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:
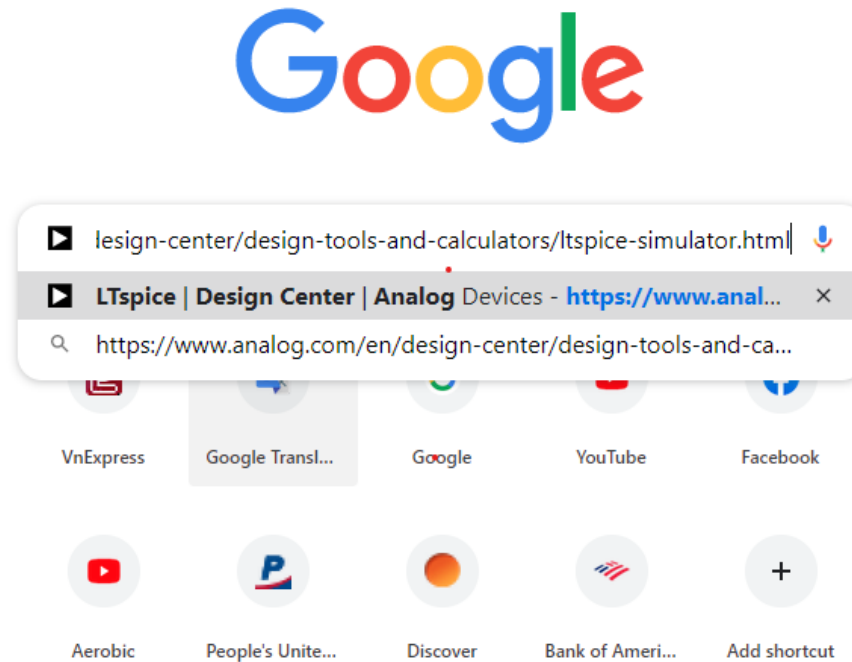
LTspice website

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