

EE698W

Department of Electrical Engineering, IIT Kanpur

Tutorial on LTspice

1. Adding library file to LTspice editor:

- Open LTspice and click on `.op` (right top corner) and type `.lib` and then click OK.
- Next right click on `.lib` two times and then browse for the model file saved in your machine (PC)

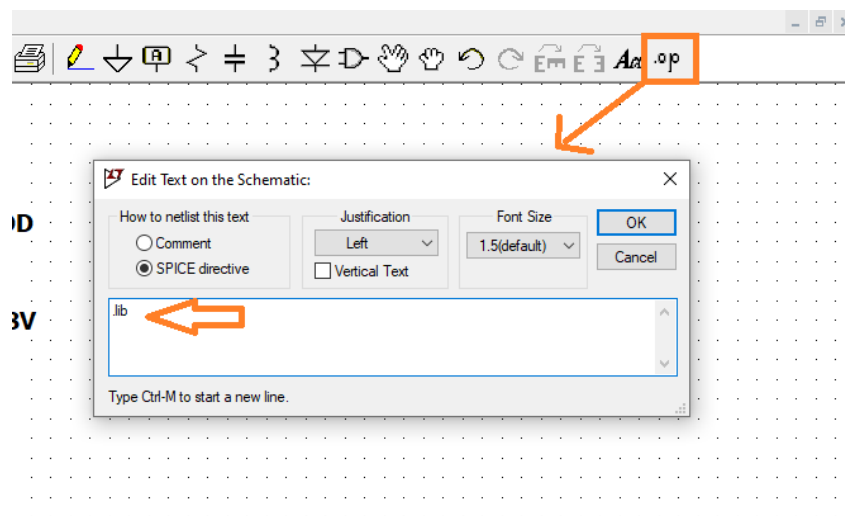


Fig: 1. Opening Command Prompt in LTspice

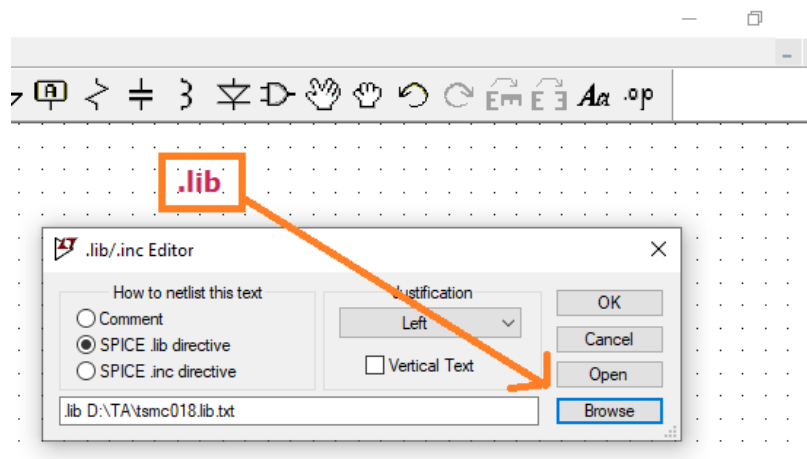


Fig 2. Adding Library File or Model file to Editor

2. Changing Transistor Model Name:

- Select any transistor from component list (lets say `nmos4`)
- Ctrl + Right click on transistor
- Then change NMOS to model name of transistor which was added earlier.
- Open model file and see name of transistor. Here its CMOSN. Change NMOS to CMOSN as shown in Fig 3.

- e) Cross mark (X) will give you an option to visible that parameter along with the instance in the editor

