

Instruction for LTspice

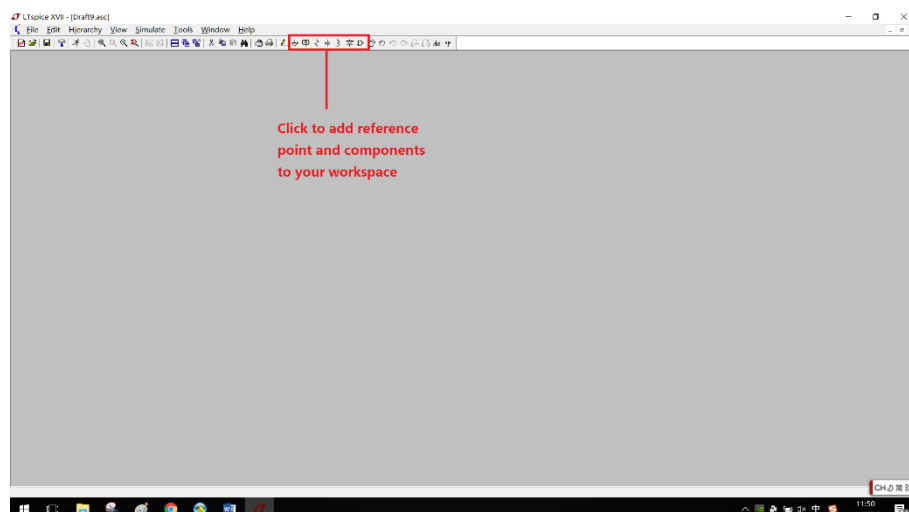
This piece of instruction aims to give a brief tutorial of LTspice for semiconductor device and power electronic circuit simulation. LTspice is a powerful software that permits to conduct time-domain simulation and parameter sweep. For the following part, examples will be provided to illustrate the usage of this software.

1. Build-up

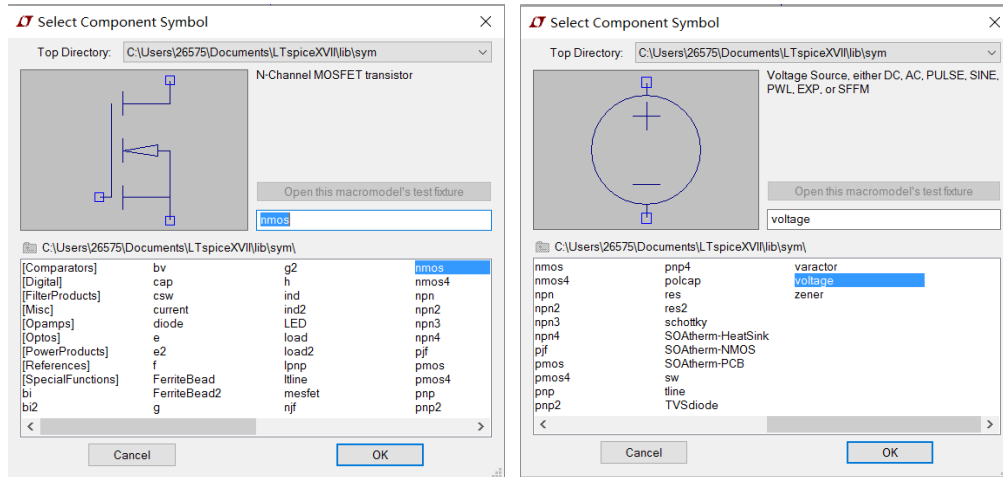
Open the software LTspice and you will see the window as below. Click on the icon *New Schematic* to create a new workspace.



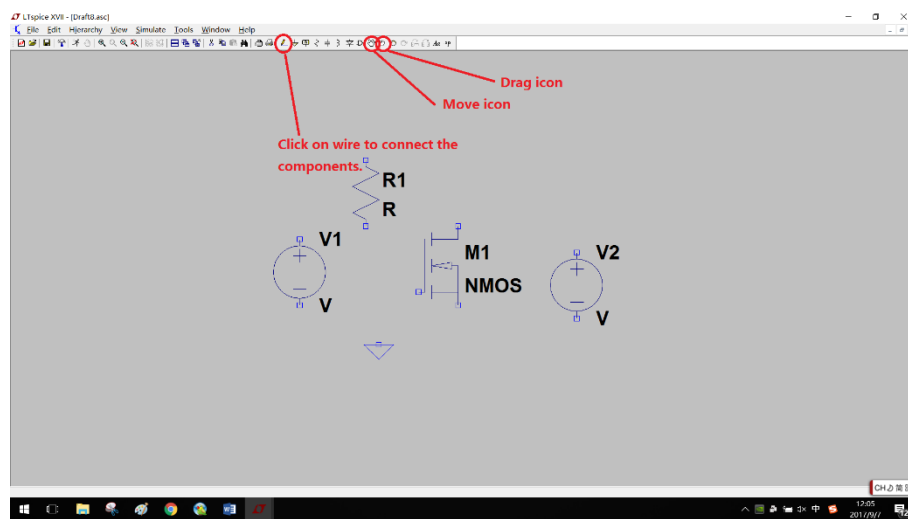
Then in your workspace, add reference point and components.



