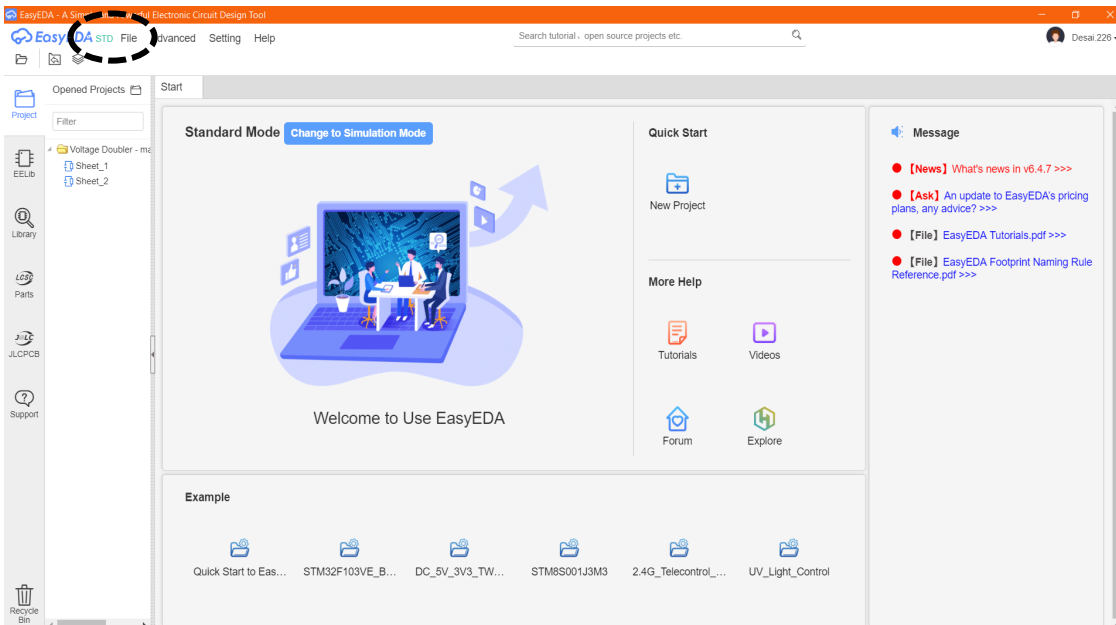


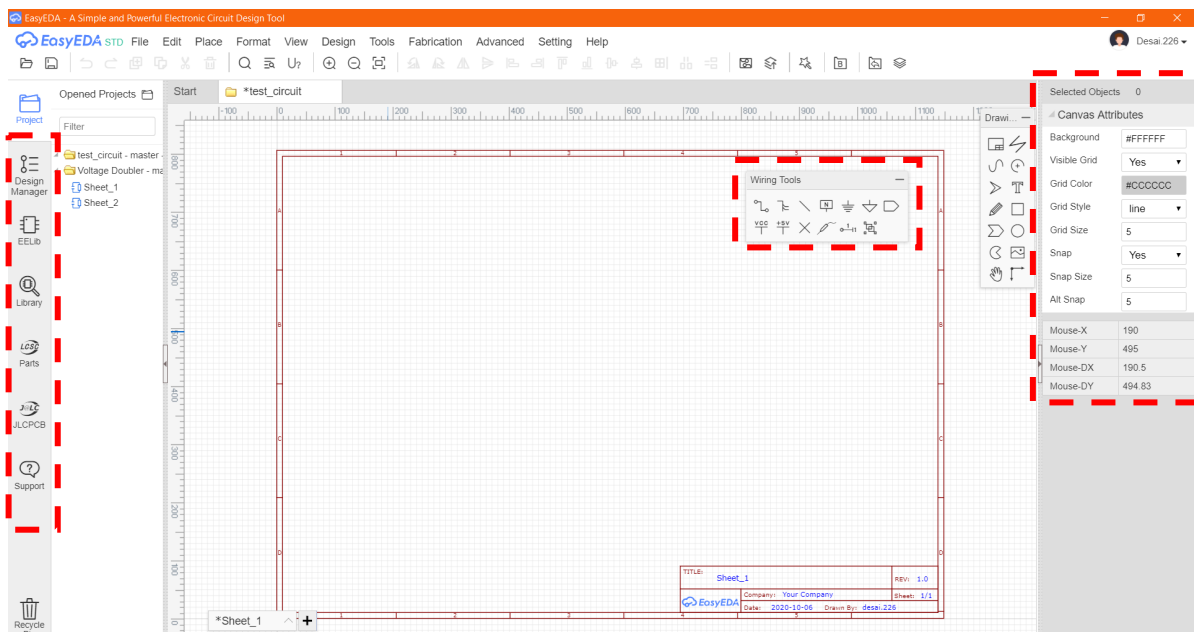
EasyEDA – Quick Tutorial

- Download EasyEDA from its website to locally build circuits for simulation.
- You can setup a free username and password for this.
- Once you have logged in, you will see the following screen:



Building a circuit:

- You have two modes to setup you circuit:
 - Standard Mode – build your circuit
 - Simulation Mode – build your circuit and run the simulation. You can use this directly before building the circuit by clicking on ‘STD’ on the top title bar of the login page as shown in the figure above.
- Using the File tab, you can start a new project. Label the project and click on Save button.
- In the circuit sheet you can build your circuit.



- There are three sections in the design sheet as highlighted above for the circuit design:

- Library of components on the left-hand side of the sheet
- Wiring tools
- If needed, you can change the value of the component in the attribute section, on the right-hand side of the sheet
- All the basic components are in the EELib tab in the library of components. In some cases, if you need to search for a particular component, you can use the Library tab.
- The following figure shows all the components in the EELib tab in the Simulation mode.



- To put a component on the design sheet, simply click on the component and release the right mouse key and bring your mouse arrow to the design sheet to drop in the component.
- With this method you can drop copies of the component in the design sheet. Hit ESC key on your keyboard to stop dropping more copies of the same component.
- For PC user, if you mouse over the component, a small triangle pops up, which gives you the option to choose a type of component. For example, for a Vcc component, it has options to choose either a +5V supply or a -5V supply and more. For Mac user, you can click on Vcc to change it to +5V or -5V for example.
- To connect the various components together and to the power supply or oscilloscope you should use the Wire (W) from the “Wiring Tools”.
