

How to use LTSpice

October 13th, 2020

Contents

How to insert third-party components	3
How to run square wave simulation on the circuit (Step response)	6
How to run frequency response simulation on the circuit	9

How to insert third-party components

- 1) Go to Apex analog website to download PA13 op-amp model spice model ([Link](#)).
- 2) Select Apex Power Operational Amplifiers zip file as shown in the picture below.

The screenshot shows the Apex Microtechnology website. At the top, there are links for "COVID-19 Updates" and "Apex Distribution Partners". Below these is a navigation bar with links for "HOME", "PRODUCTS", "APPLICATIONS", "SUPPORT", "SALES", "QUALITY", "COMPANY", and "SEARCH". The main content area is titled "SPICE Models" and contains a section for "SPICE Models – Power Operational Amplifiers". This section lists three download links: "Apex Power Operational Amplifiers (ZIP, 150 KB)", "Intusoft Symbols (ZIP, 9 KB)", and "TosSPICE Symbols (ZIP, 4 KB)". A red arrow points to the first link. On the right side of the page, there is a "Contacts" section with information about the company's location and contact details.

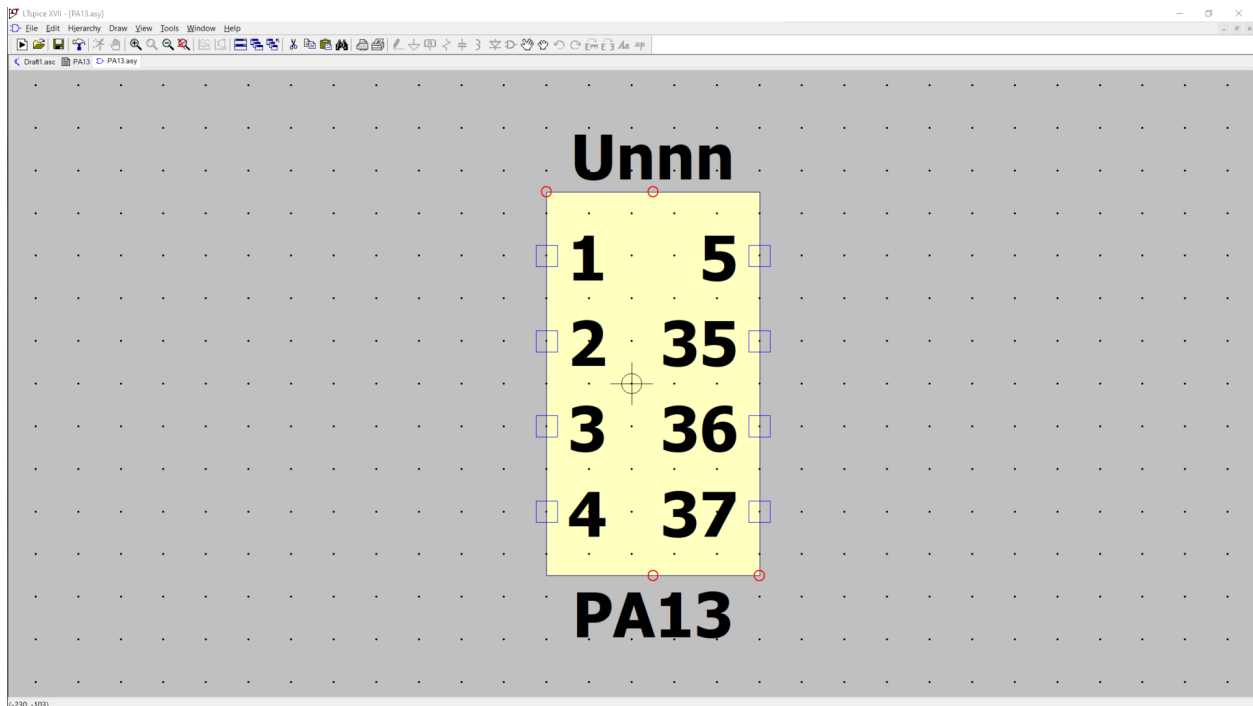
- 3) Unzip apexpoweropamps.zip file at any file location of your choice and look for **PA13.lib** file inside the apexpoweropamps unzipped folder.
- 4) Copy and paste **PA13.lib** file in the same file directory as your LTSpice saved file (.asc type).
- 5) Open **PA13.lib** file in your LTSpice by drag and drop the file or go to File -> Open. After inserted the file, you should get the same code as below on your LTSpice.

