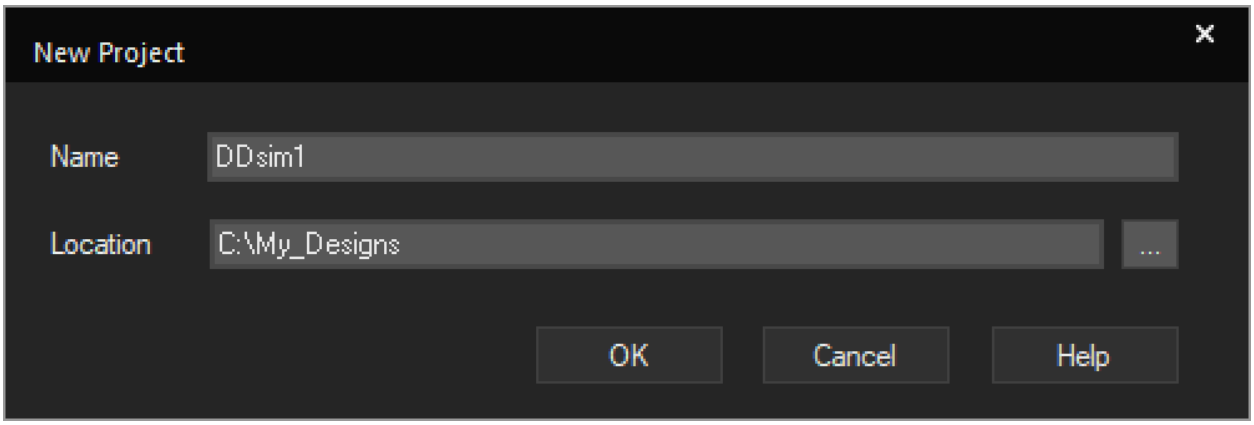
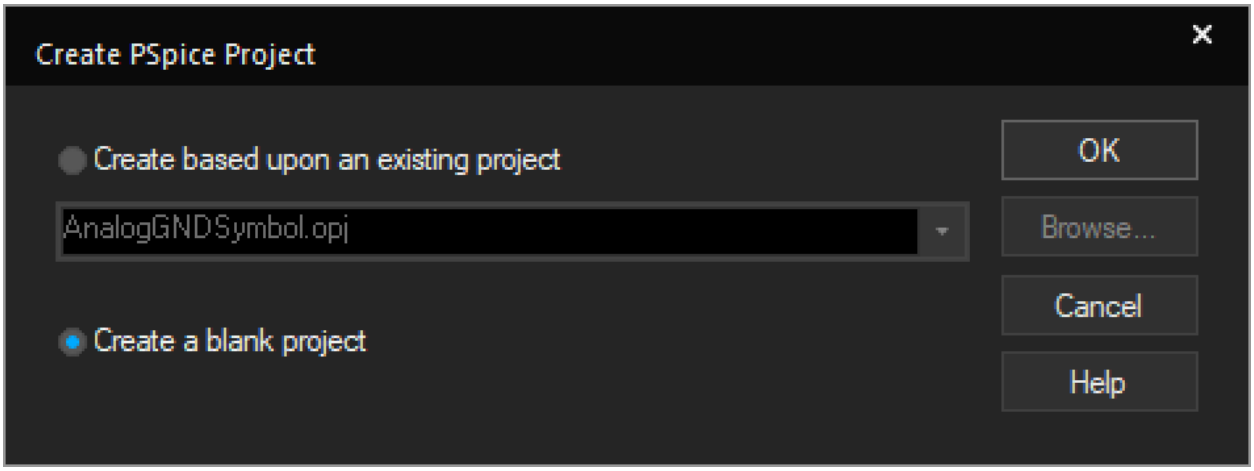
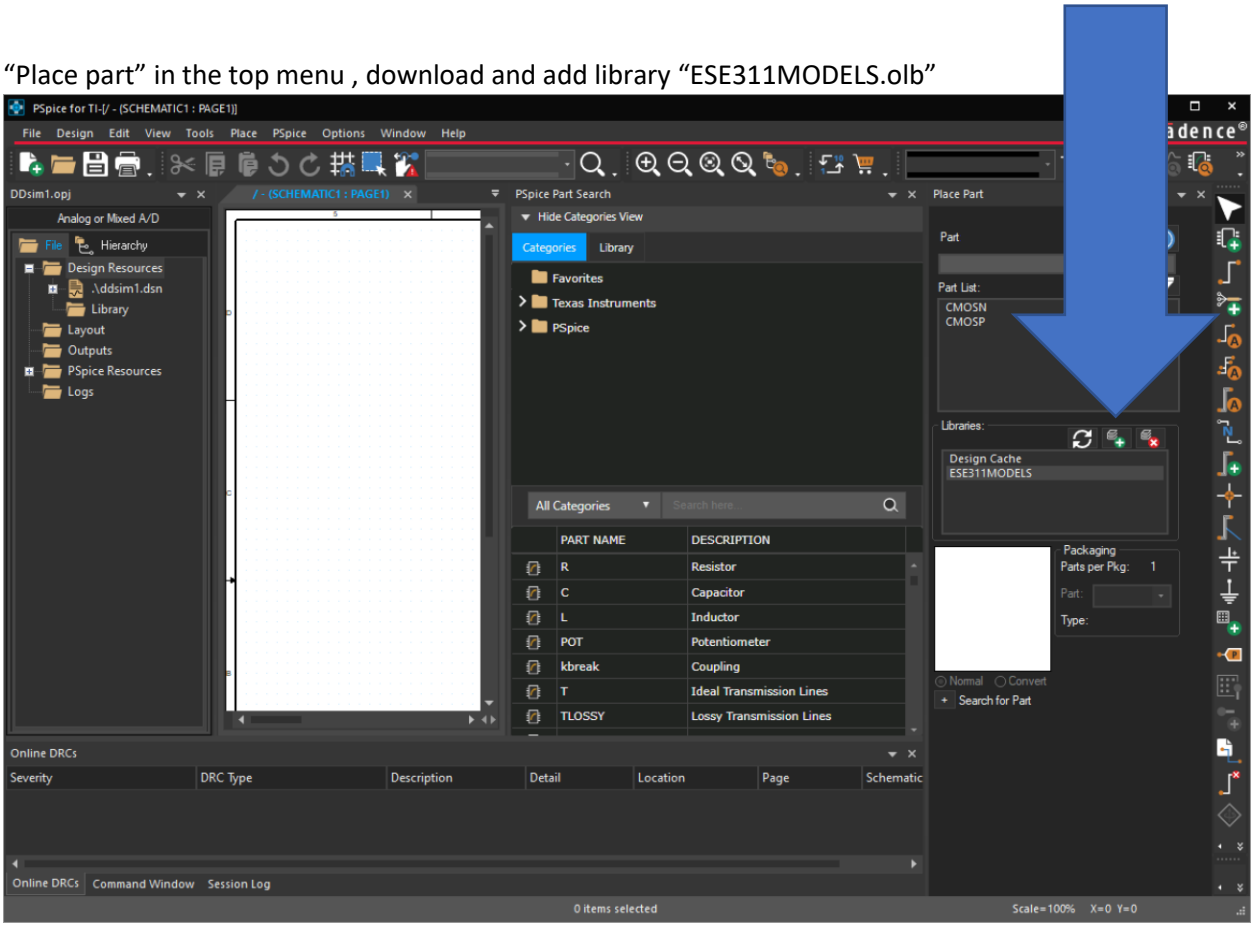
Screenshots for PSPICE for TI



