaded.



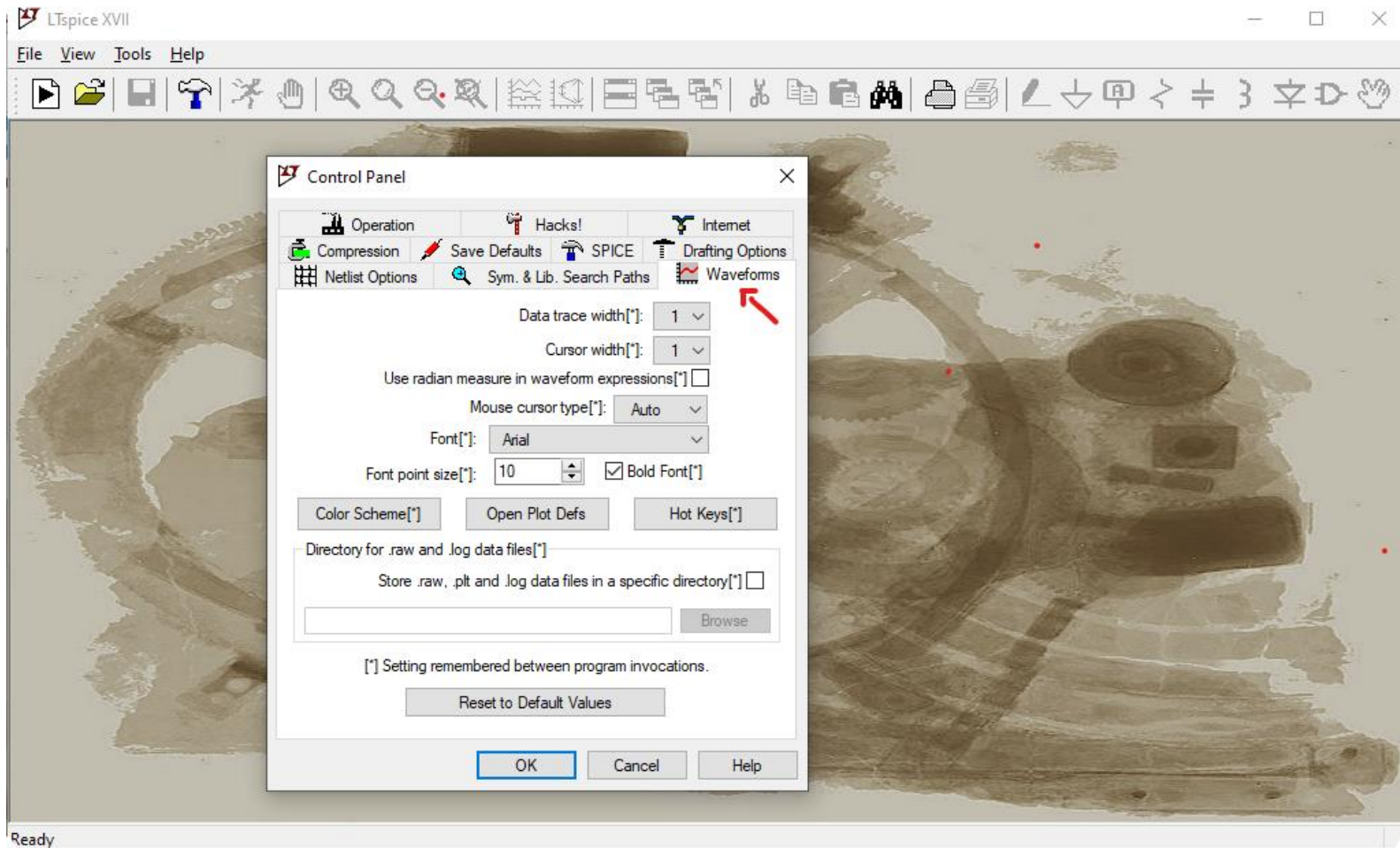
🏠 Change background of waveform display :