The screenshot shows the LTSpice XVII interface with the PA13.lib file open. The file content is displayed in a text editor window, showing the model parameters and pinout for the PA13 op-amp. The code includes a revision number, a description of the model, and a list of pinout parameters. The pinout parameters are listed in a table format, with columns for pin number, pin name, and pin value. The pinout parameters are as follows:

Pin	Pin Name	Pin Value
1	Q1	10 1 6 Q11
2	Q2	11 2 9 Q12
3	R3	12 8 2.69E+03
4	R4	12 9 2.69E+03
5	I2	12 5 4.00E-05
6	CL	12 5 1.43E-12
7	R5	12 5 2.50E+07
8	R1	4 10 3.99E+03
9	R2	4 11 3.99E+03
10	C2	10 34 15E-12
11	R2	34 11 40
12	I1	4 5 8.56E-03
13	G1	6 15 11 10 2.51E-04
14	G2	6 15 12 15 1.41E-09
15	R6	6 15 1.00E+05
16	D1	6 15 DD
17	D2	15 6 DD
18	C3	6 7 1.00E-11
19	G3	15 7 15 6 7.08E+00
20	R7	7 15 1E3
21	D3	7 16 DD
22	V1	18 16 2.70E+00
23	D4	17 7 DD
24	V2	17 19 4.50E+00
25	RE1	15 0 0.001
26	E2	18 0 4 0 1
27	E3	19 0 5 0 1
28	R8	7 20 50
29	C4	20 15 1E-12
30	Q3	19 20 21 QOP
31	Q4	18 20 22 QON
32	Q5	4 23 29 QON
33	Q6	5 24 30 QOP
34	Q7	25 27 3 Q1NA
35	Q8	26 28 3 Q1PA
36	R11	21 23 3.97E-02
37	RF1	27 37 20E3
38	RF2	28 37 20E3
39	R13	22 24 3.97E-02
40	D5	23 25 DL
41	D6	26 24 DL
42	R9	27 35 280
43	R10	28 36 280
44	I3	18 23 1.64E-02
45	I4	24 19 1.64E-02
46	RO1	29 35 0.134
47	RO2	30 36 0.134
48	RF11	37 35 10E6
49	RF22	37 36 10E6

