

4th question is simulated using LTSPICE

First of all, we need to models that are given to us.

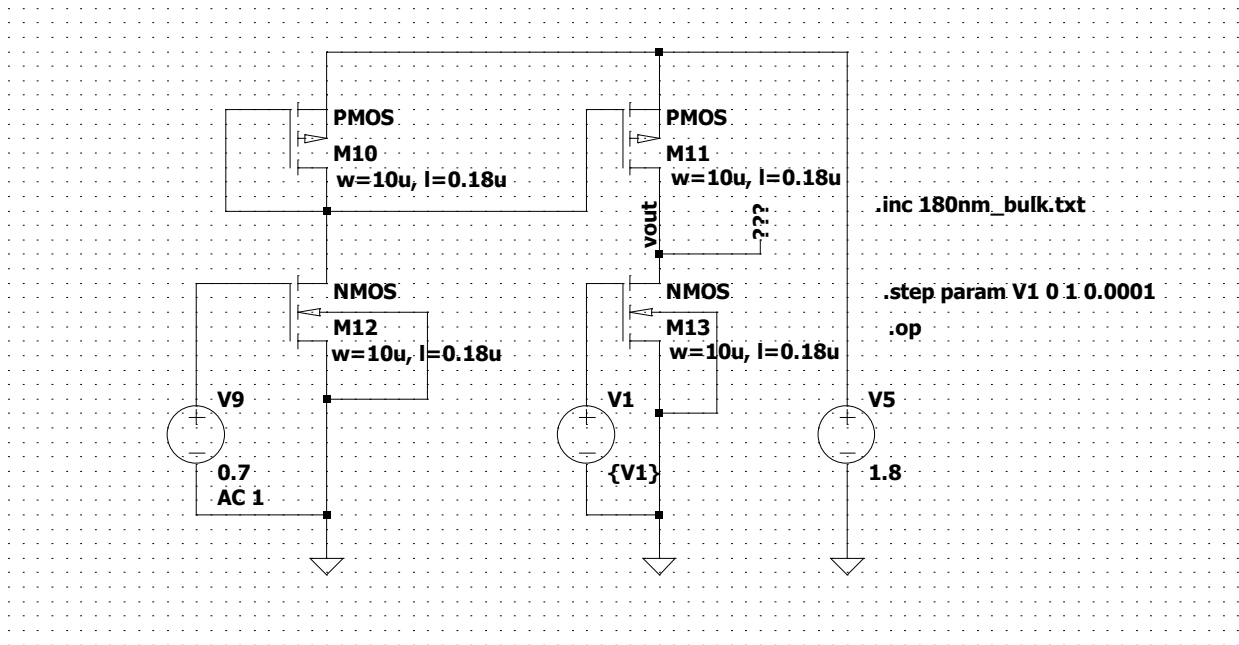
In order to add these models in LTspice,

1-) Models codes are copied into a txt file named 180nm_bulk.txt

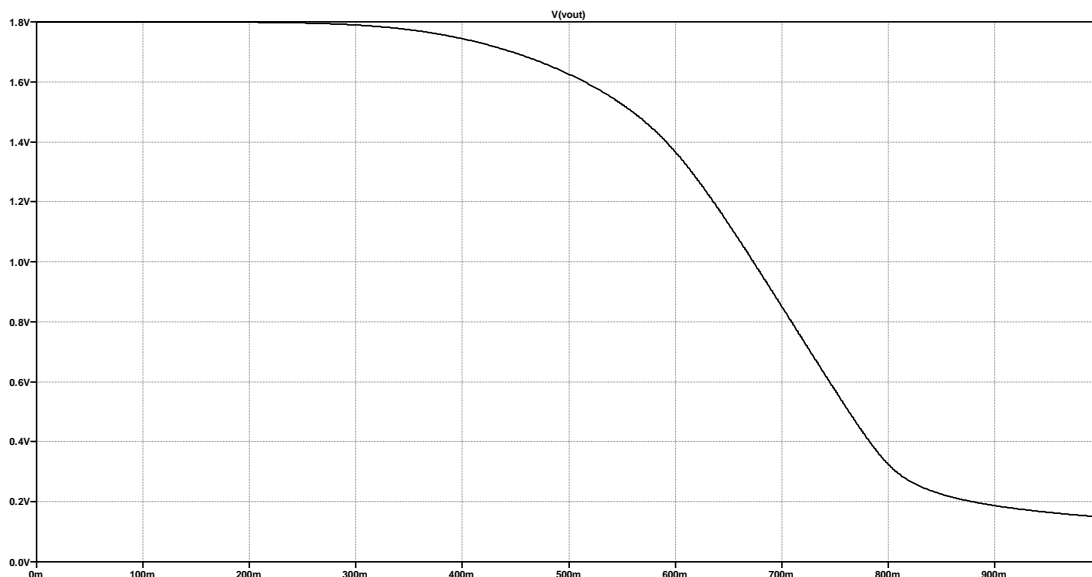
2-) We include this txt file using the command: `.inc 180nm_bulk.txt`