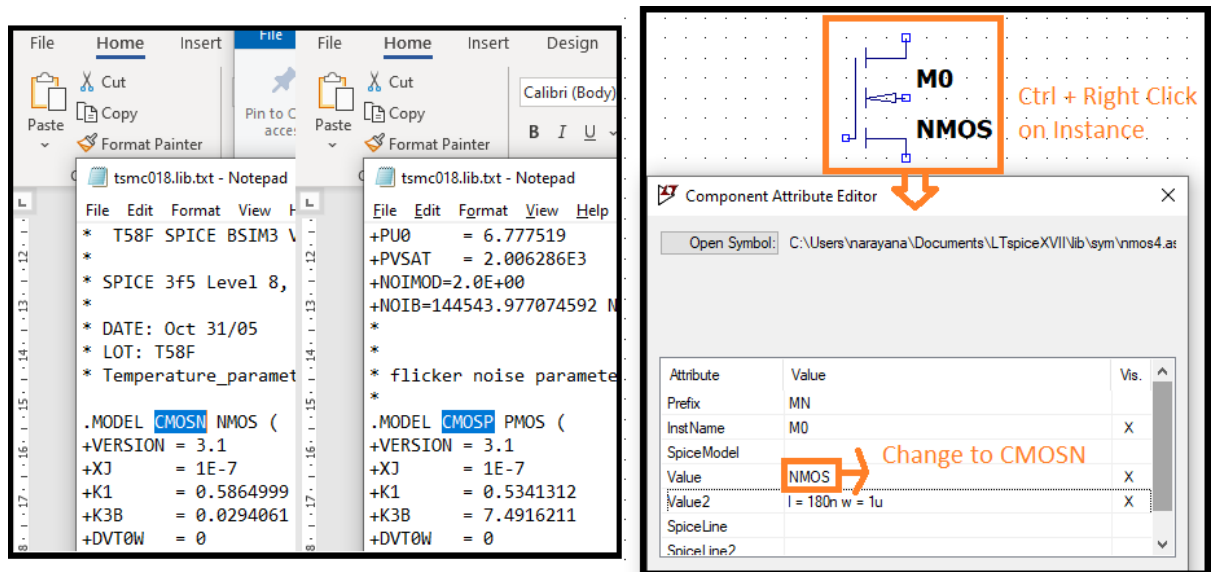


Fig 3. Changing Model name of Transistor

3. DC Operating Point Analysis:

- To perform DC operating point analysis, go to Edit and click on Spice Analysis. Select "dc op pnt" and type .op (or Simply click on .op (right top corner) and type .op)
- Next go to Simulate and click on Run
- To see the DC operating points, go to View and click on SPICE Error Log (Shortcut: Ctrl + L)

SPICE Error Log: D:\TA\id_vs_vds.log

Vds:	1.000e+00
Vbs:	0.00e+00
Vth:	4.89e-01
Vdsat:	1.75e-01
Gm:	4.16e-03
Gds:	1.24e-04
Gmb:	1.02e-03
Cbd:	0.00e+00
Cbs:	0.00e+00
Cgsov:	8.23e-15
Cgdov:	8.23e-15
Cgbov:	1.46e-19
dQgdVgb:	2.71e-14
dQgdVdb:	-8.18e-15
dQgdVsb:	-1.80e-14
dQddVgb:	-1.23e-14
dQddVdb:	8.20e-15
dQddVsb:	5.26e-15
dQbdVgb:	-2.40e-15
dQbdVdb:	3.75e-18
dQbdVsb:	-7.08e-16

Arrows point from the following capacitor definitions to the corresponding values in the log:

- C_{gg}
- C_{gd}
- C_{gs}
- C_{dg}
- C_{dd}
- C_{ds}
- C_{bg}

Fig 4. DC operating point of Transistor and different capacitor definitions

This capacitor definitions can get from Operation and Modelling of MOS Transistor by Tsividis, 3e - Page No: 416.

4. Performing Parametric Analysis:

- First define a variable on which you would like to do parametric analysis.
- Let's say I would like to vary width of the transistor and define that as $w = w1$
- Add command `.step param w1 1u 10u 2u`
- It means $w1$ is varying from $1\mu\text{m}$ to $10\mu\text{m}$ with a step of $2\mu\text{m}$

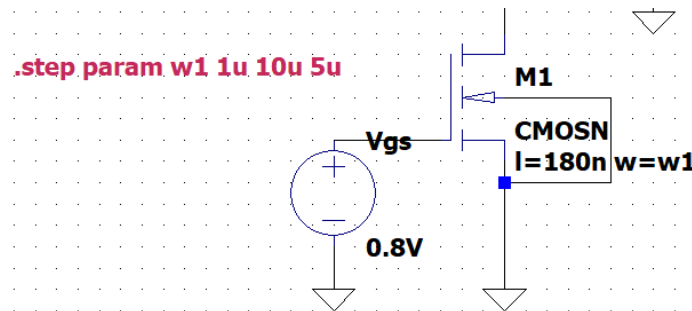


Fig 5. Example of Parametric analysis command

5. Input and output characteristic of NMOS:

To obtain **input characteristics** connect the circuit as shown in Fig 6.