.

ഽ✸ഽ

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

Step 3 : The **LTspice** screen appears

Step 4 : Choose Folder to **Save as** … the file The LTspice.
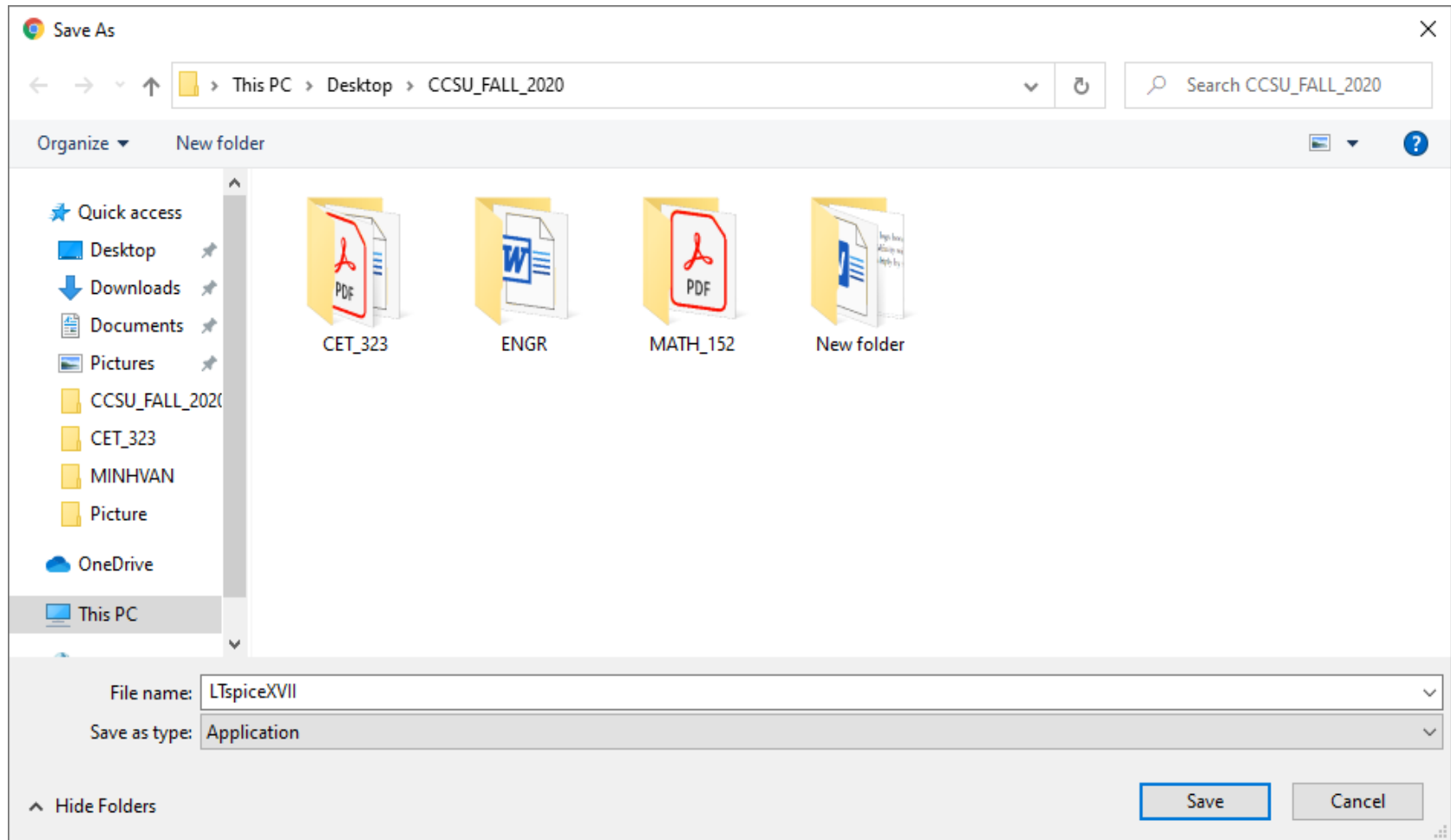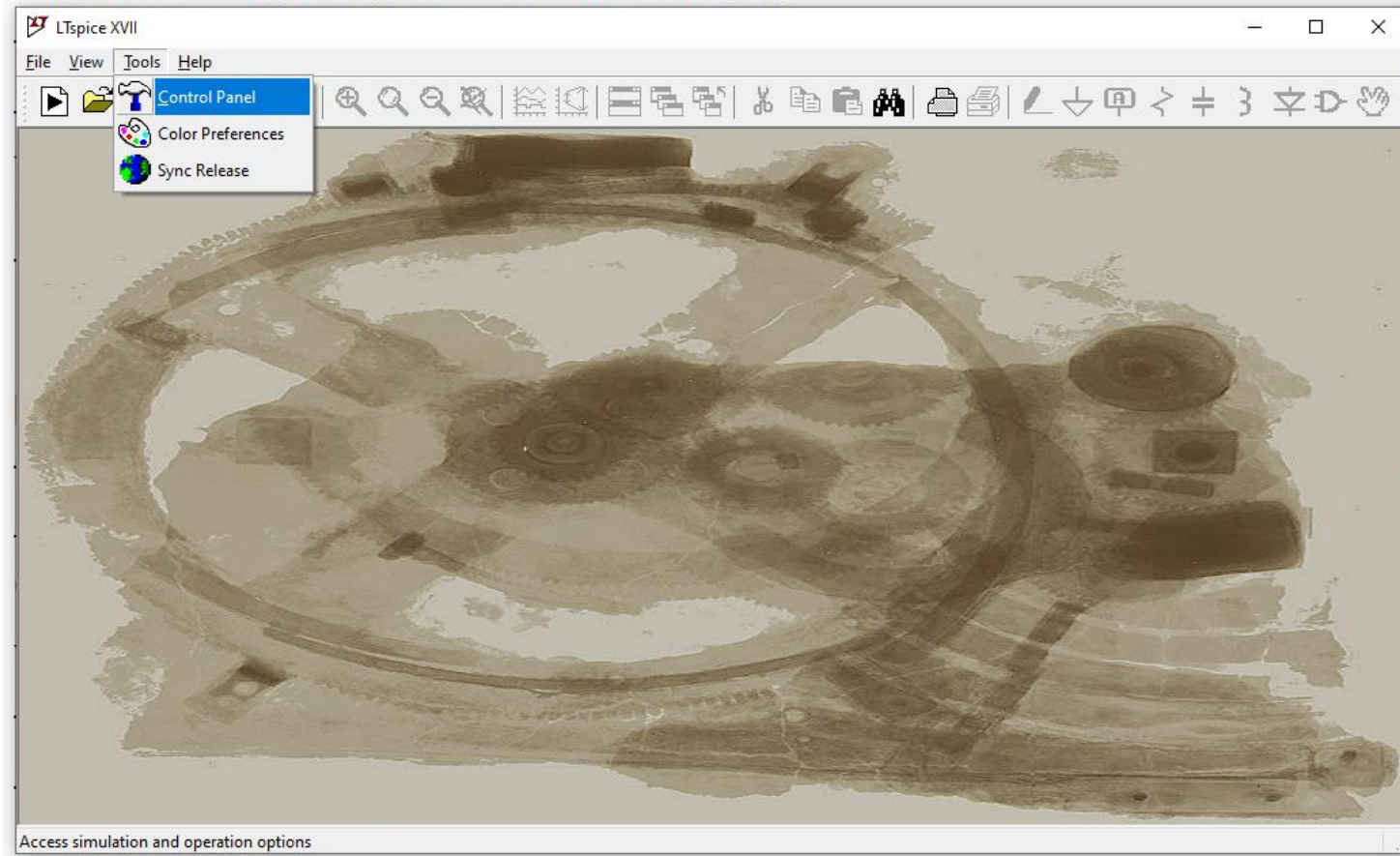