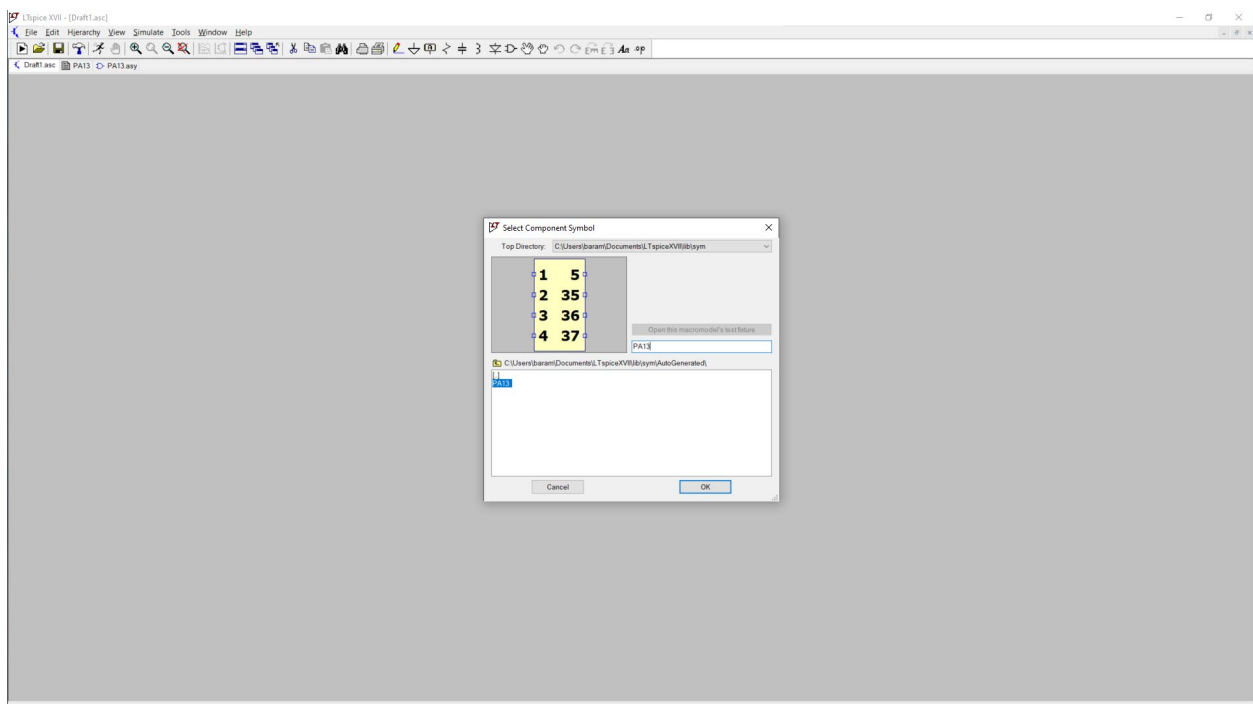
6) Right click on the blue letter in the code that start with .SUBCKT then select Create symbol.

7) LTSpice should be able to autogenerate PA13 component for you.

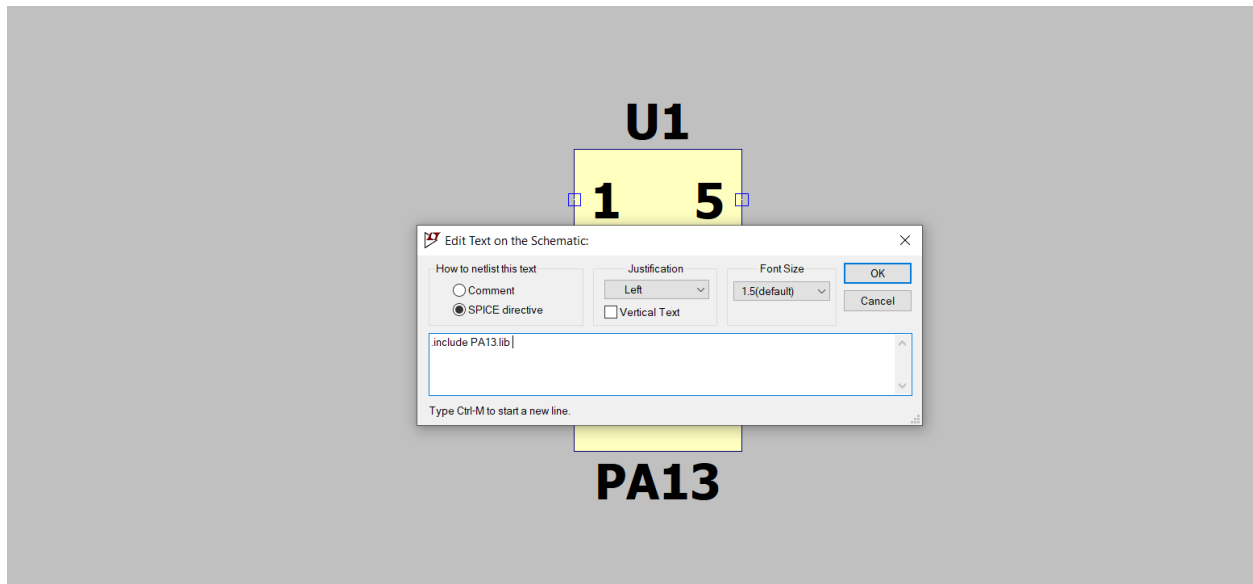


Note: To find out what these pins number represent. Go back to PA13.lib file and look at the top section of the file.