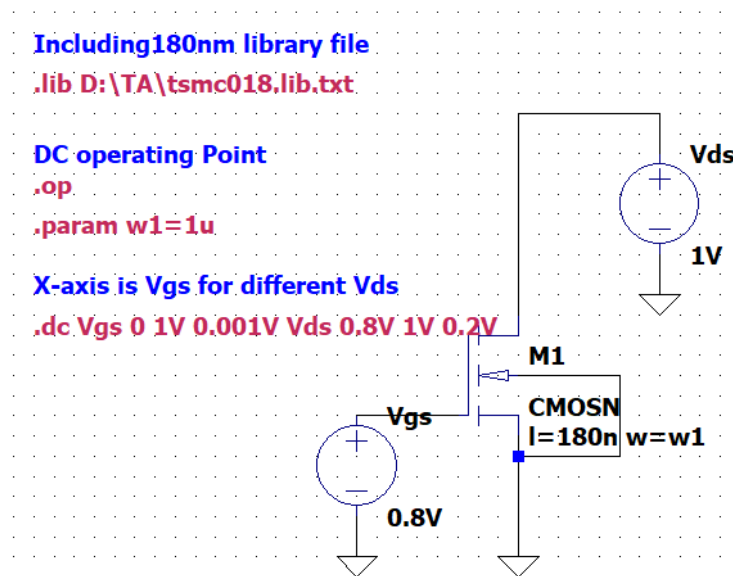


Fig 6. Circuit diagram to get input characteristics

- Go to Edit and click on Spice analysis
- Go to DC Sweep and enter values of V_{gs} in 1st Source
- To get i_d vs V_{gs} plot for different values of V_{ds} , enter values of V_{ds} in 2nd Source.

`.dc Vgs 0 1V 0.001V Vds 0.8V 1V 0.2V`

- Then Run simulation

- e) Select Current of transistor (place the cursor at drain of transistor until you get black colour probe symbol. It indicates current. If its simple red pen type symbol then it indicates node voltage)

How to get gm plot from I_D vs V_{gs} plot?

- a) Got to Plot Settings and click on Add trace (Shortcut: Ctrl + A)
 b) In expression field write $d(I_D(M1))$ and click OK.
 c) $d()$ indicates that differentiating a parameter with respect to X-axis, since X-axis is V_{gs} . $\Rightarrow \frac{dI_D}{dV_{gs}} = g_m$