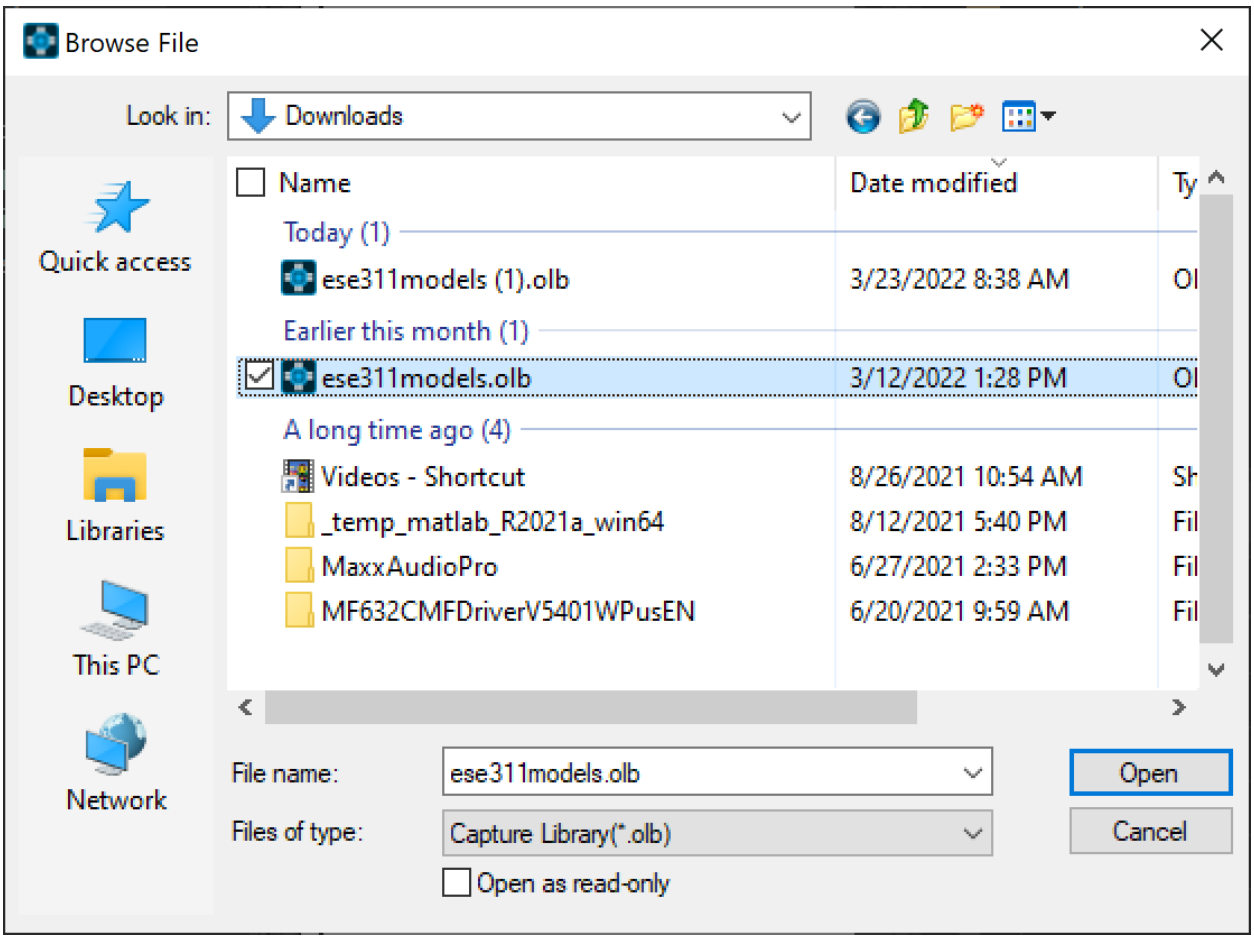


Screenshots for PSPICE for TI

“Place part” in the top menu , download and add library “ESE311MODELS.olb”