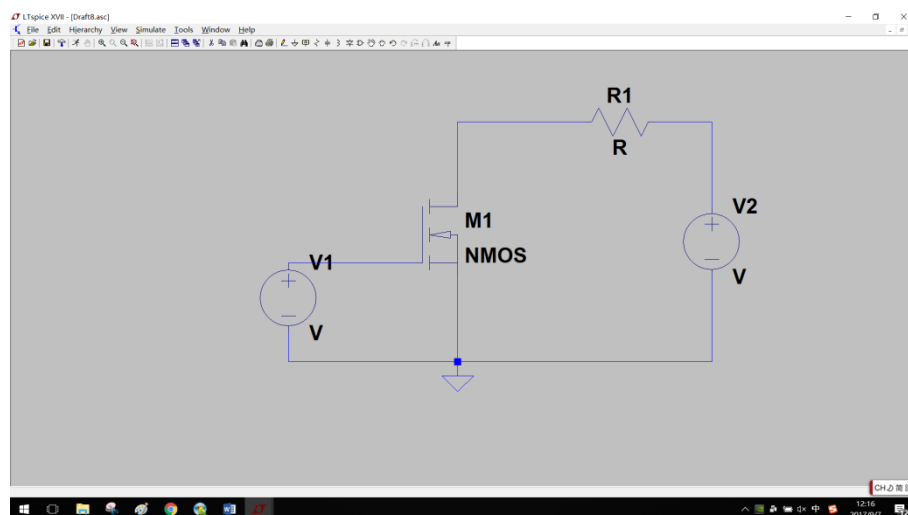
Click on the icon *component* then add nmos and voltage source shown as below.



Now you have got all the components you want in your workspace.

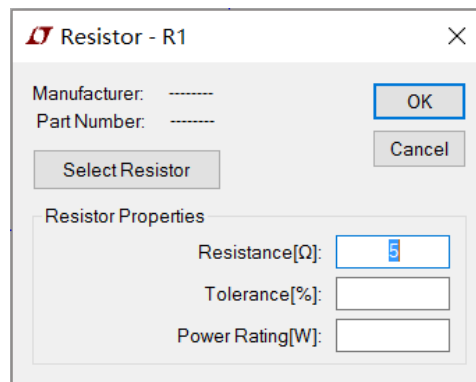


Rearrange the circuit and click on the icon wire to properly connect them together. You will find the icons for changing the position quite useful. Your circuit topology should be like as below.



Right-click on each component to configuration. Right-click on text would enable to change the name of each component.

- Resistor



Resistor - R1

Manufacturer: -----

Part Number: -----

Select Resistor

OK

Cancel

Resistor Properties

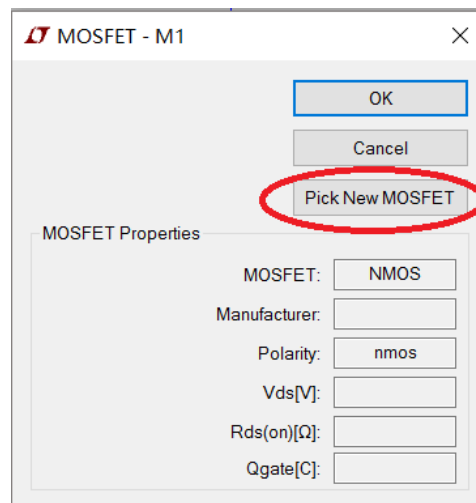
Resistance[Ω]: 5

Tolerance[%]:

Power Rating[W]:

Right-click on resistor **R1** and configure the resistance to be 5 Ohm.

- MOSFET



MOSFET - M1

OK

Cancel

Pick New MOSFET

MOSFET Properties

MOSFET: NMOS

Manufacturer:

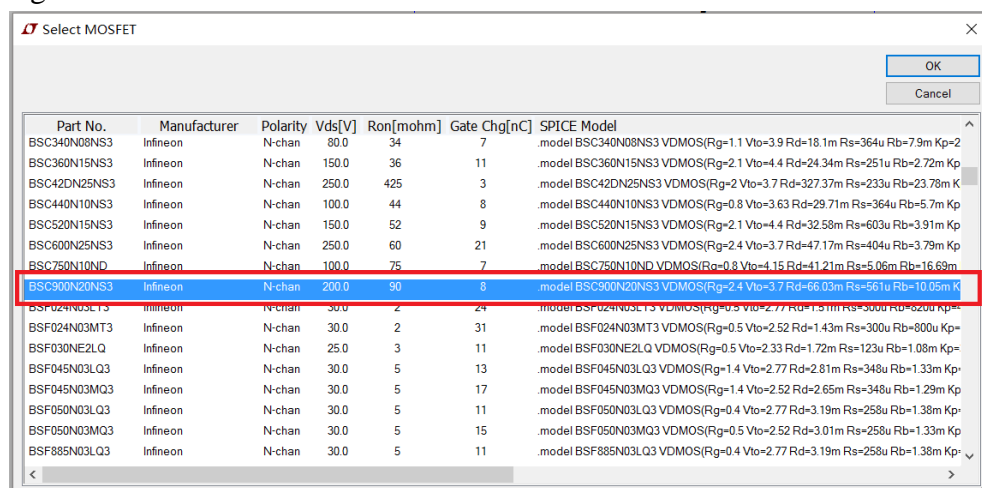
Polarity: nmos

Vds[V]:

Rds(on)[Ω]:

Qgate[C]:

Right-click on MOSFET **M1** and choose *Pick New MOSFET*.