8) You should be able to insert PA13 component into your schematic at this point



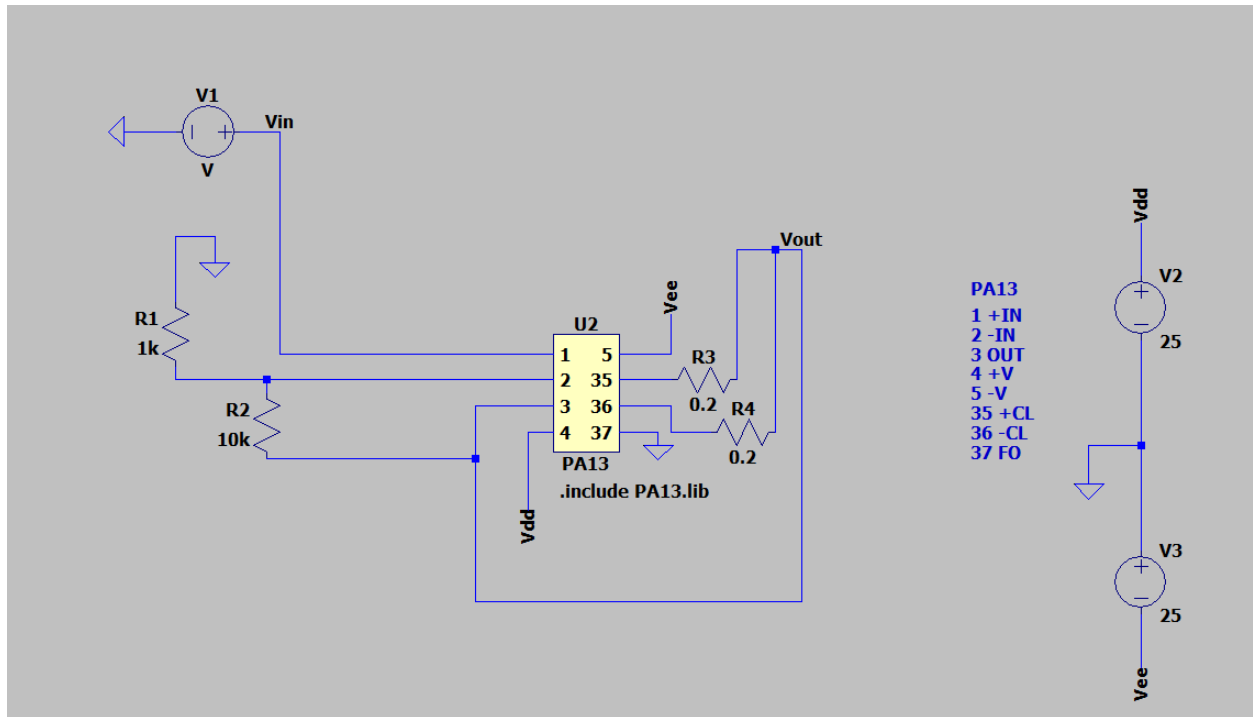
9) Insert PA13 op-amp simulation file into LTSpice by go to Edit -> Spice Directive. Then type **.include PA13.lib**



Note: Make sure that you have PA13.lib file in the same folder as your LTSpice file.