- Once you have placed a passive component, you can change its value in the attribute section. First double click on the component and then change the values of the component. Remember for a resistor of 1 Meg you must type in the value of 1 Meg and not 1 M. EasyEDA does not distinguish between m and M. Both stand for milli.
- Use a signal generator to apply an AC signal such as sine or square wave.
- The circuit must contain a ground component in order to run the simulation. This allows the signal to be referenced with respect to ground. If it is inappropriate to connect the circuit to ground, add a ground without connecting to the circuit.
- Use the oscilloscope to displace the output signals. It works the same as a desk oscilloscope. To visualize the waves in channels 1 and 2, click on the oscilloscope in the design sheet and the attribute section to change the oscilloscope settings. Click on view meter and it will pop up a window with the waveforms in channels 1 and 2.
- For a digital circuit, you can use “Logicstate” as the input to a logic gate and “Logic_Probe” to look at the output of a circuit. After you run the simulation, the probe should indicate the status of the circuit.
- For a circuit with JK flip-flops (JKFF’s), use a “Logic Analyzer” to indicate the output of circuit.
- Unlike the standard JKFF in the lab, the JKFF in EasyEDA has the following unusual characteristics:
 - It triggers on the rising edge of the clock instead of the falling edge. Add an inverter to the master clock so that it is easier to design the counter.

- Reset (R) and set (S) operate at active high, i.e. tie it high (5 V) if you want it to reset or set the output Q (and Q bar).
- Q (Q bar) is high (low) when the JK is just being turned on ($t = 0$).
- For a DACs, use “Voltage_Source” to set the input voltages and multimeter to measure the output voltage.

Run Simulation:

- Press CTRL+J and it will pop a window where you can select the type of simulation you want to run:
 - Transient, AC Analysis, DC sweep, DC transfer, DC op pnt.
 - For our circuits, we will use one of the first three choices.
- For the transient analysis, set the window of start and stop times and click on run for the simulation.