Select MOSFET

Part No.	Manufacturer	Polarity	Vds[V]	Ron[mohm]	Gate Chg[nC]	SPICE Model
BSC340N08NS3	Infineon	N-chan	80.0	34	7	.model BSC340N08NS3 VDMOS(Rg=1.1 Vto=3.9 Rd=18.1m Rs=364u Rb=7.9m Kp=2
BSC360N15NS3	Infineon	N-chan	150.0	36	11	.model BSC360N15NS3 VDMOS(Rg=2.1 Vto=4.4 Rd=24.34m Rs=251u Rb=2.72m Kp=
BSC42DN25NS3	Infineon	N-chan	250.0	425	3	.model BSC42DN25NS3 VDMOS(Rg=2 Vto=3.7 Rd=327.37m Rs=233u Rb=23.78m K
BSC440N10NS3	Infineon	N-chan	100.0	44	8	.model BSC440N10NS3 VDMOS(Rg=0.8 Vto=3.63 Rd=29.71m Rs=364u Rb=5.7m Kp
BSC520N15NS3	Infineon	N-chan	150.0	52	9	.model BSC520N15NS3 VDMOS(Rg=2.1 Vto=4.4 Rd=32.58m Rs=603u Rb=3.91m Kp
BSC600N25NS3	Infineon	N-chan	250.0	60	21	.model BSC600N25NS3 VDMOS(Rg=2.4 Vto=3.7 Rd=47.17m Rs=404u Rb=3.79m Kp
BSC750N10ND	Infineon	N-chan	100.0	75	7	.model BSC750N10ND VDMOS(Rg=0.8 Vto=4.15 Rd=41.21m Rs=5.06m Rb=16.69m
BSC900N20NS3	Infineon	N-chan	200.0	90	6	.model BSC900N20NS3 VDMOS(Rg=2.4 Vto=3.7 Rd=66.03m Rs=561u Rb=10.05m K
BSF024N03LT3	Infineon	N-chan	30.0	2	24	.model BSF024N03LT3 VDMOS(Rg=0.5 Vto=2.77 Rd=1.51m Rs=300u Rb=620u Kp=
BSF024N03MT3	Infineon	N-chan	30.0	2	31	.model BSF024N03MT3 VDMOS(Rg=0.5 Vto=2.52 Rd=1.43m Rs=300u Rb=800u Kp=
BSF030NE2LQ	Infineon	N-chan	25.0	3	11	.model BSF030NE2LQ VDMOS(Rg=0.5 Vto=2.33 Rd=1.72m Rs=123u Rb=1.08m Kp=
BSF045N03LQ3	Infineon	N-chan	30.0	5	13	.model BSF045N03LQ3 VDMOS(Rg=1.4 Vto=2.77 Rd=2.81m Rs=348u Rb=1.33m Kp=
BSF045N03MQ3	Infineon	N-chan	30.0	5	17	.model BSF045N03MQ3 VDMOS(Rg=1.4 Vto=2.52 Rd=2.65m Rs=348u Rb=1.29m Kp=
BSF050N03LQ3	Infineon	N-chan	30.0	5	11	.model BSF050N03LQ3 VDMOS(Rg=0.4 Vto=2.77 Rd=3.19m Rs=258u Rb=1.38m Kp=
BSF050N03MQ3	Infineon	N-chan	30.0	5	15	.model BSF050N03MQ3 VDMOS(Rg=0.5 Vto=2.52 Rd=3.01m Rs=258u Rb=1.33m Kp=
BSF885N03LQ3	Infineon	N-chan	30.0	5	11	.model BSF885N03LQ3 VDMOS(Rg=0.4 Vto=2.77 Rd=3.19m Rs=258u Rb=1.38m Kp=

Then select the **Part No.** to be **BSC900N20NS3** which is going to be the model of our illustration.

- **Voltage Source**

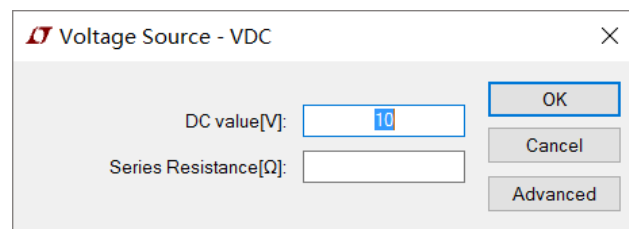
Change the name of the two voltage sources to **VGS** and **VDC** respectively. Detailed configuration of each voltage source will be provided in **Time-domain simulation** and **DC-sweep**.

2. Time-domain simulation

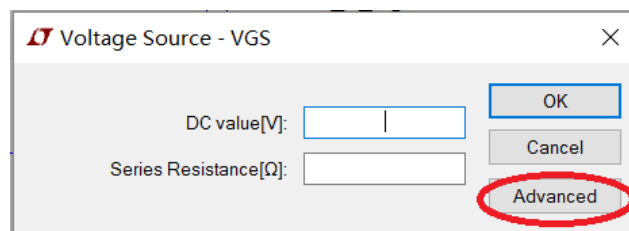
In this part, suppose that VDC is set to 10V and VGS is a square-wave signal enabling to turn on and turn off M1 alternatively. What we want to observe include the instantaneous value of **VDS** (the voltage of M1 across D and S terminal), **Id** (the current flowing through M1) and the power dissipated in M1 which could be calculated as $V_{DS} \cdot I_d$.