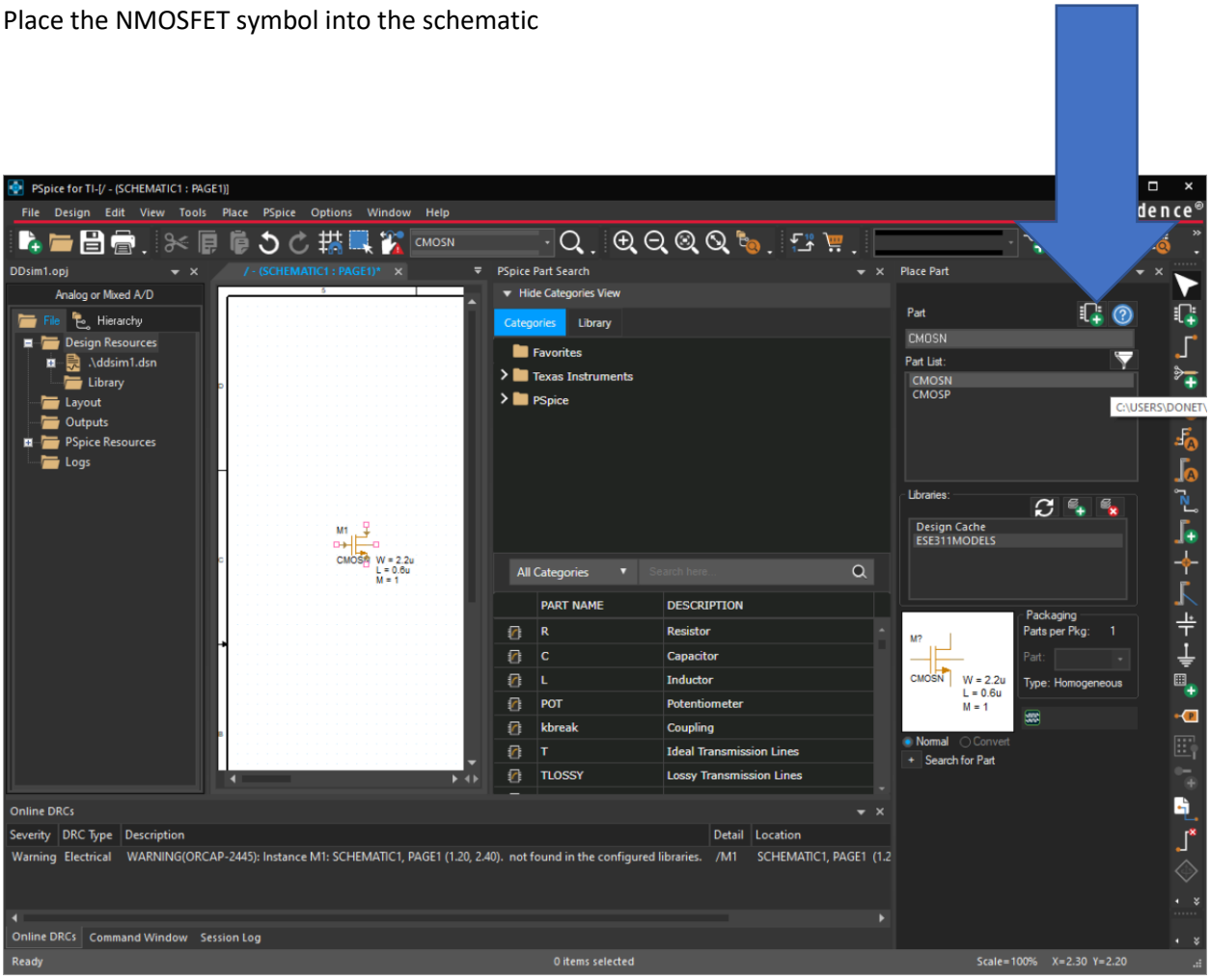


Screenshots for PSPICE for TI



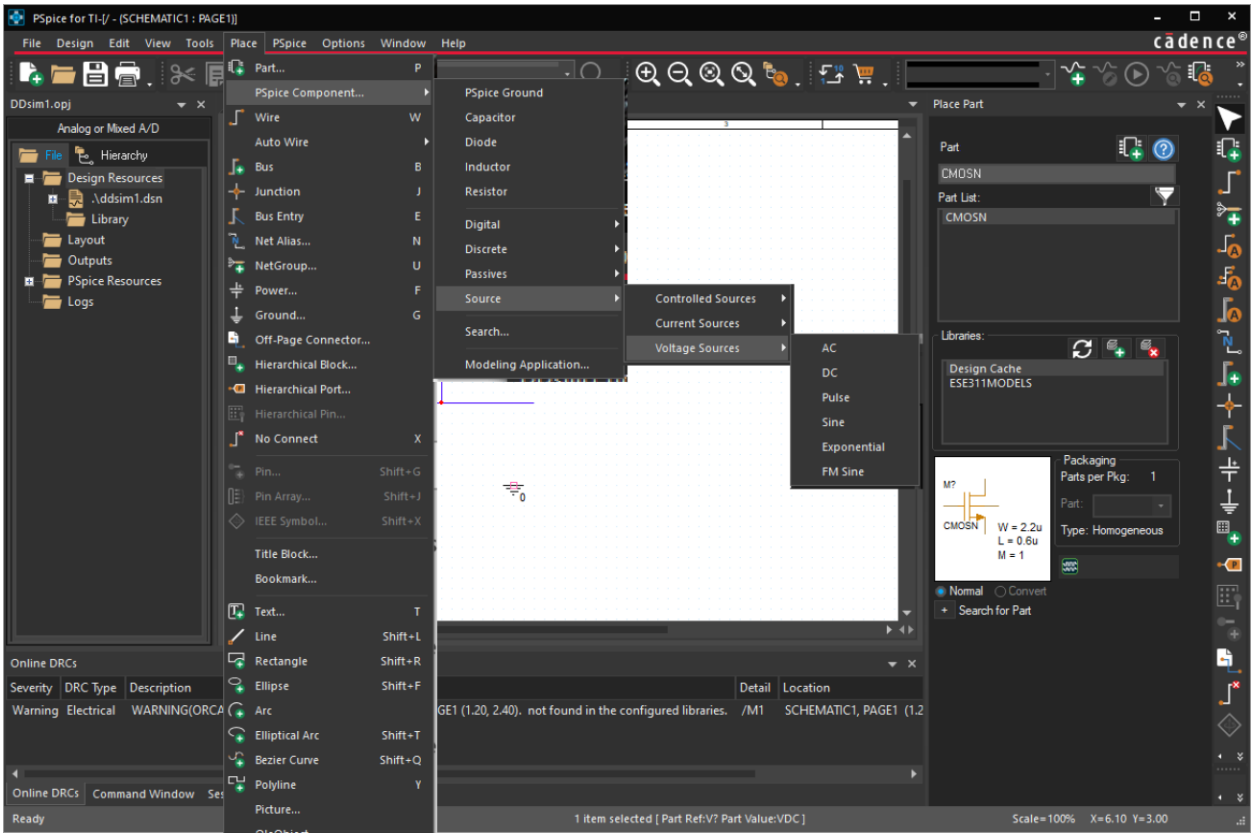
Screenshots for PSPICE for TI

Place the NMOSFET symbol into the schematic



Screenshots for PSPICE for TI

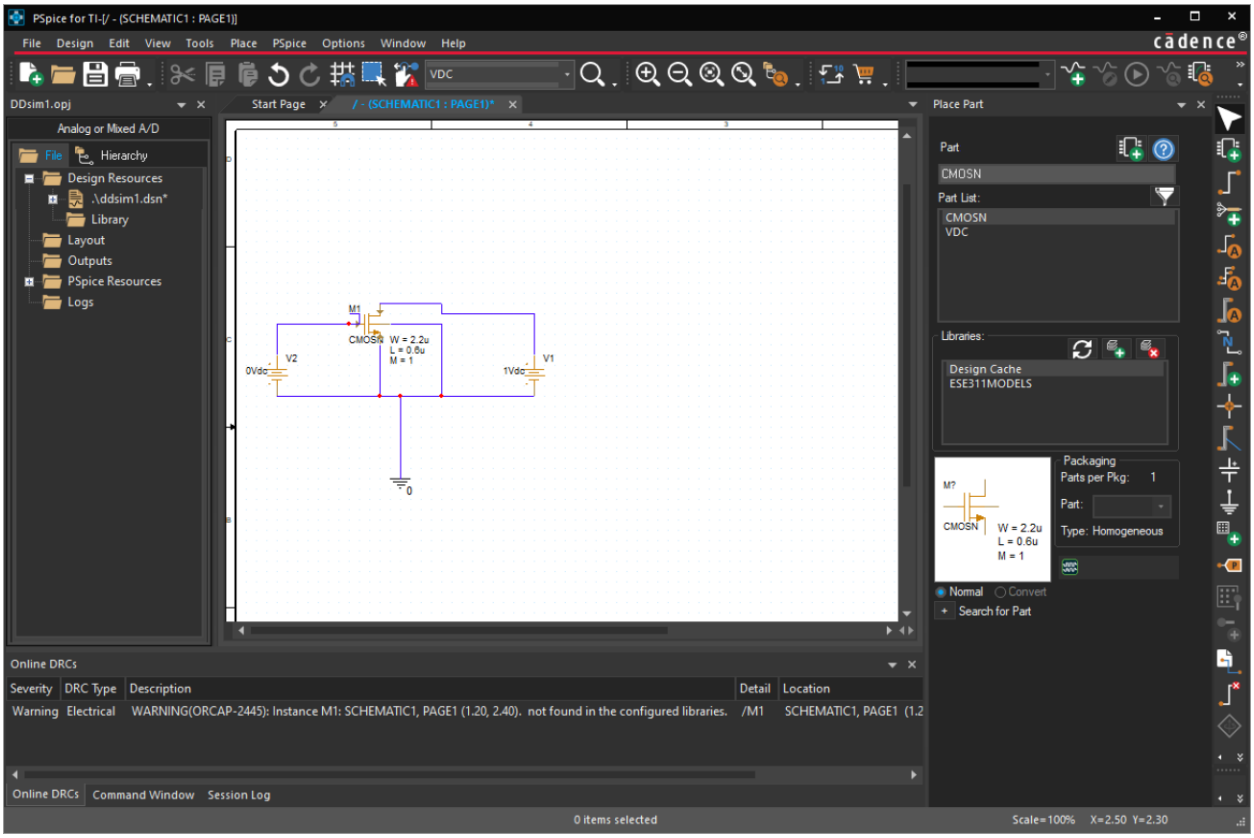
Obtain the schematic for Simulation 1: place PSPICE components: ground with 0, two DC voltage sources.



Screenshots for PSPICE for TI

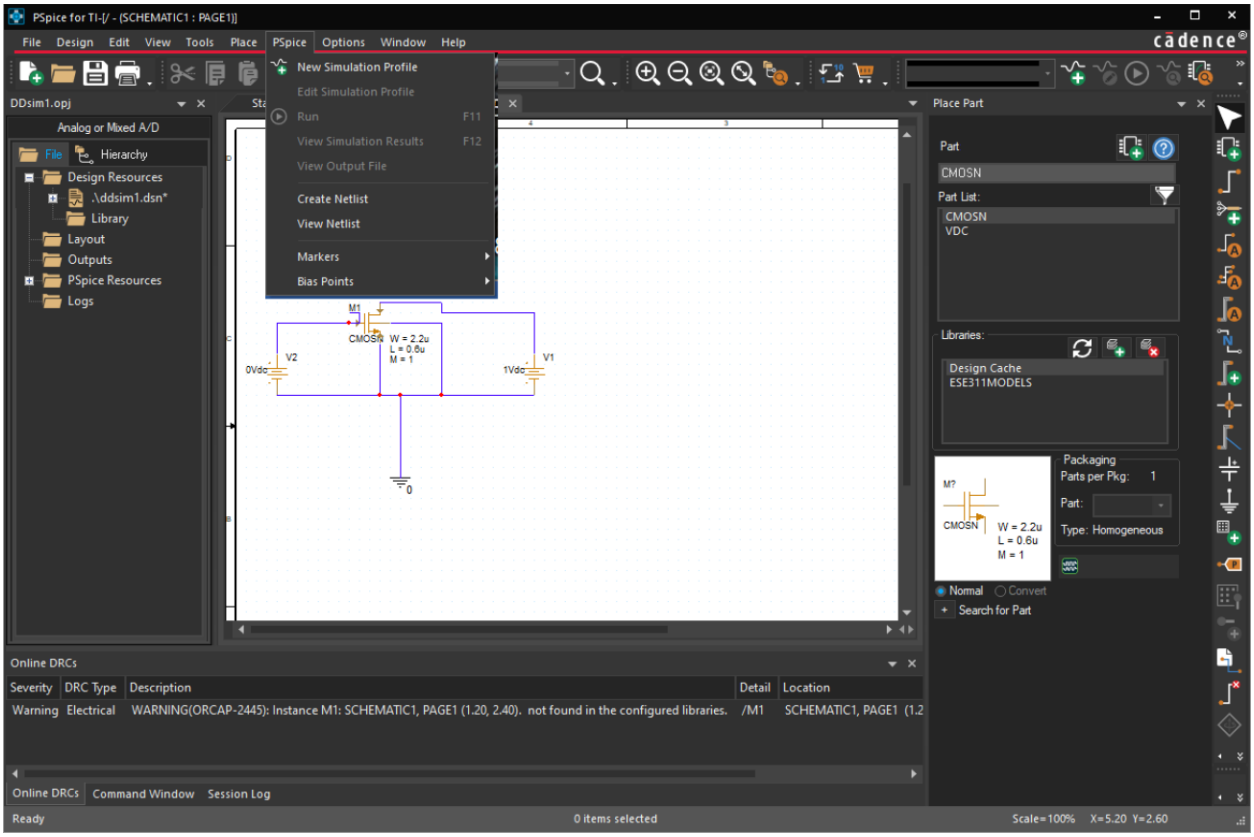
Click and adjust DC voltage: set 1V for the drain to source voltage V1.

