Measure Average Current in a Specific Time Region

1. Select the 'source' of the driving inverter to be plotted

   In ADE, there are two ways to select a node to be plotted.

   - **Select the node on schematic**
     1) Select the driving gate (the first inverter in this case)
     2) Press 'X' to descend in hierarchy and open in current tab. Or you can do Edit --> Hierarchy --> Descend Read

   ![ADE Schematic Editor](image)

   3) You should see the transistor-level design of your gate.
   4) Now go back to ADE window: Outputs --> To Be Plotted --> Select On Schematic. Click on the source of the pmos, so the current flowing through this node will be plotted in the waveform (after simulation.)
5) After selecting the source, you can press 'B' or Edit --> Hierarchy --> Return to the higher level (in this example it's also the top-level.)

- **Manually key-in the Outputs field**

Outputs --> Setup

Similar to adding a transient voltage, but now it is for transient current. Here the Name field is left empty (it's optional.)

For Expression field: IT("/I0/M0/S")
'I0' is the instance value for the driving inverter. 'M0' is the device value for the pmos. 'S' stands for source.

2. Define the time region to measure the current

1) After the simulation, you will have three transient responses.

2) Zoom-in to a specific time region (0.1ns before input falls and 0.1ns after input falls.) Change the axis to this specific time. In this example, that is from 3.6ns to 3.9ns.

Click on axis first, Axis--> Edit
3. Evoke Calculator to compute the average current

1) Click on the 'Calculator' icon

2) After the calculator window opens, click on the waveform (this clip of signal will be put into the buffer area.)

3) then select the 'average' function under Special Functions. Then click the 'Evaluate' button.