- **Voltage Source Configuration**

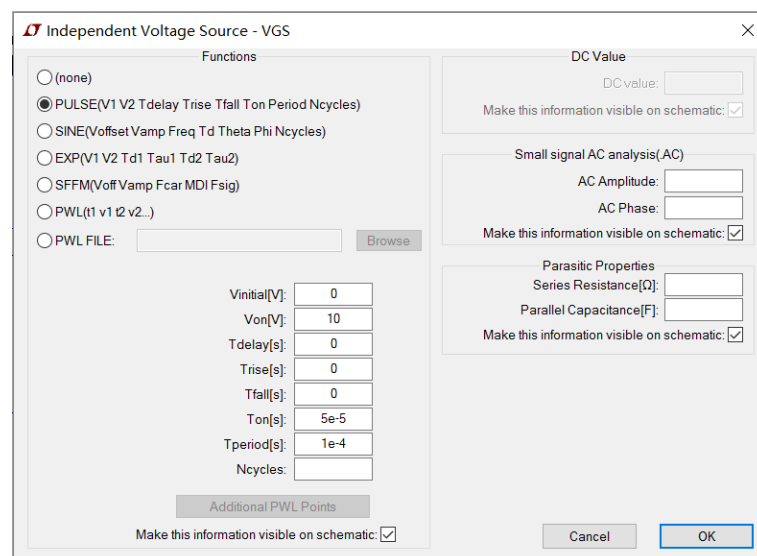
First configure VDC to be 10V.



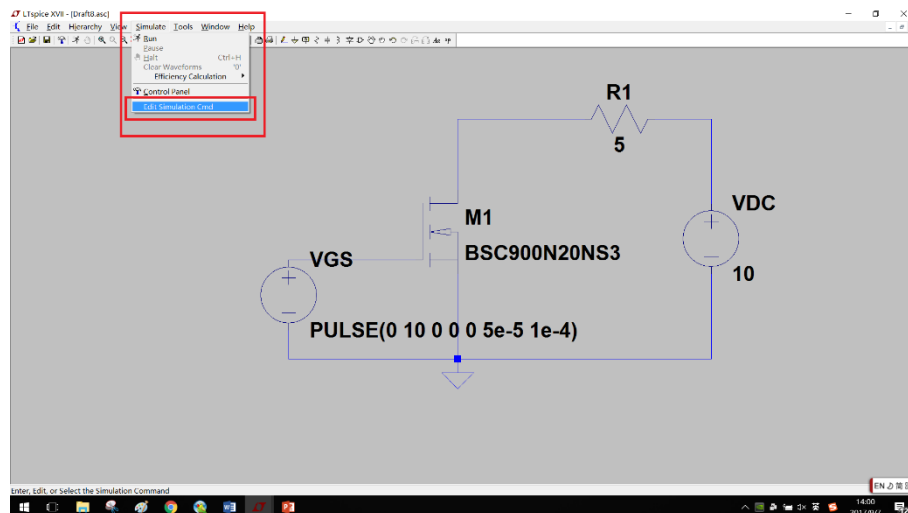
Then configure VGS. Click on **Advanced**.



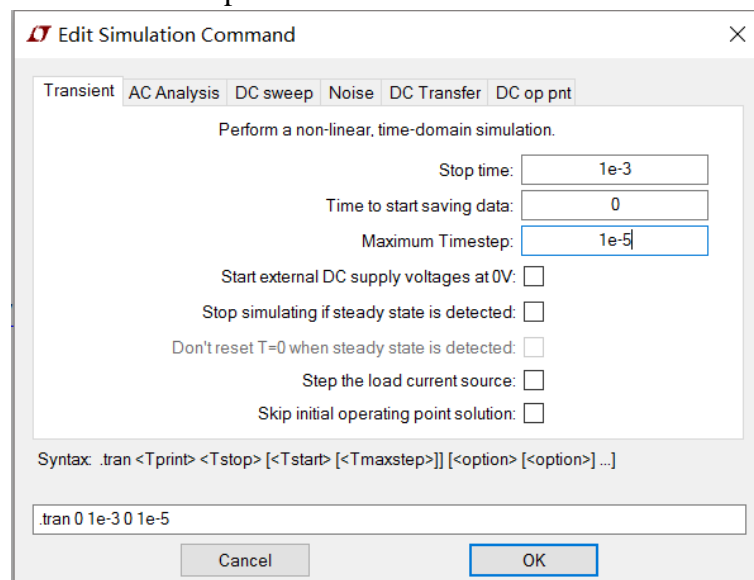
Set VGS to be **PULSE** with a Von of 10V, a frequency of 100us and 50%'s duty cycle.