<u>Step 5</u> :

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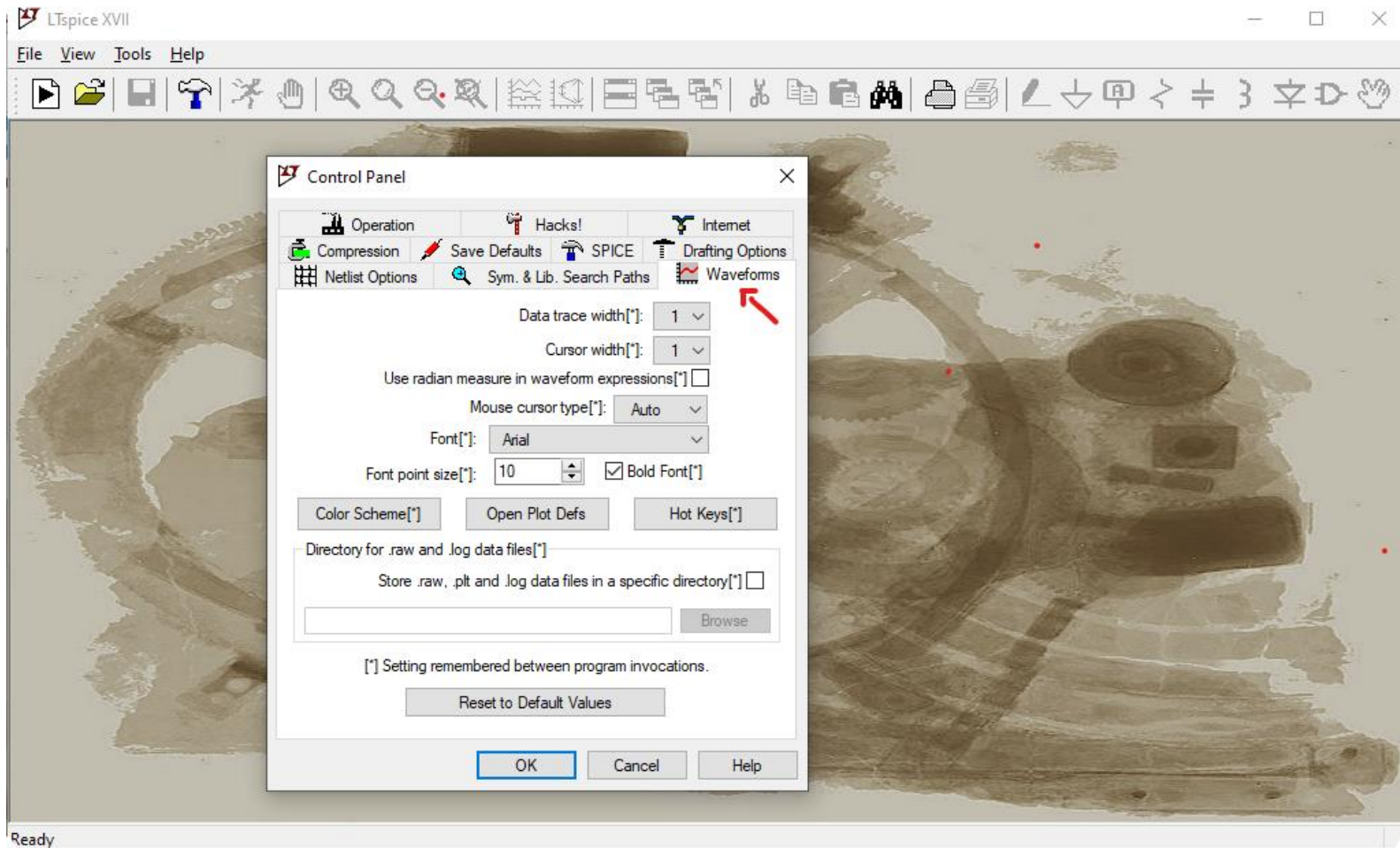

❉
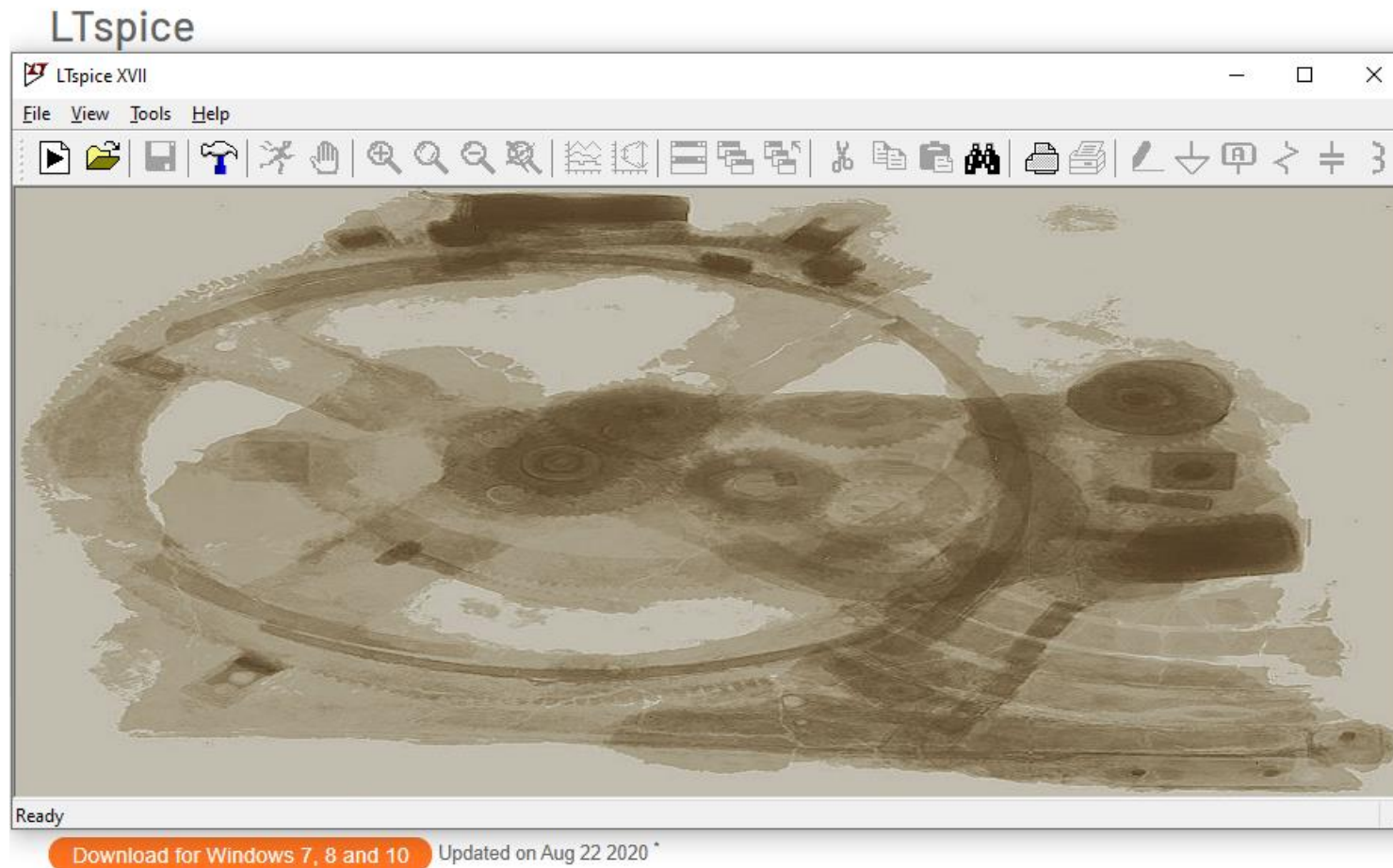