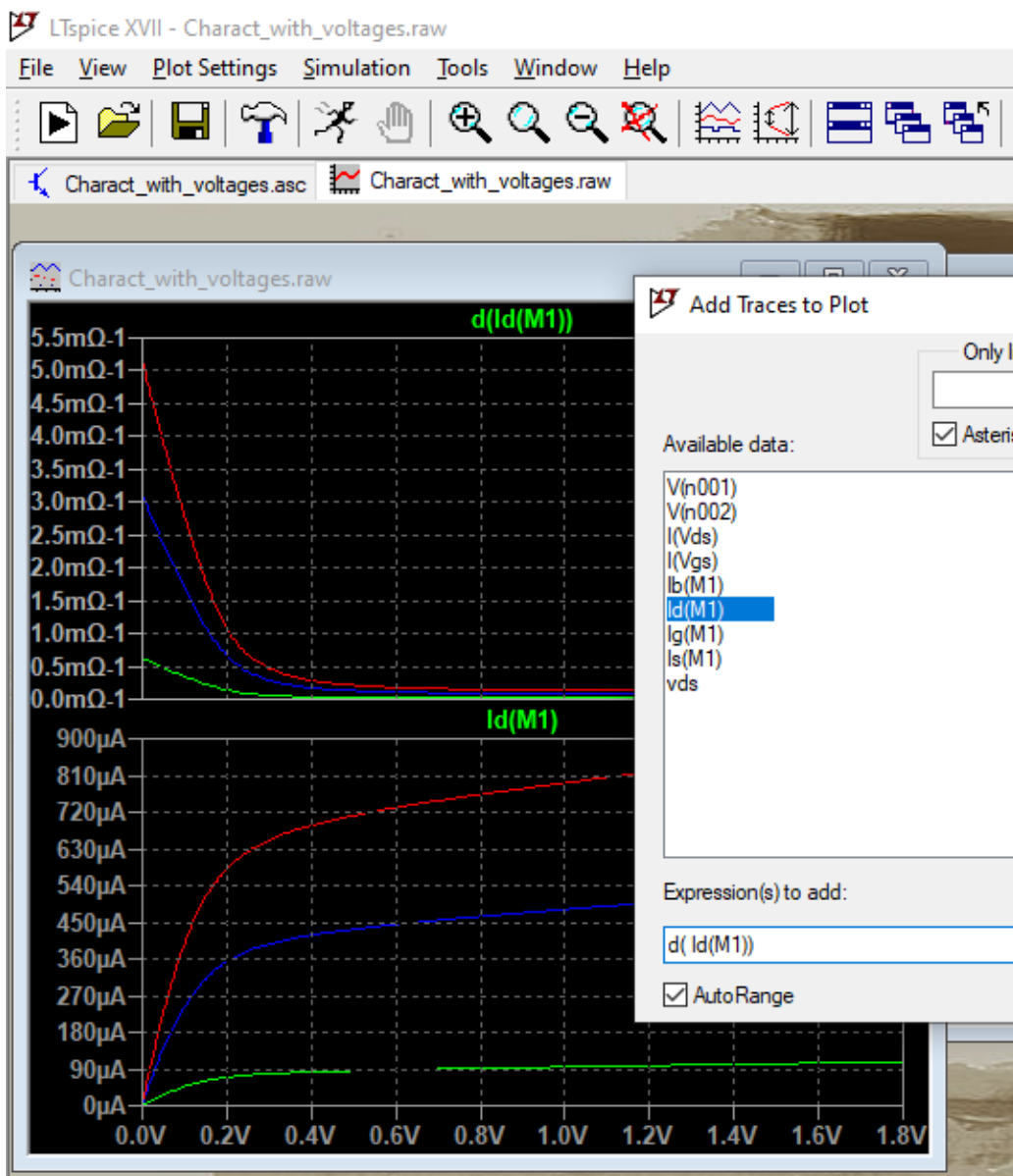


Fig 7. Plotting g_m from I_D vs V_{gs} plot

To obtain **output** characteristics of Fig 6, add command as follows

- Go to Edit and click on Spice analysis
- Go to DC Sweep and enter values of V_{ds} in 1st Source
- To get i_d vs V_{ds} plot for different values of V_{gs} , enter values of V_{gs} in 2nd Source.

```
.dc Vds 0 1V 0.001V Vgs 0.6V 0.8V 0.1V
```

- Then Run simulation and Select Current of transistor
- gds plot can be obtained from I_D vs V_{gs} by doing as follow
 - Press Ctrl+A and write $d(I_D(M1))$
 - Then you will get plot i_d vs V_{ds}

6. Obtaining gm vs Current density plot:

Connect the circuit and enter the commands as shown in Fig 8

- Command `.param` is used to assign value to defined variable.
- Here, sweeping parameter is I_0 ($I_0/w_1 =$ Current density) and it will come in X-axis.

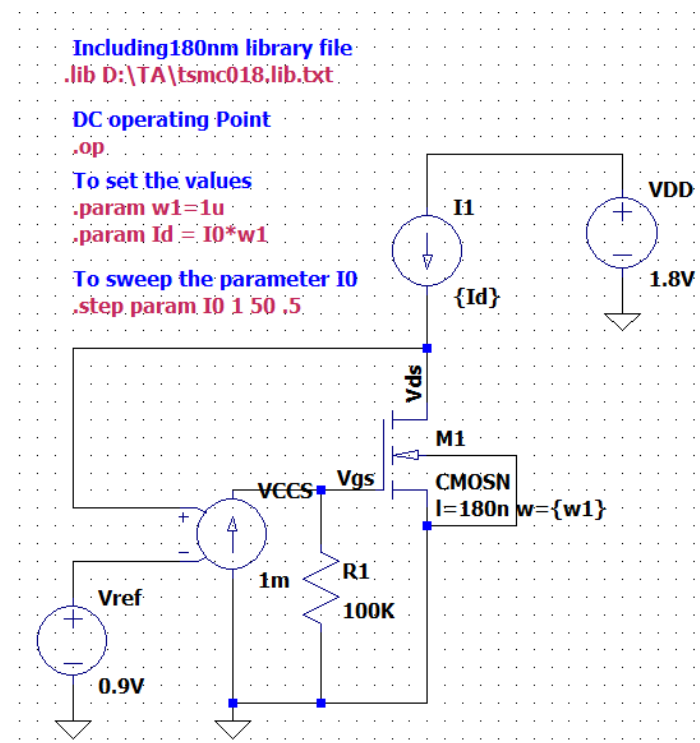


Fig 8. Circuit diagram to characterise NMOS

- Run the simulation
- Press Ctrl + A and write $d(I_D(M1)) / d(V(vgs))$ in expression field. Then you will get gm vs Current density.
- Suppose if you write simply $d(I_D(M1))$ in expression field, that means you are differentiating I_d of M1 transistor with respect to Current density.