V2 can be left at 0 at this time (it will be swept)

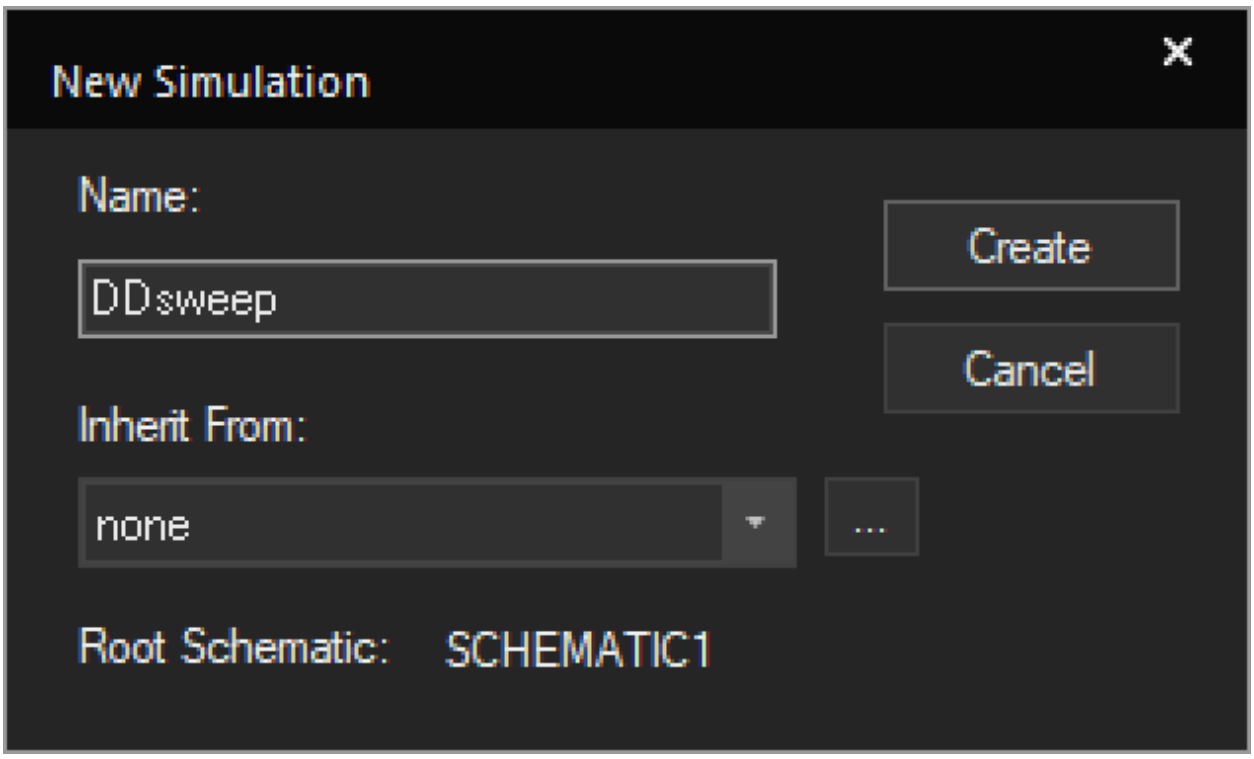


Screenshots for PSPICE for TI

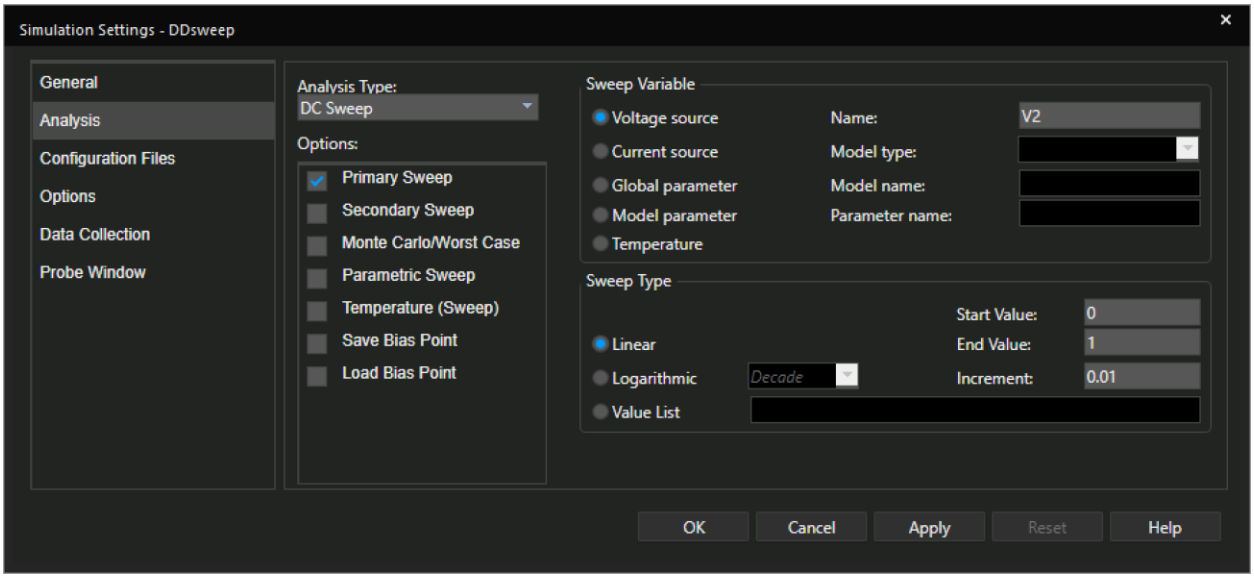
Create a new simulation profile



Screenshots for PSPICE for TI

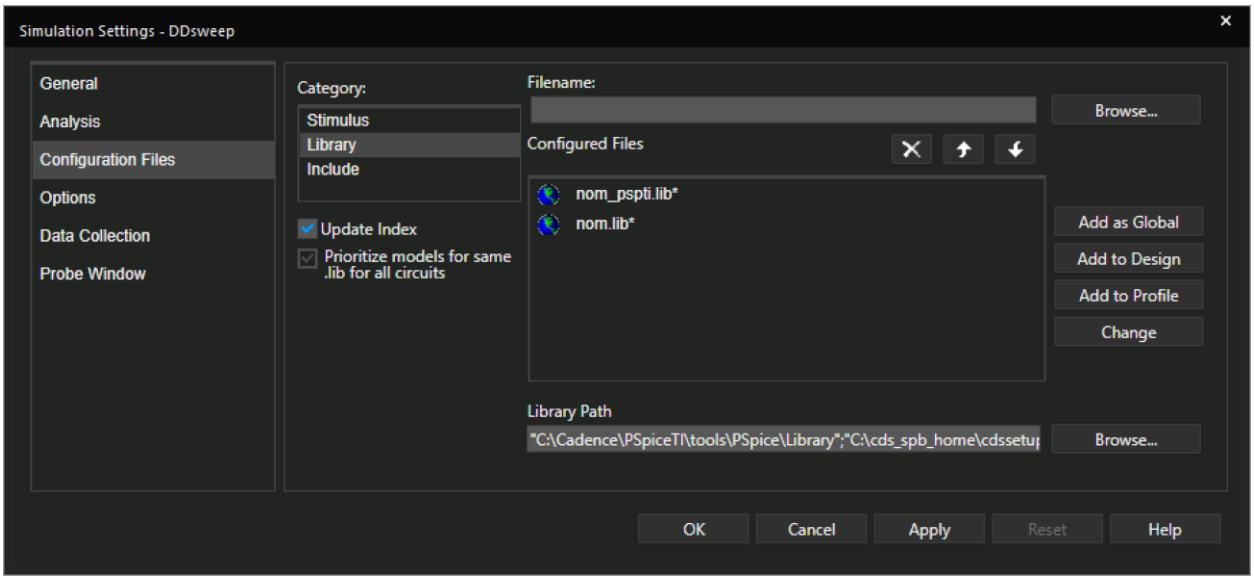