🏴 Change background of waveform display :
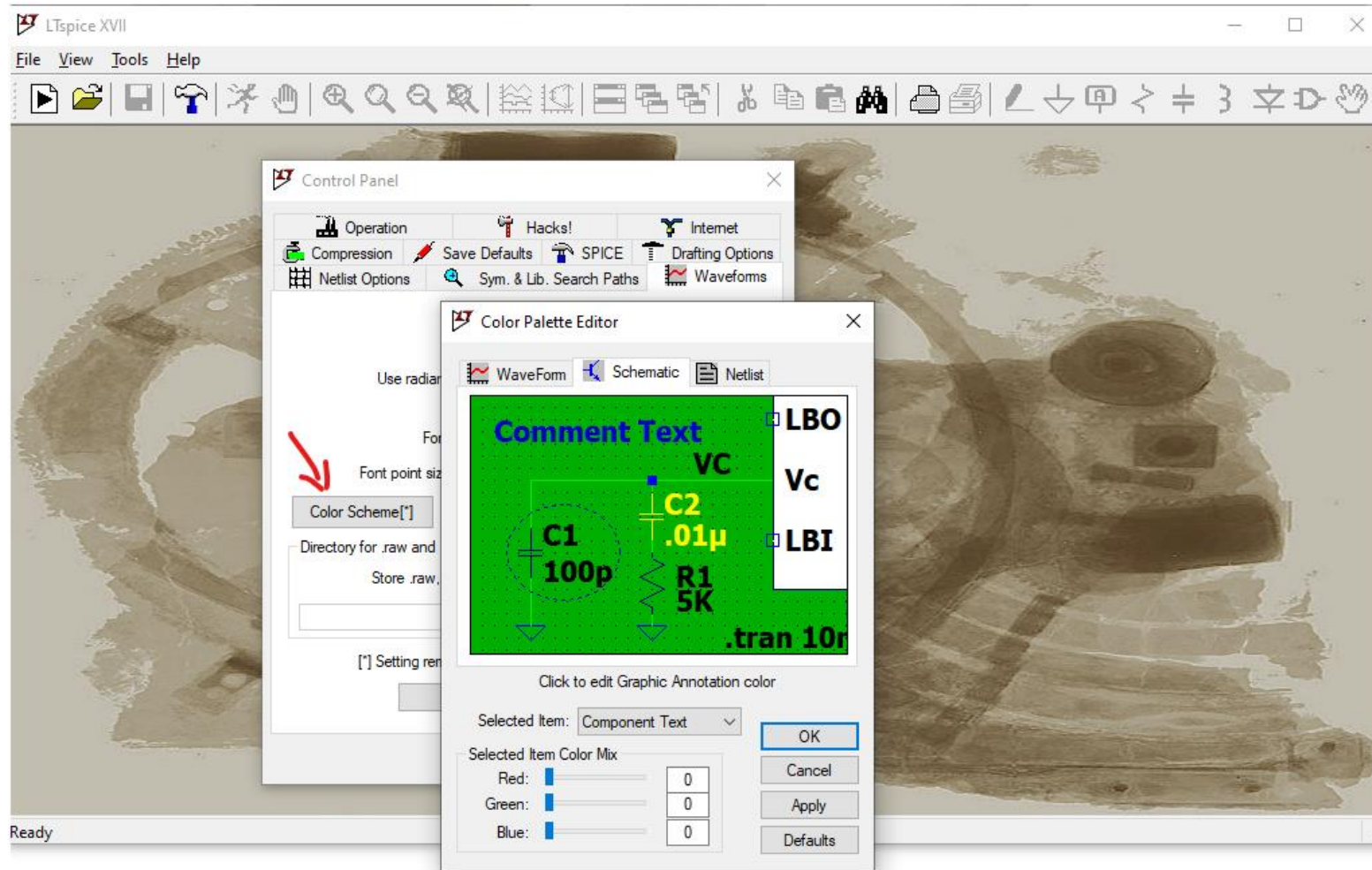
👉 Go to Tools -> Control Panel