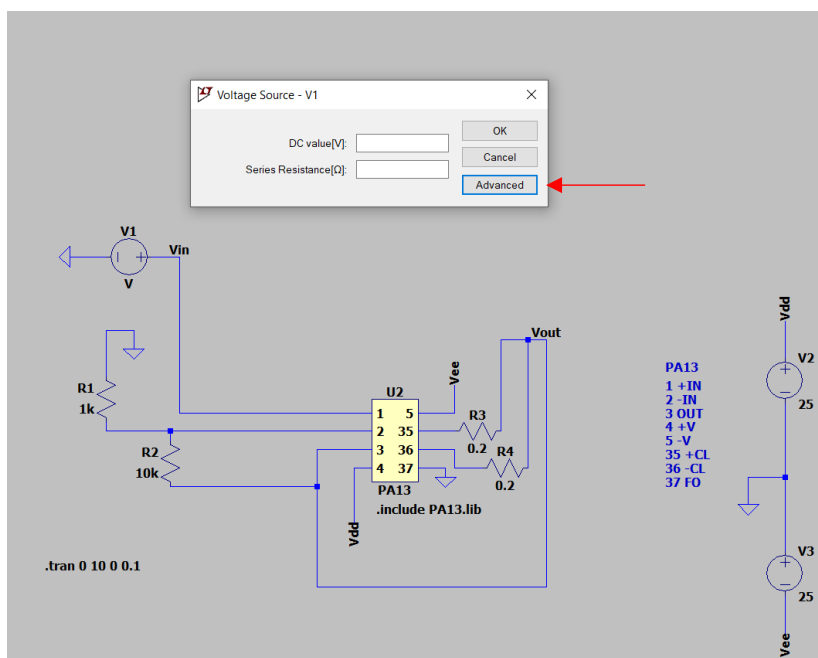
How to run square wave simulation on the circuit (Step response)

1) Assemble a circuit on LTSpice as shown in the picture below

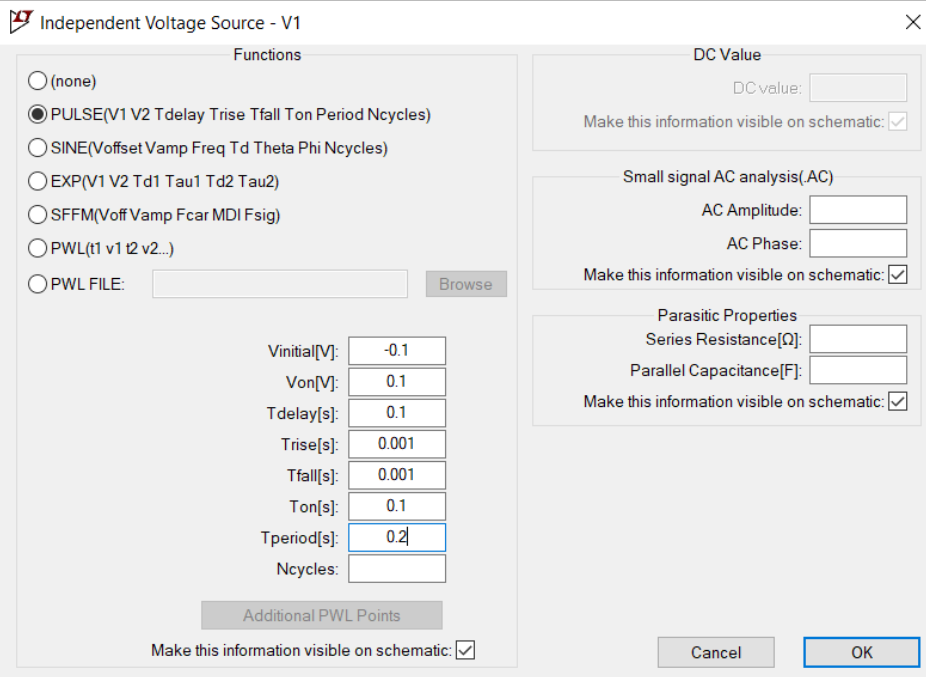


Note: The PA13 table on the schematic is for pin labeling only. It's optional to the circuit

2) Right click on voltage source 1 and select Advanced.