Set the DC voltage range for the gate to source voltage source V2

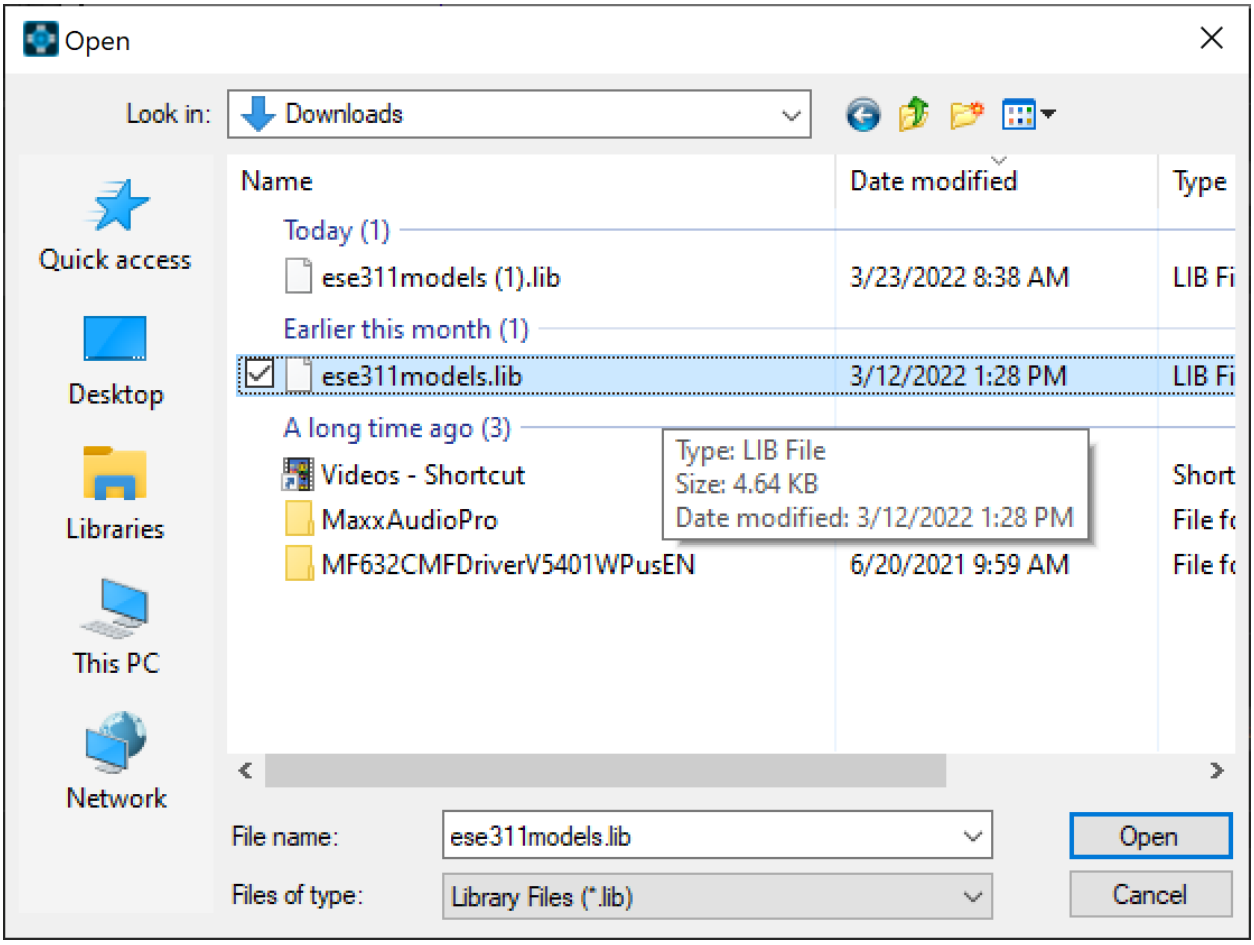


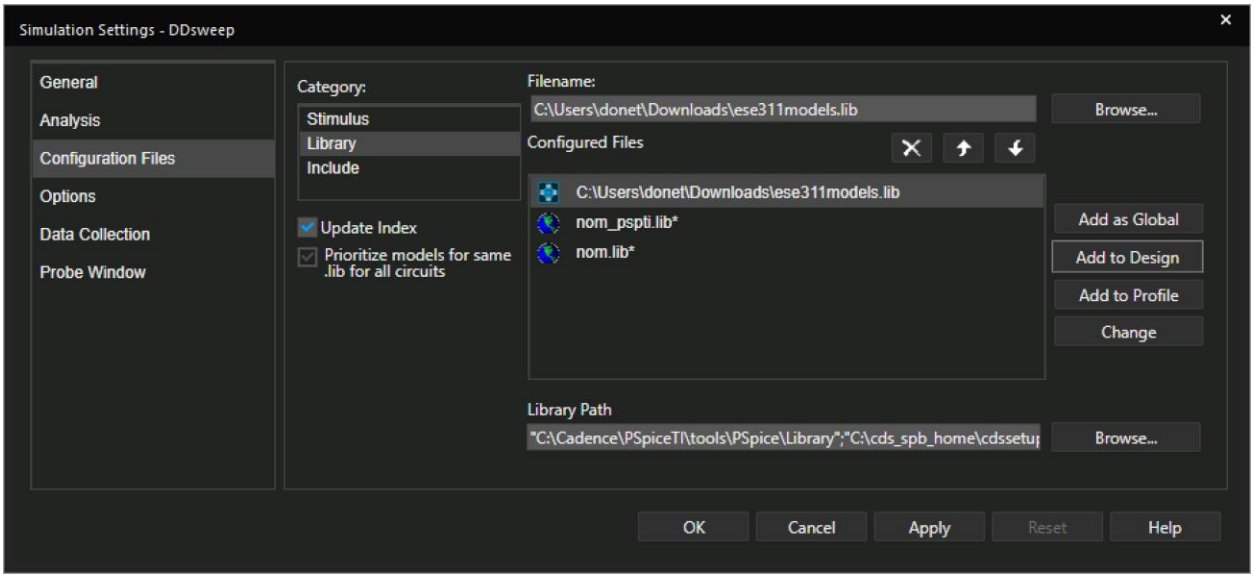
Screenshots for PSPICE for TI

Download the MOSFET models (ESE311models.lib), show a path to the file and “Add to Design” in the Library Category