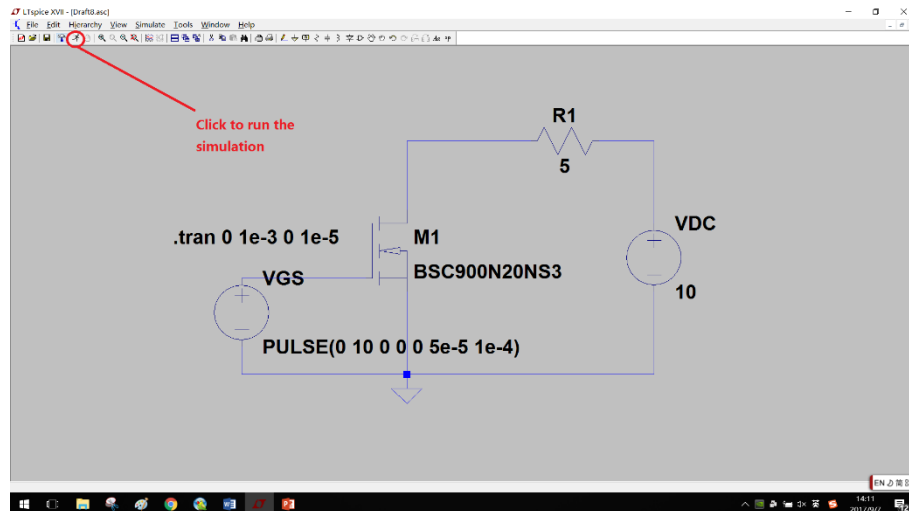
- Simulation command configuration
In the menu *Simulate*, choose *Edit Simulation Cmd.*



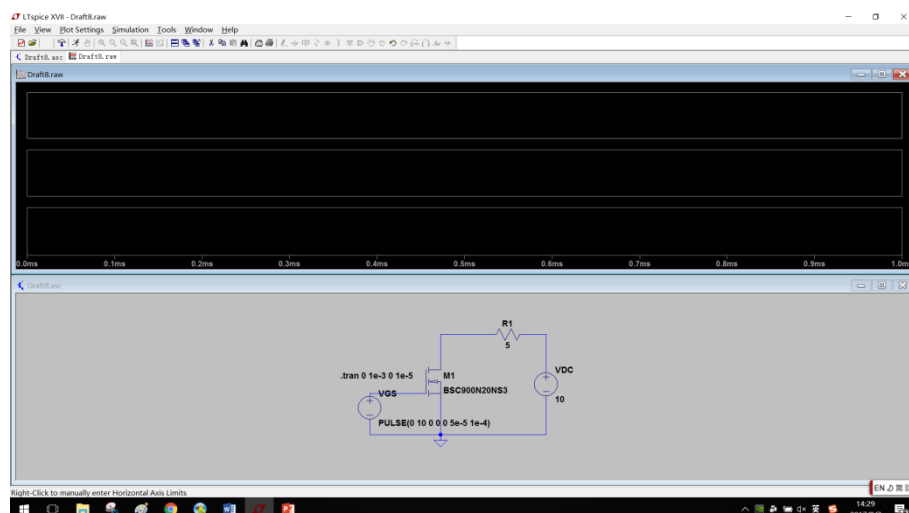
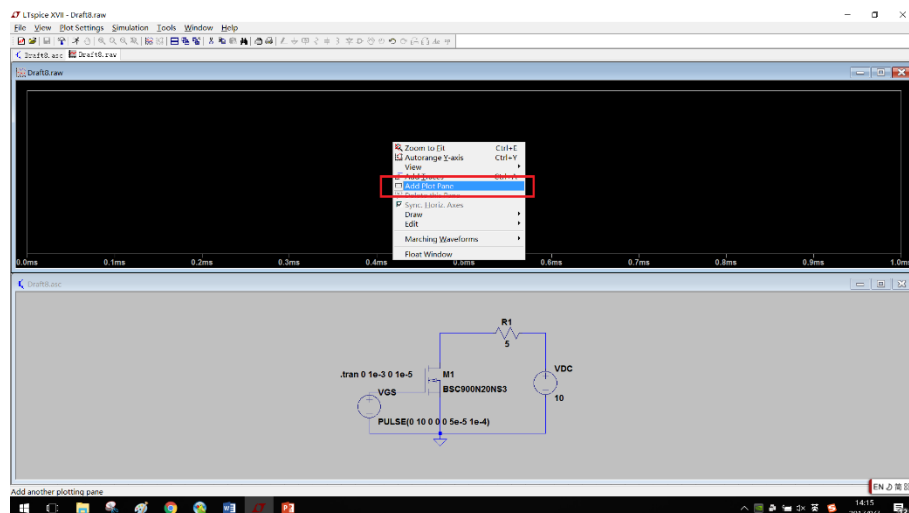
Configure the *Transient* option as below.

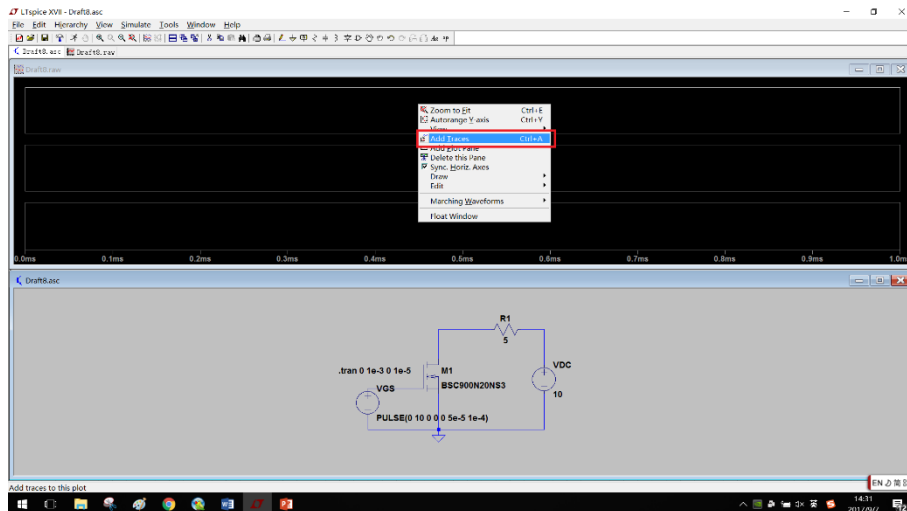


Once these configurations are done, simulation could be conducted. Click on the icon **run** to start the simulation.