👉 Go to **Tools** -> **Control Panel**

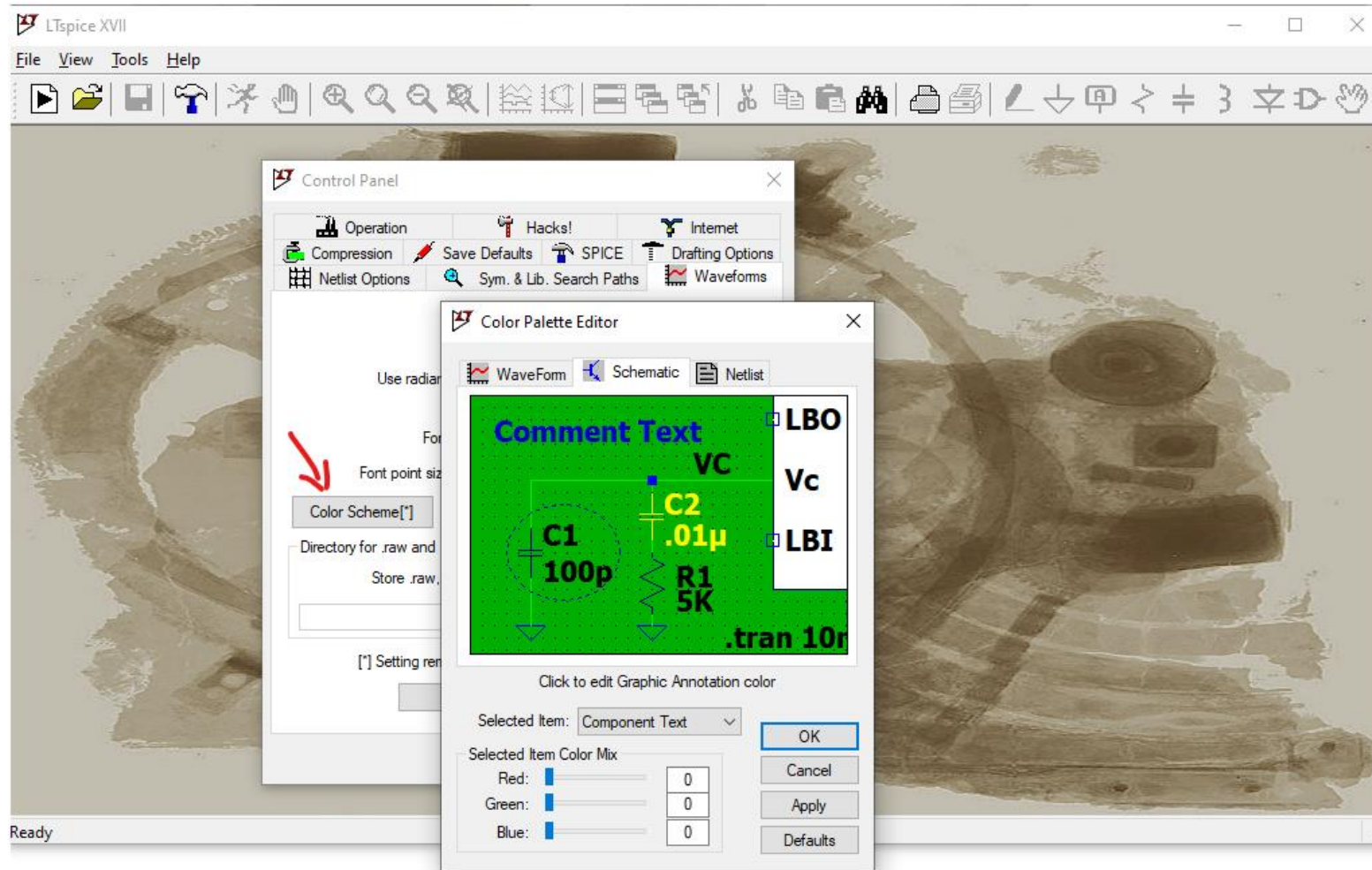
[TIP] how to change background of waveform display.