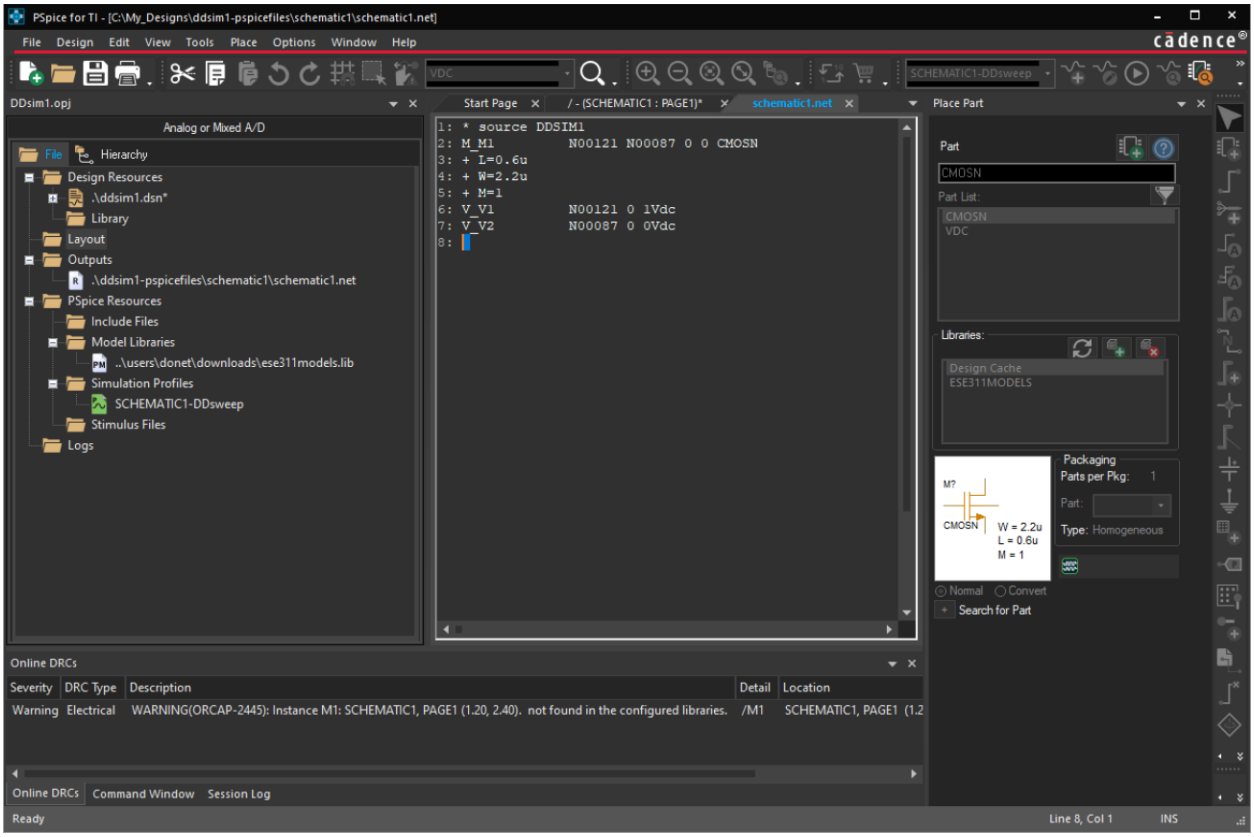
Screenshots for PSPICE for TI





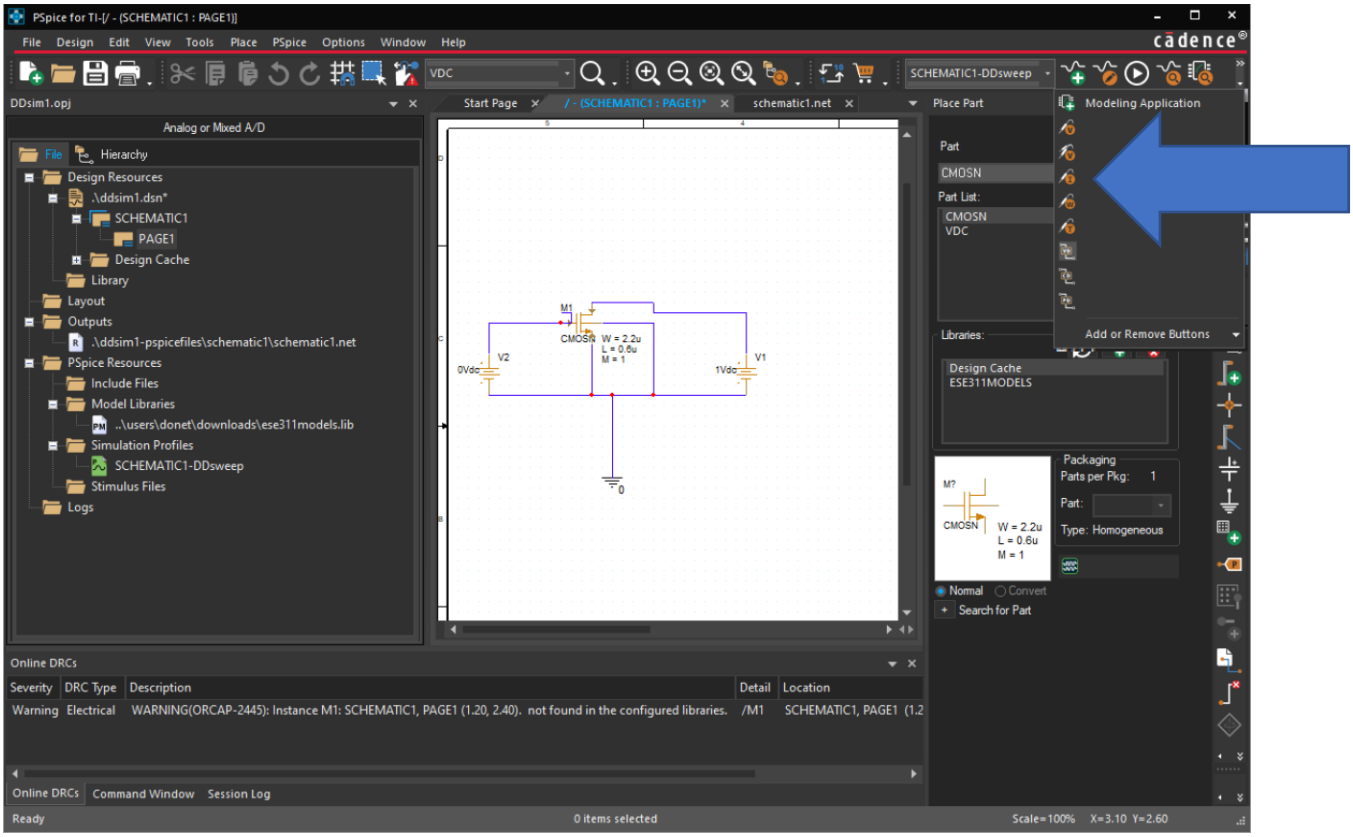
Screenshots for PSPICE for TI

The Hierarchy window should show the lib file in PSPICE resources under Model Libraries folder