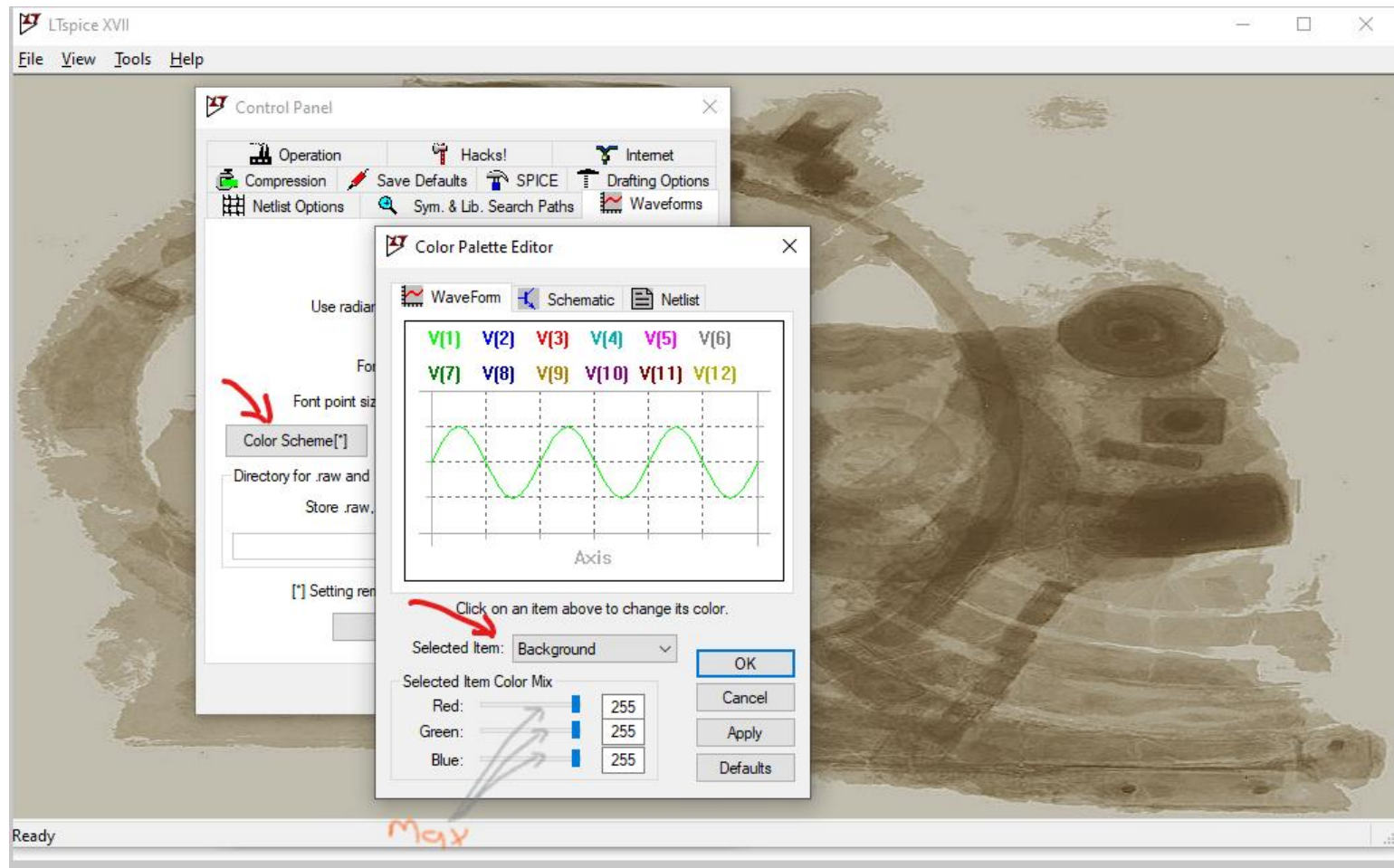✌ From **Control Panel** window, select Waveform Tab

❧✦❧

✍ 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.**