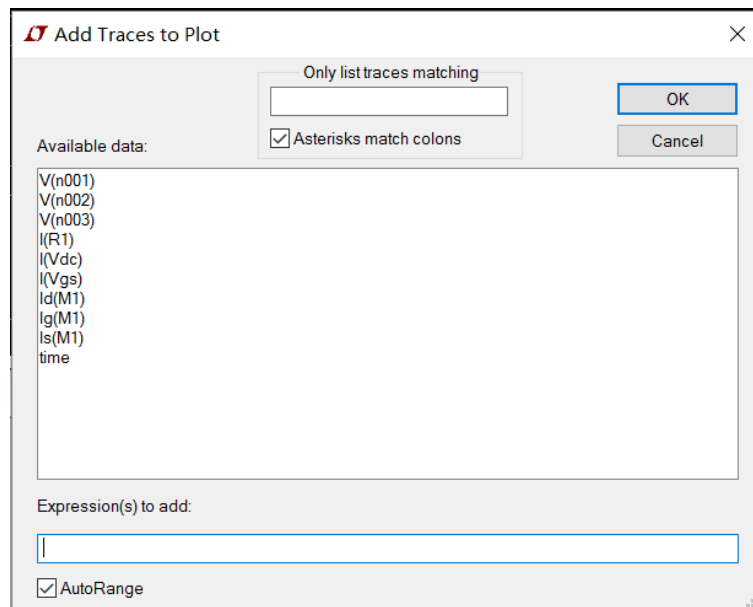


Right-click on the black plot pane and select Add Plot Pane to add two more plot panes.

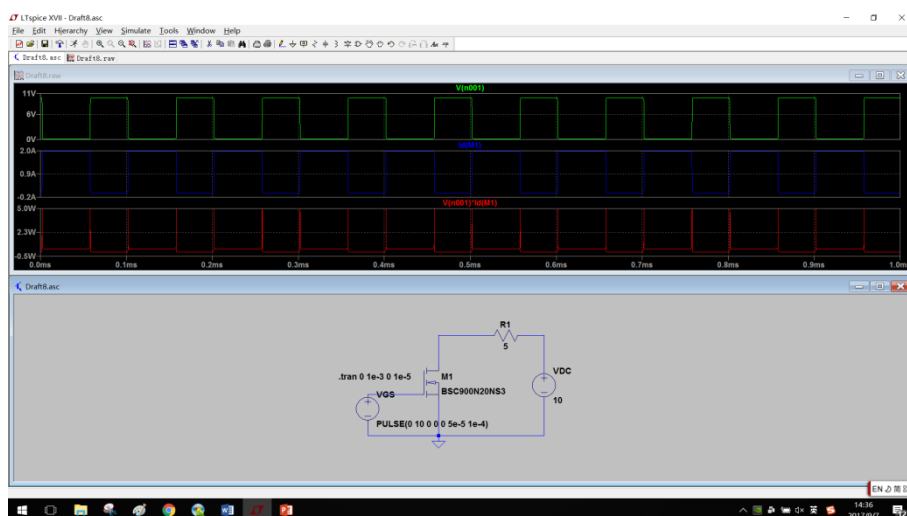




Then in each plot pane, right-click to choose Add Traces. Add V(n001), Id(M1) and V(n001)*Id(M1) to each plot pane respectively.



Eventually, you should be able to see the waveform similar to the one given below.



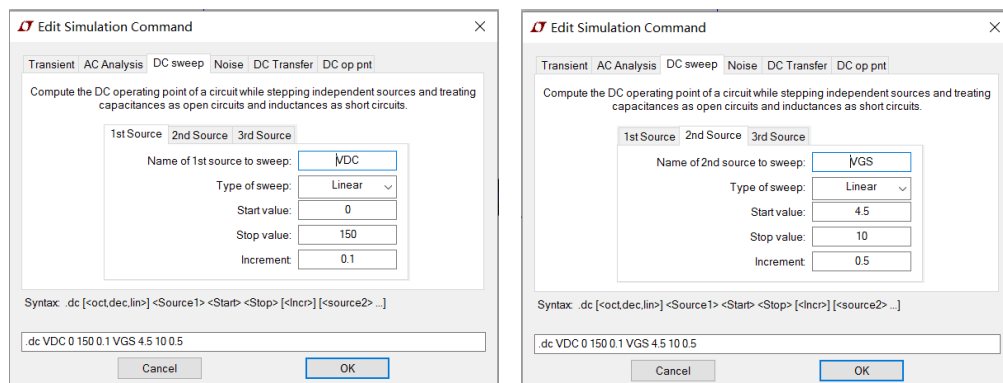
3. DC-sweep

In this part, we aim to study the output characteristics of MOSFET. To be more specific, a study of the relationship between **VDS** (the voltage across the MOSFET) and **Id** (the current flowing through it) under different gate voltage **VGS** should be looked into.