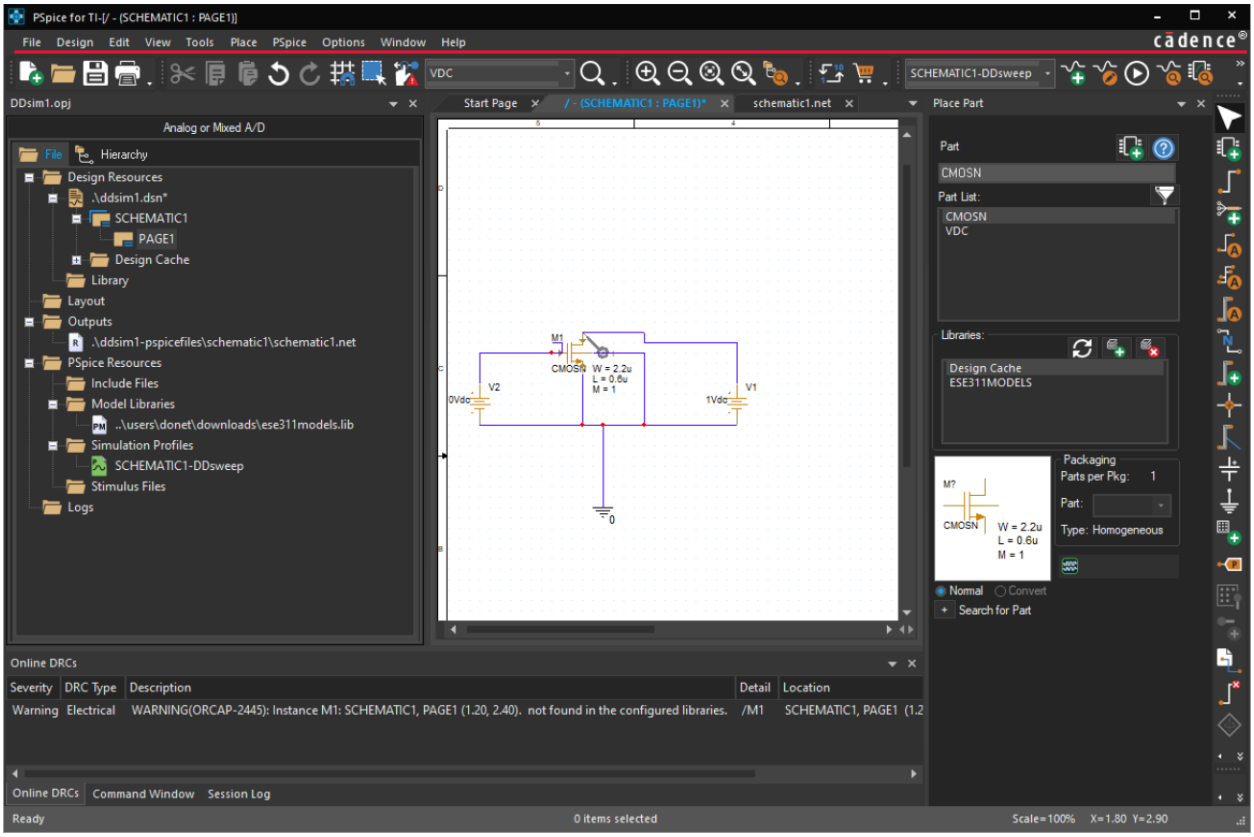
Screenshots for PSPICE for TI

Place a DC current probe into the MOSFET drain terminal



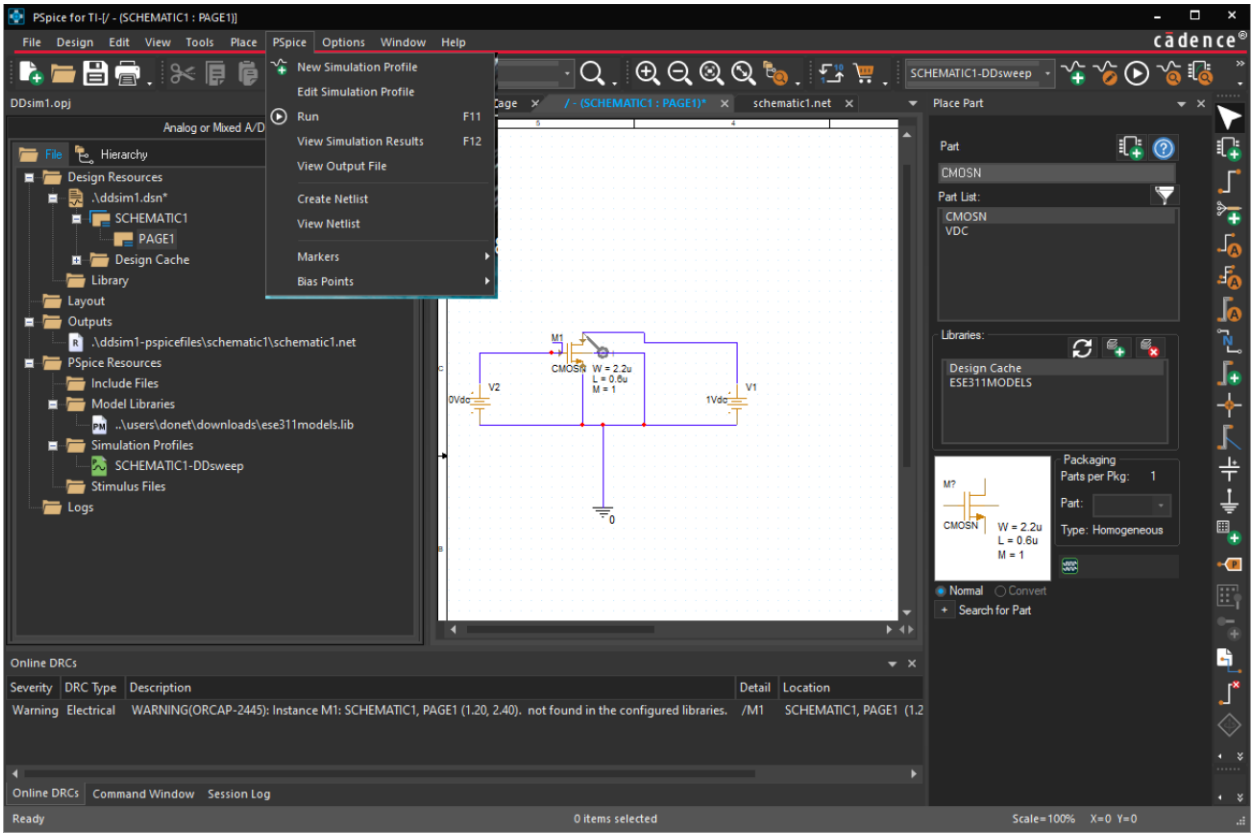
Screenshots for PSPICE for TI

The complete schematic with the DC current probe at the drain terminal: ready to simulate

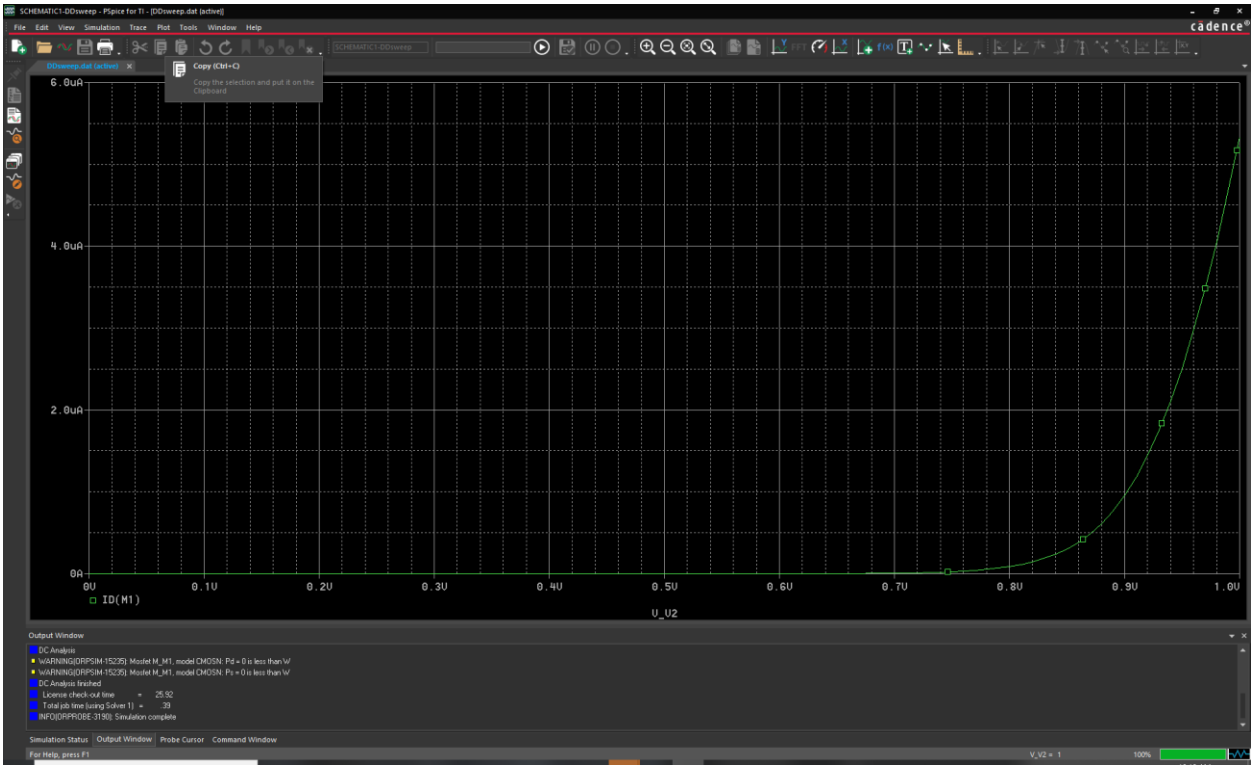


Screenshots for PSPICE for TI

Run the simulation (F11)

