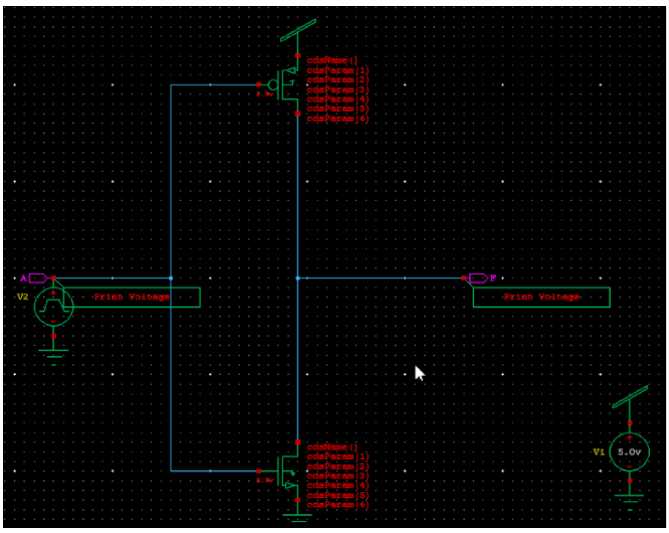
Batch 08- Simulation of circuits in T-SPICE

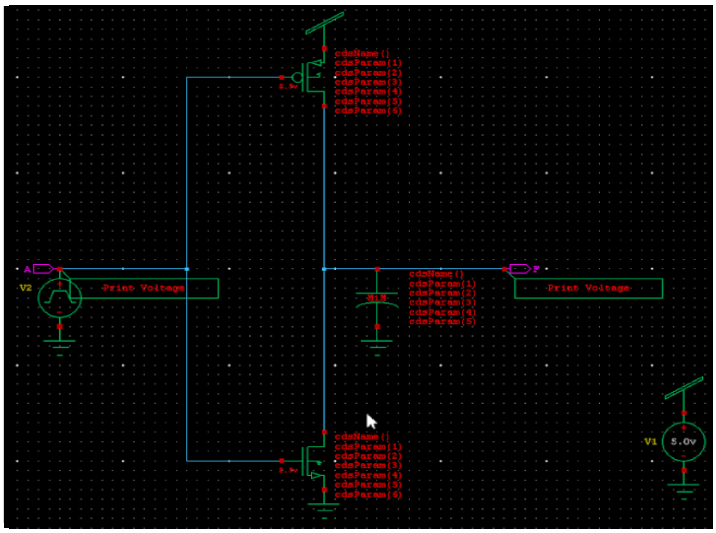
Objective: Develop accurate and efficient simulation models for electronic circuits using T-Spice to analyse and predict circuit behaviour under varying conditions and optimize circuit performance and reliability

Novelty: Advanced integration of compact device models, offering accurate representation of semiconductor behaviour. Its enhanced capabilities provide engineers with a powerful tool for development of integrated circuits.

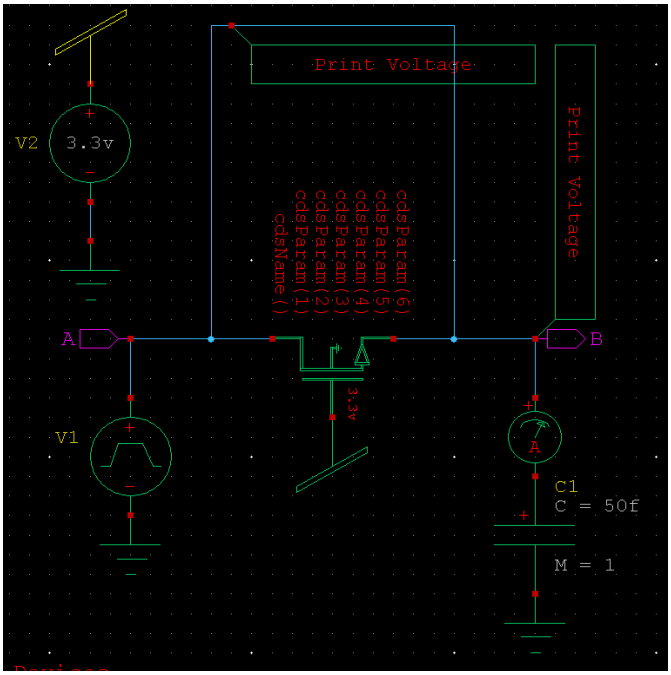
Circuits:

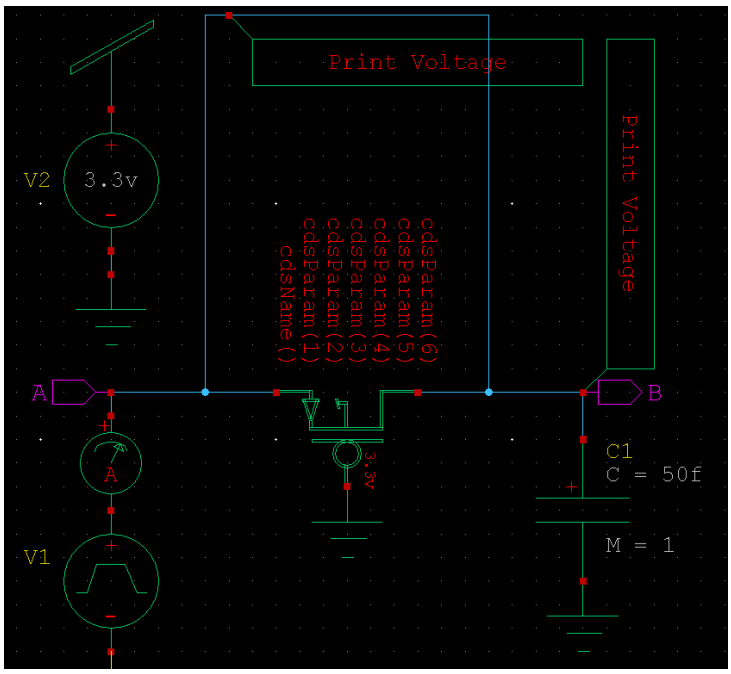
1.CMOS Inverter – (without and with capacitor)