Simulate → **Edit Simulation Cmd** → choose **DC sweep** options

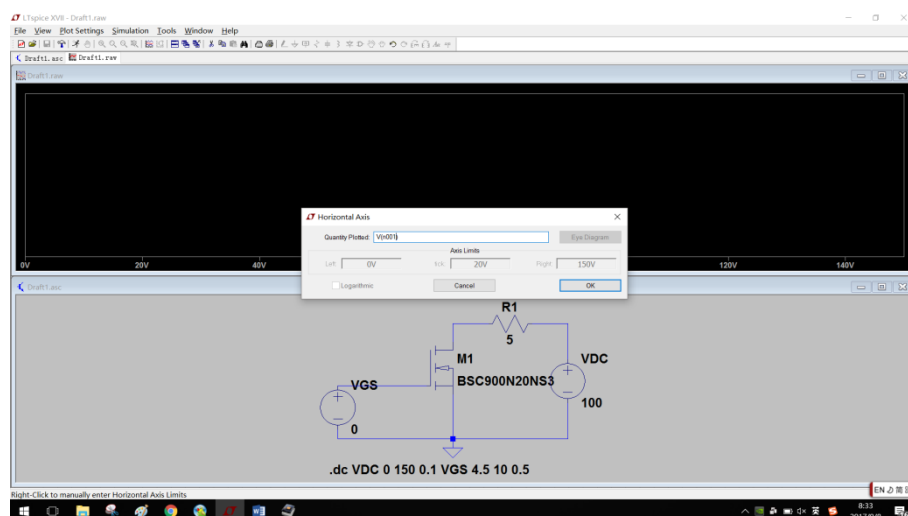
Now we have 2 variables VGS and VDC. Typically, what DC-sweep does is for each VGS, change the value of VDC and get the corresponding Id and plot them in one figure. Thus we have VGS represents test condition while VDC represents excitation.

Configurations are given as follows where VDC changes from 0 to 150V for each VGS and VGS ranges from 4.5V to 10V.

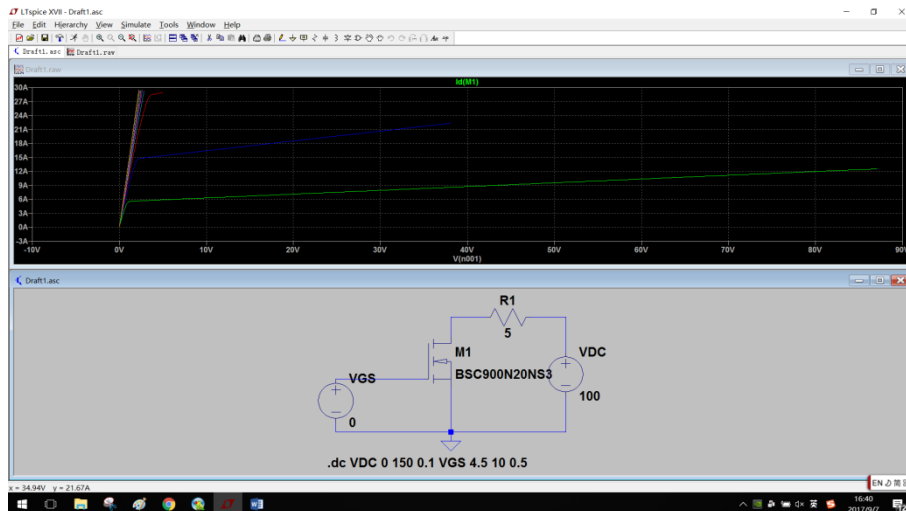


Then run the simulation.

In plot pane, right-click on the bottom to set x-axis as V(n001) which is VDS and then add traces of Id(M1).



The result should be like as below.

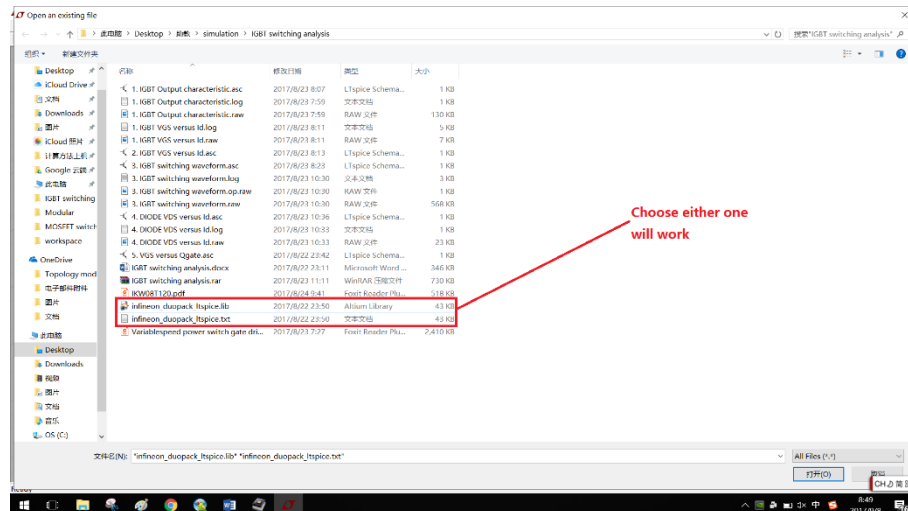


4. Import model

So far you have got all the basic knowledge required to carry out real-time simulation as well as DC sweep to test the characteristics of a MOSFET of certain model. However, there does exist this kind of situation where a specific component is not included in the LTspice library. For example, there is no IGBT model in the library. Thus here a brief tutorial of how to import model from Spice file will be offered.