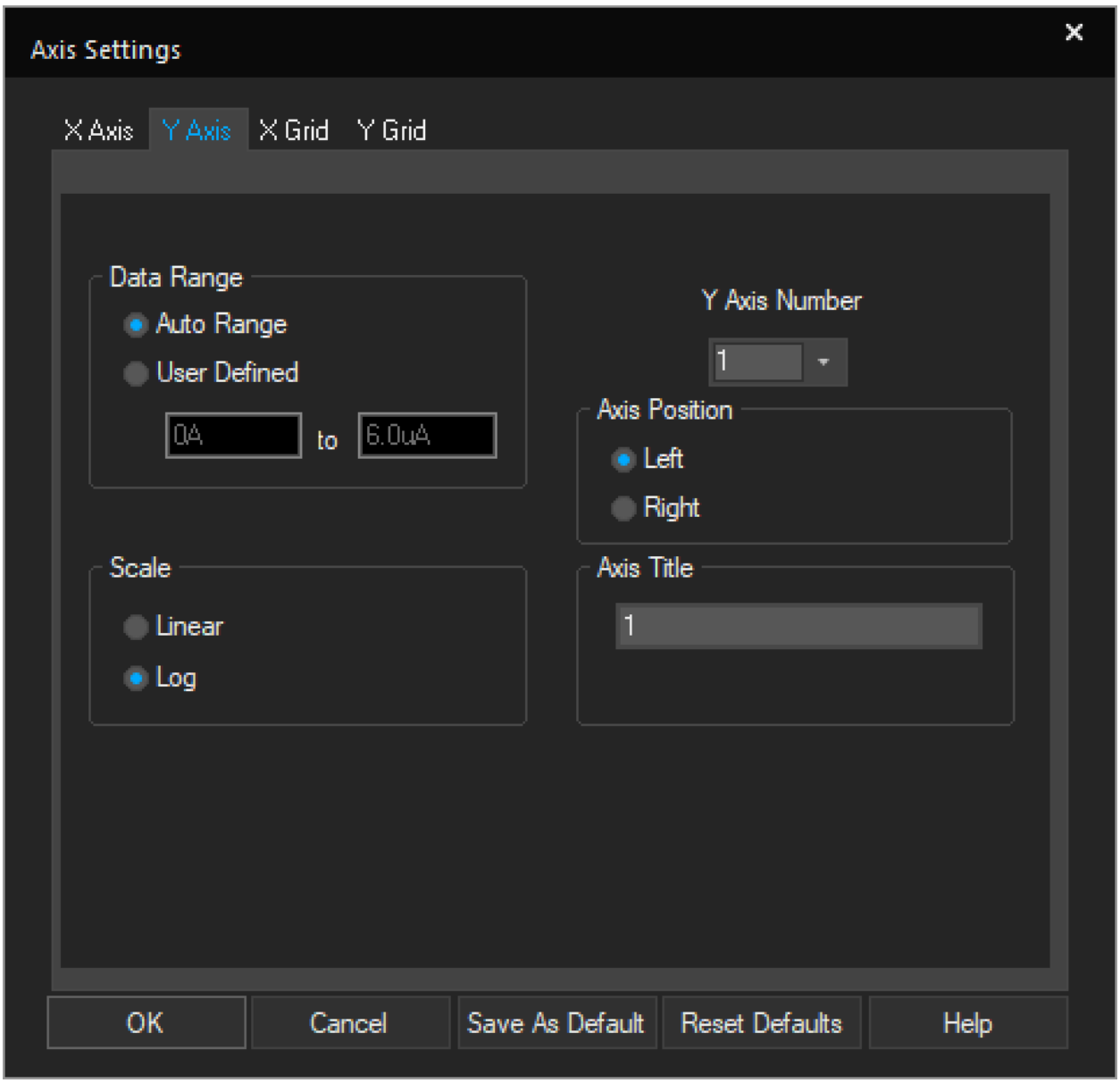


Screenshots for PSPICE for TI

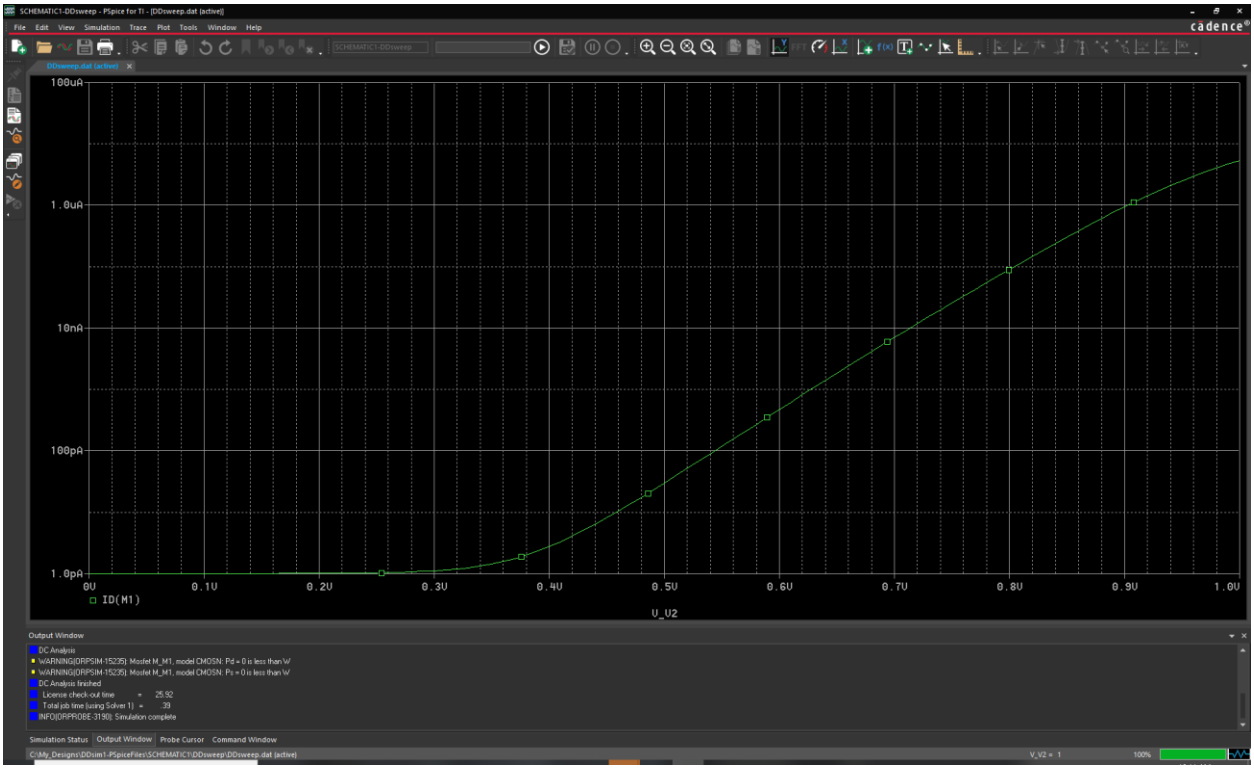


Click Y axis and change the scale to log

Screenshots for PSPICE for TI



Screenshots for PSPICE for TI



Output characteristics can be obtained similarly. A family of output characteristics are obtained by sweeping two DC sources: specify voltage ranges and increments for the primary and secondary sweeps.