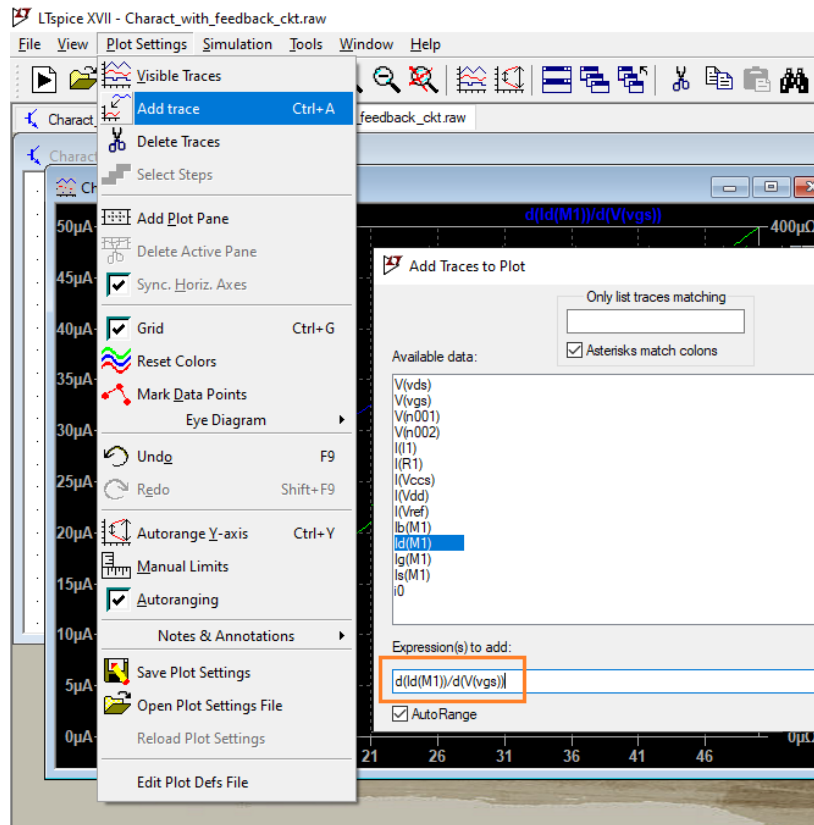
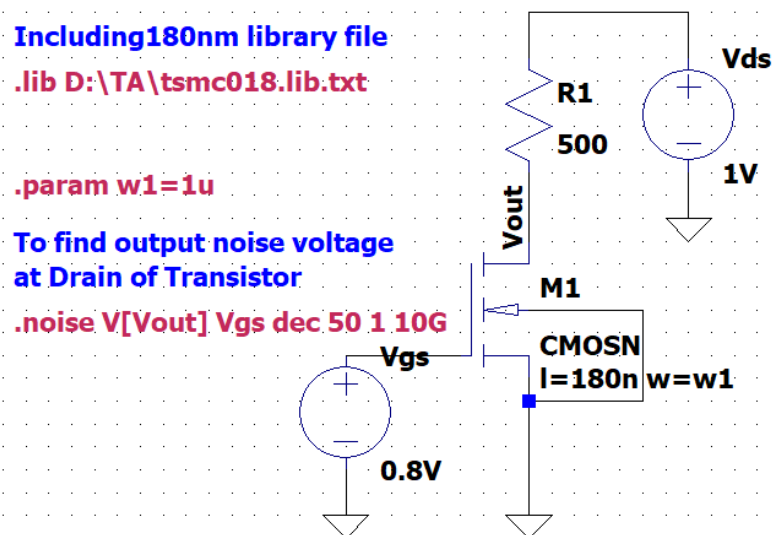


Fig 9. Plotting g_m with respect to current density

7. Noise Analysis:

Output Noise voltage of transistor can be plotted as follows



- Go to Edit and click on Spice analysis
- Go to Noise and write V[Vout] in Output field. Here, at the drain of transistor we would like to find noise voltage. Node voltage should keep inside the V[]
- Enter input voltage, since Vgs
- Select type of sweep and enter frequency range (let 1Hz to 10G with 50 per decade)
- The overall command will be as follows

.noise V[Vout] Vgs dec 50 1 10G

- f) Run the Simulation and select V_{out} the
- g) Export the plot to text file and plot it in Matlab or simply right click on Y-axis of the plot and change it to logarithmic.