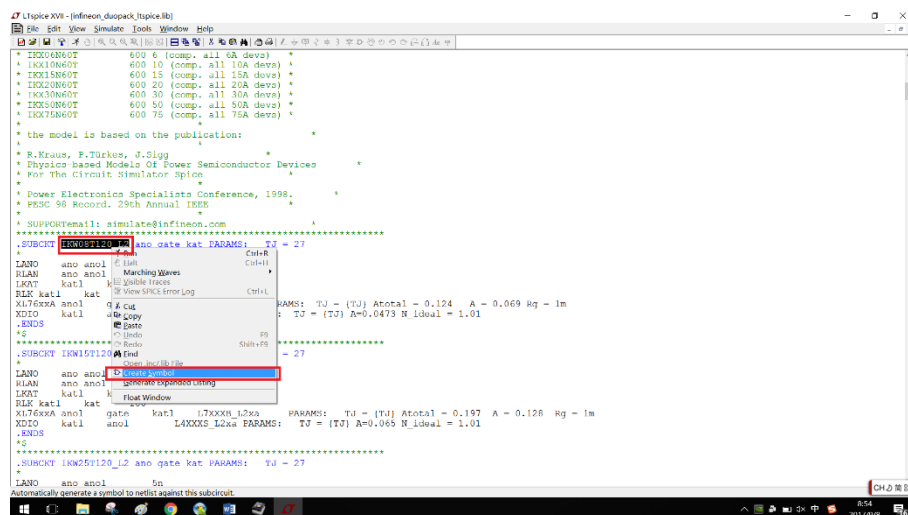


File → Open

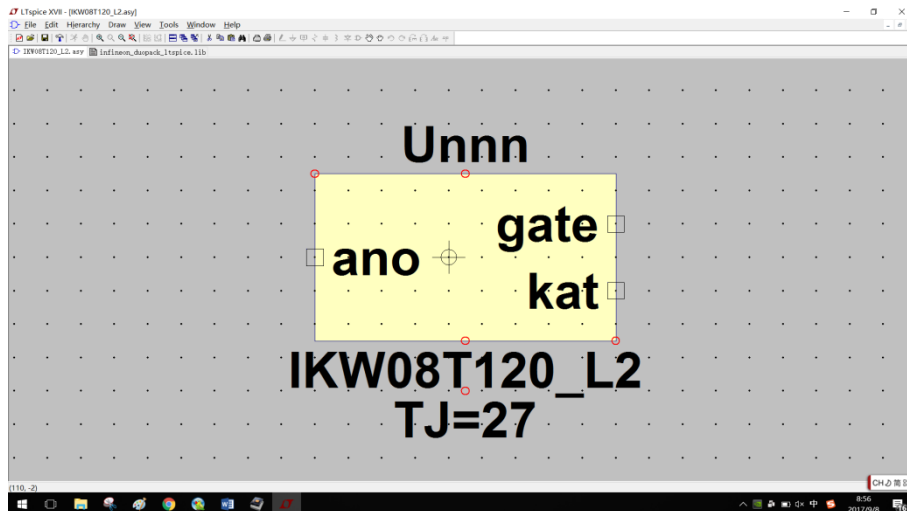


Open the corresponding file in the directory.

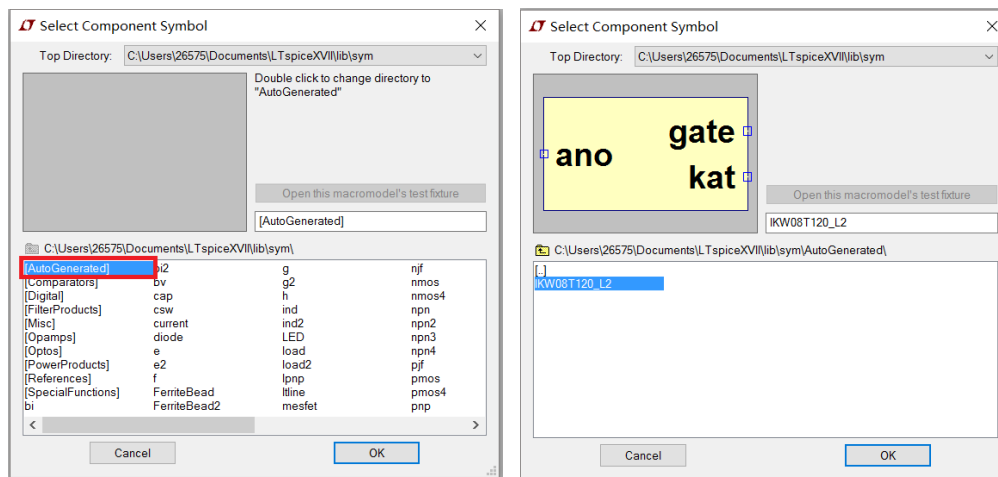
Here we aim to test the characteristics of IKW08T120. In the lib file, select the corresponding model and right-click on it. Click on *Create Symbol*.



Then you create the IGBT of model IKW08T120. Save it and usually it will be in the directory LTspiceXVII\lib\sym\AutoGenerated. To save it in your personal directory would also be possible.



Now create a new schematic. Click on the icon **Component**. In the corresponding directory, you would be able to find your newly-import model.



Then import it to your workspace so that you could carry out your simulation.