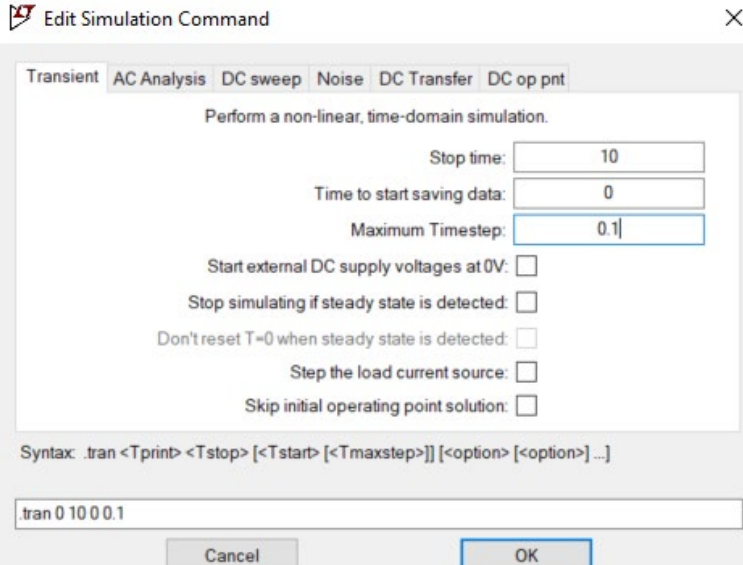
3) In the new popped up window, input the following values in the picture below for 10Hz square wave.



The dialog box 'Independent Voltage Source - V1' contains the following settings:

- Functions:**
 - ☐ (none)
 - ☒ PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)
 - ☐ SINE(Voffset Vamp Freq Td Theta Phi Ncycles)
 - ☐ EXP(V1 V2 Td1 Tau1 Td2 Tau2)
 - ☐ SFFM(Voff Vamp Fcar MDI Fsig)
 - ☐ PWL(t1 v1 t2 v2...)
 - ☐ PWL FILE:
- Initial Conditions:**
 - Vinitial[V]: -0.1
 - Von[V]: 0.1
 - Tdelay[s]: 0.1
 - Trise[s]: 0.001
 - Tfall[s]: 0.001
 - Ton[s]: 0.1
 - Tperiod[s]: 0.2
 - Ncycles:
- Additional PWL Points:**
- Make this information visible on schematic:** ☒
- DC Value:**
 - DC value:
 - Make this information visible on schematic:** ☒
- Small signal AC analysis (AC):**
 - AC Amplitude:
 - AC Phase:
 - Make this information visible on schematic:** ☒
- Parasitic Properties:**
 - Series Resistance[Ω]:
 - Parallel Capacitance[F]:
 - Make this information visible on schematic:** ☒
- Buttons:**

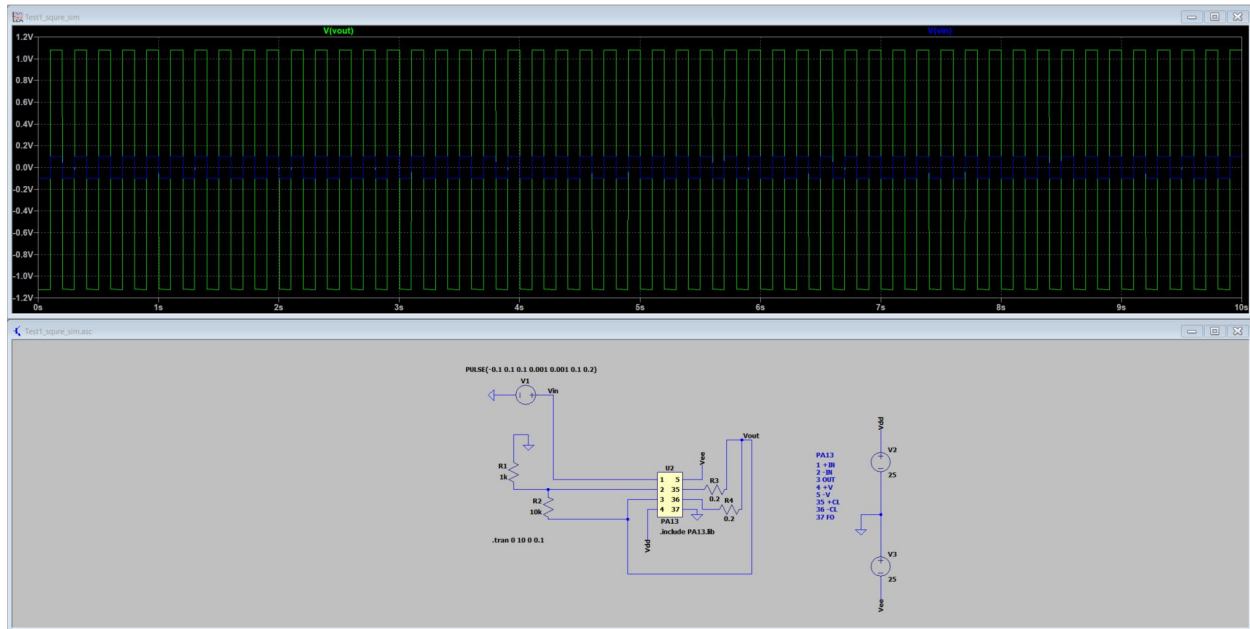
4) Run simulation by go to Simulate -> Run and input simulation detail as shown below