✌ From **Control Panel** window, select **Waveform** Tab

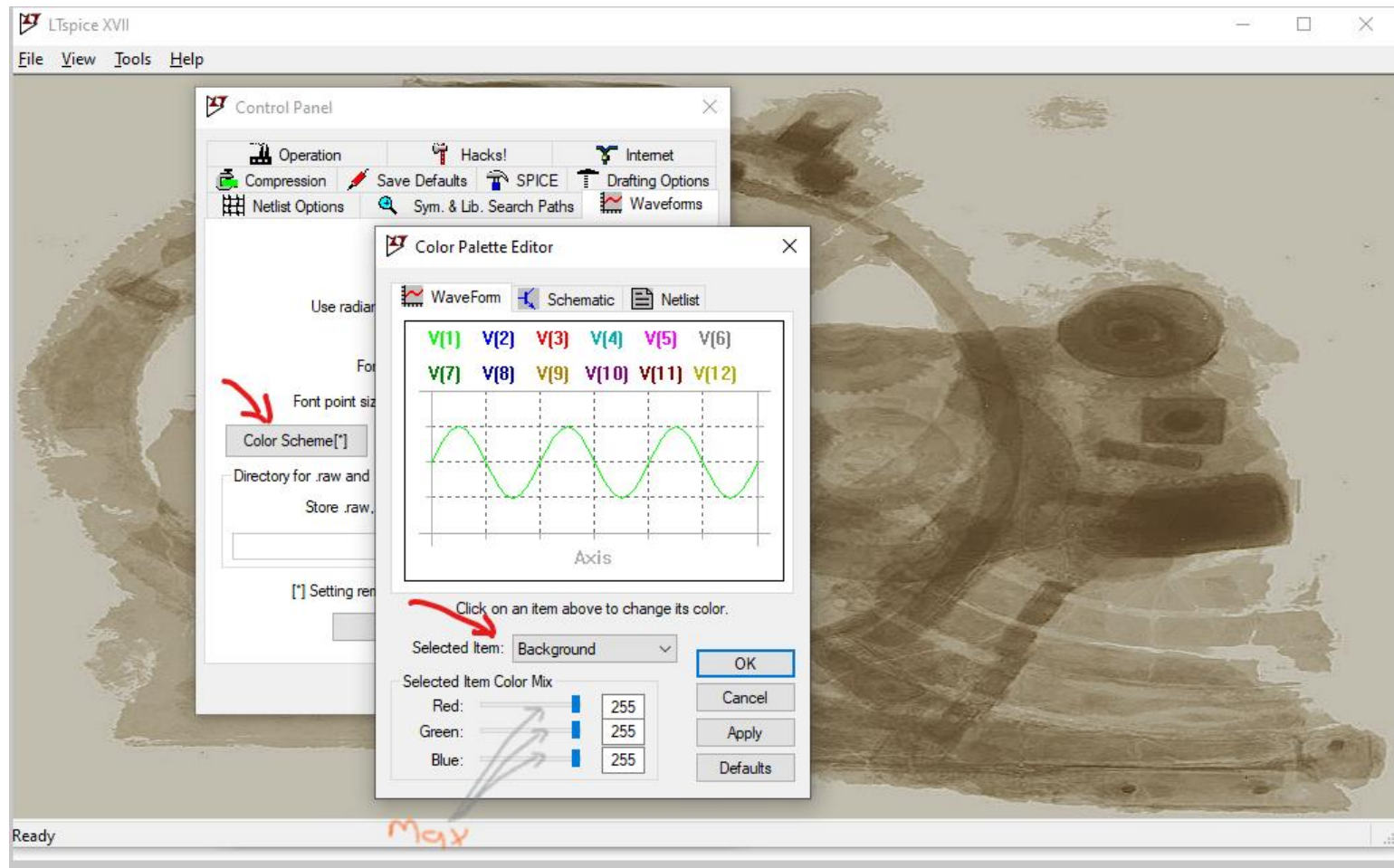


Click **Color Scheme** Tab => Appears *Color Palette Editor* window .



Drop-down menu of **Selected Item** choose **Background**

You can then move the Red, Green, and Blue sliders to maximum to white background. Then click OK> and click OK.



 Use 3 keys: [**Windows + Shift + S**] from the keyboard to take screenshots and paste in the Word window.

 Use the Save As ... window to *Convert* the word.doc file to. **PDF file.**