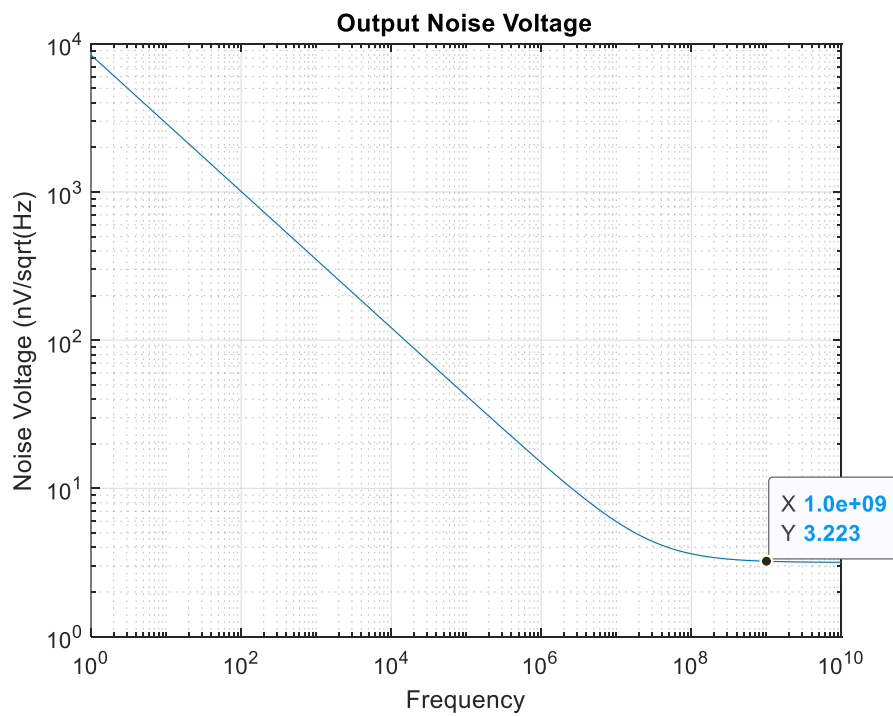


Fig 10. Output noise voltage at drain of transistor.

Note:

- You can export the plot to text file and then can plot it in Matlab. Refer Fig 11
- Select the parameters which you want to export

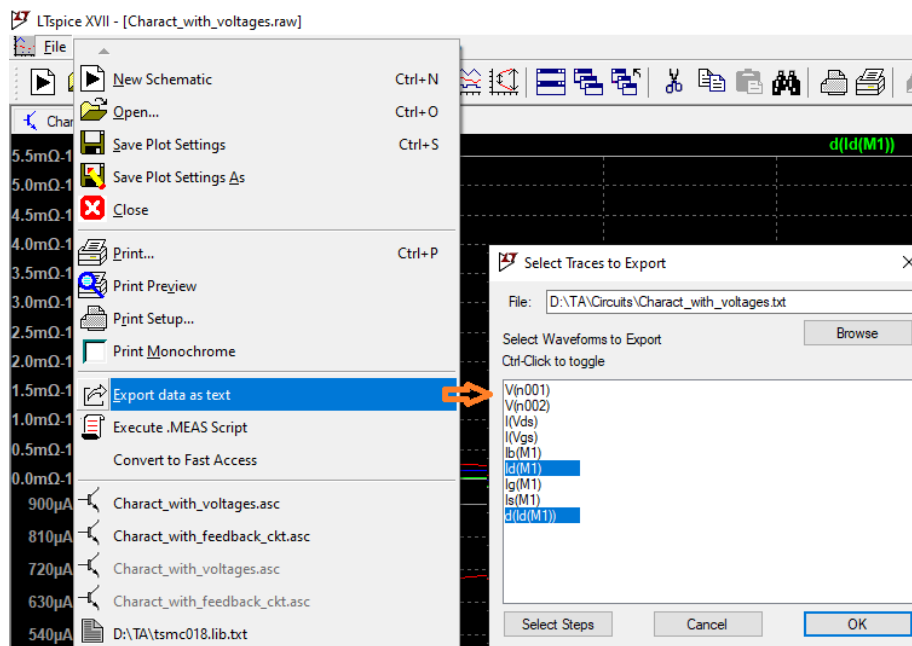


Fig 11. Exporting different plots to text file