The 'Edit Simulation Command' dialog box shows the following configuration for a transient simulation:

- Simulation Type:** Transient (selected)
- Perform a non-linear, time-domain simulation.**
- Stop time:** 10
- Time to start saving data:** 0
- Maximum Timestep:** 0.1
- Start external DC supply voltages at 0V:** ☐
- Stop simulating if steady state is detected:** ☐
- Don't reset T=0 when steady state is detected:** ☐
- Step the load current source:** ☐
- Skip initial operating point solution:** ☐
- Syntax:** .tran <Tprint> <Tstop> [<Tstart> [<Tmaxstep>]] [<option> [<option>] ...]
- Command Line:** .tran 0 10 0 0.1
- Buttons:**

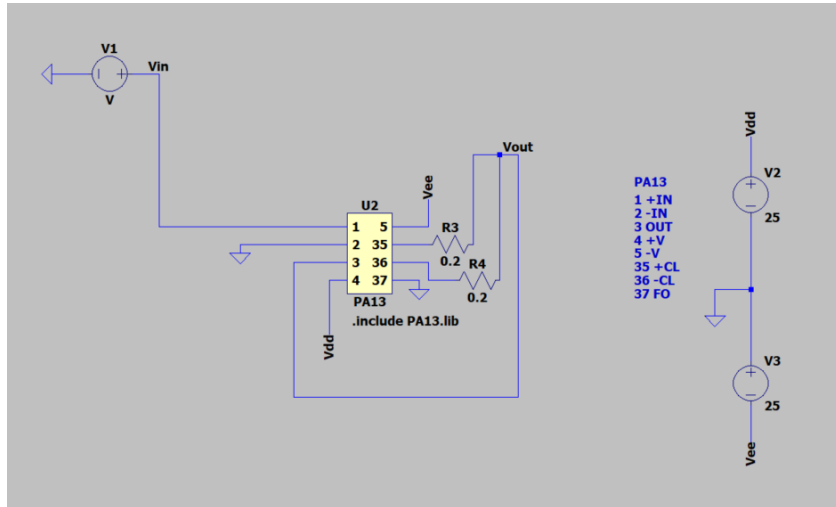
5) Put a probe symbol cursor on both V_{in} and V_{out} to read the simulated output. If done correctly, your LTSpice should look like picture shown below.



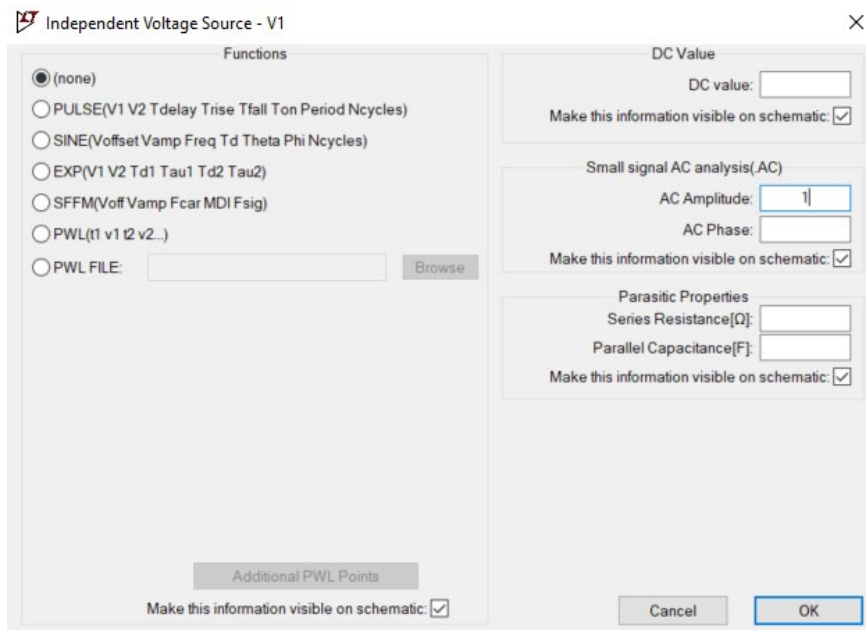
6) Zoom into the rising edge of the waveform and take the reading according to the lab manual