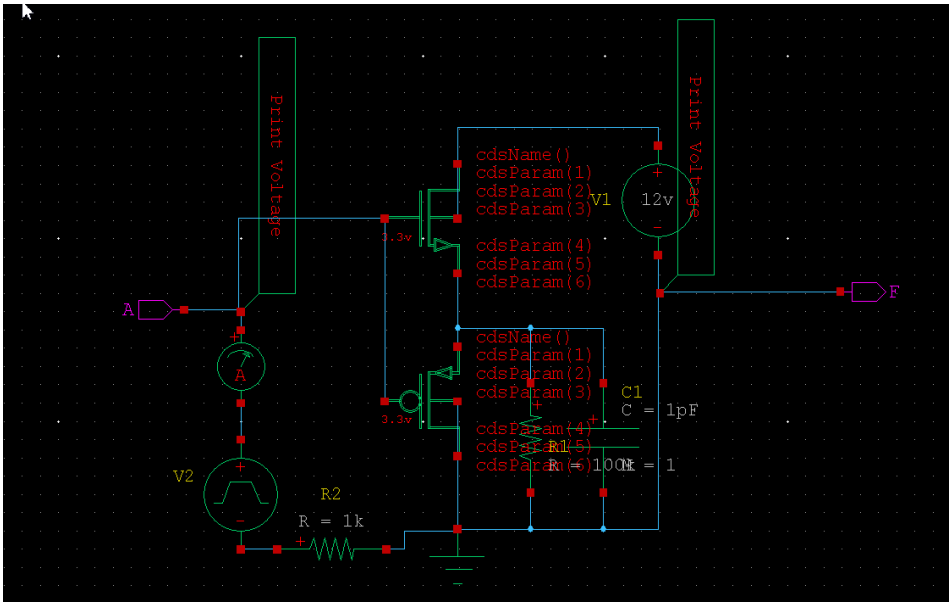


2.Effective Resistance of CMOS Transmission Gate

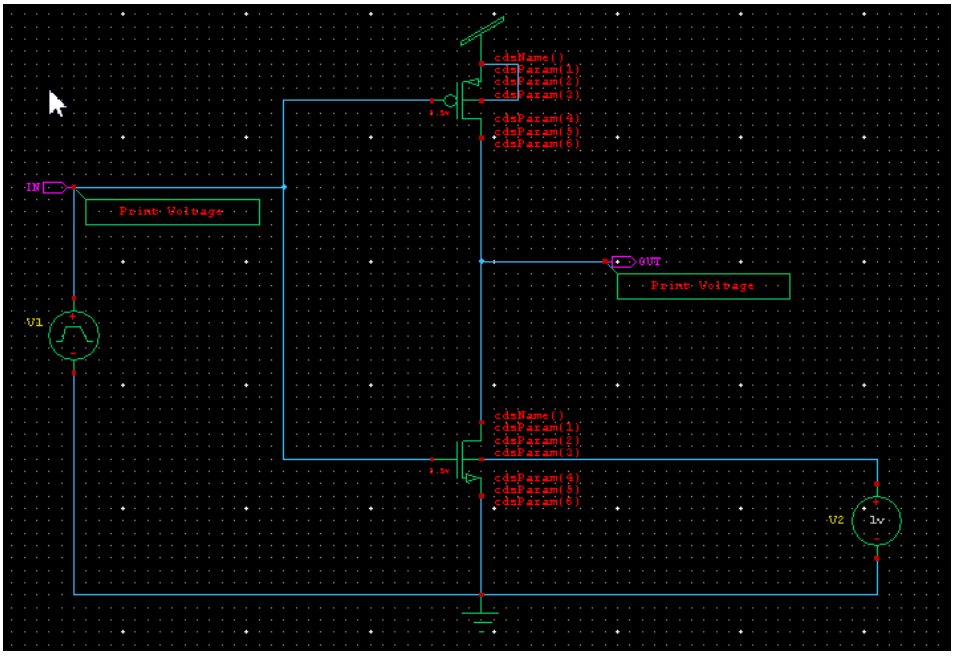




3.Gate Capacitance extension using ckt simulator.



4.Threshold voltage circuit.



<u>Contributions</u>:

1. Arjit Avadhanam (S20210020257) – Simulation of the circuits – Effective Resistance of CMOS Transmission Gate, Gate Capacitance extension using ckt simulator.

2. Moses Jalli (S20210020282) – CMOS circuit with and without capacitor, Threshold voltage circuit.