How to run frequency response simulation on the circuit (Bode plot)

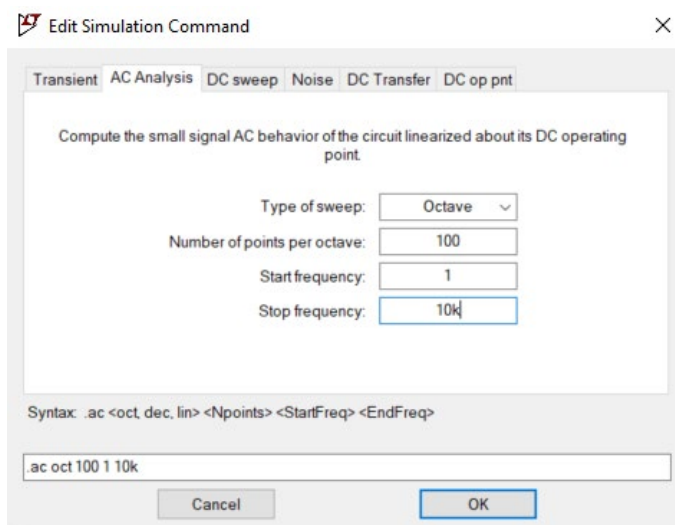
1) Create a schematic as shown in the picture below.



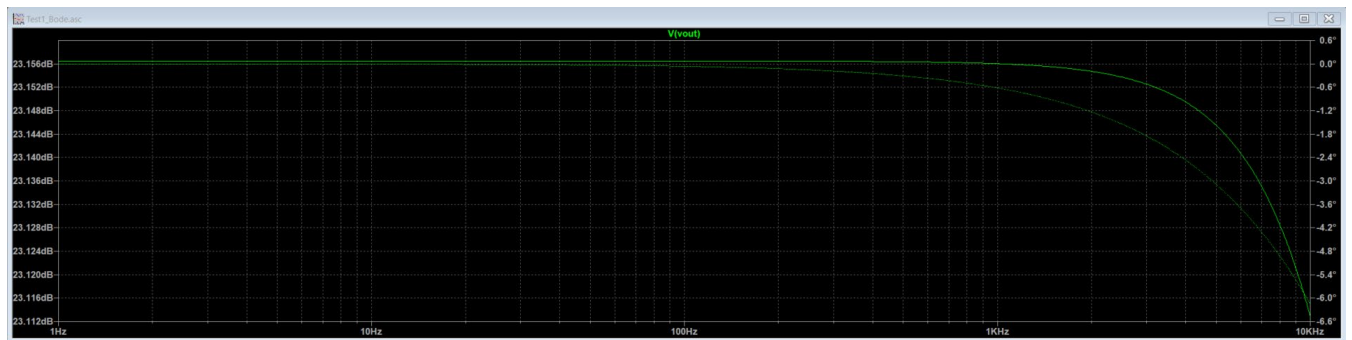
2) Right click on voltage source and select advanced setting. Set AC amplitude to 1 as shown in the picture below.



3) Go to Simulate -> Run -> AC Analysis and input the following setting.



4) Press run and put a probe cursor on the Vout or output of the Op-amp. You should get the following frequency response.



Note: The solid green line is the gain and the dotted line is the phase.