After the inclusion of the models, draw the asked circuit and select the asked transistors sizing.

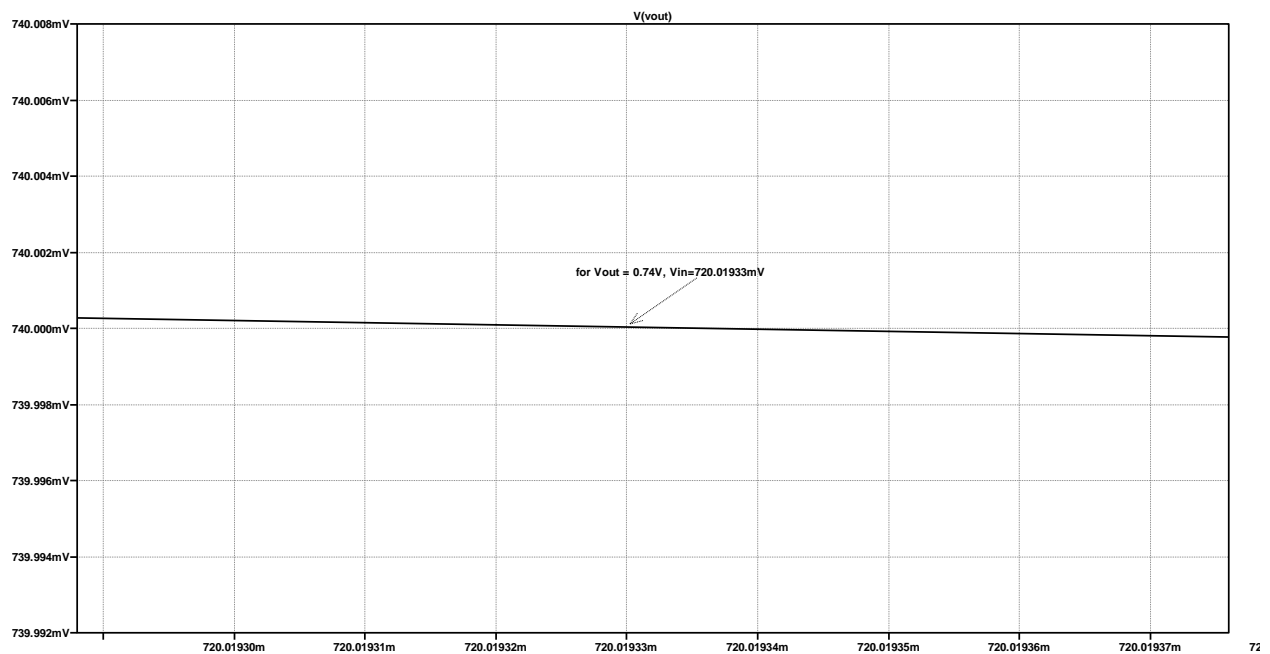
The circuit is drawn as shown:



Then, in order to determine the V1 voltage (input to M13) that corresponds to an output voltage of $V_{out} = 0.74V$, I increased V1 from 0V to 1V with small increments. The reason of increasing with small steps is the sensitivity of the circuit. It is very sensitive to small V1 voltages and the $V_{out} - V1$ relationship can be seen from the figure:

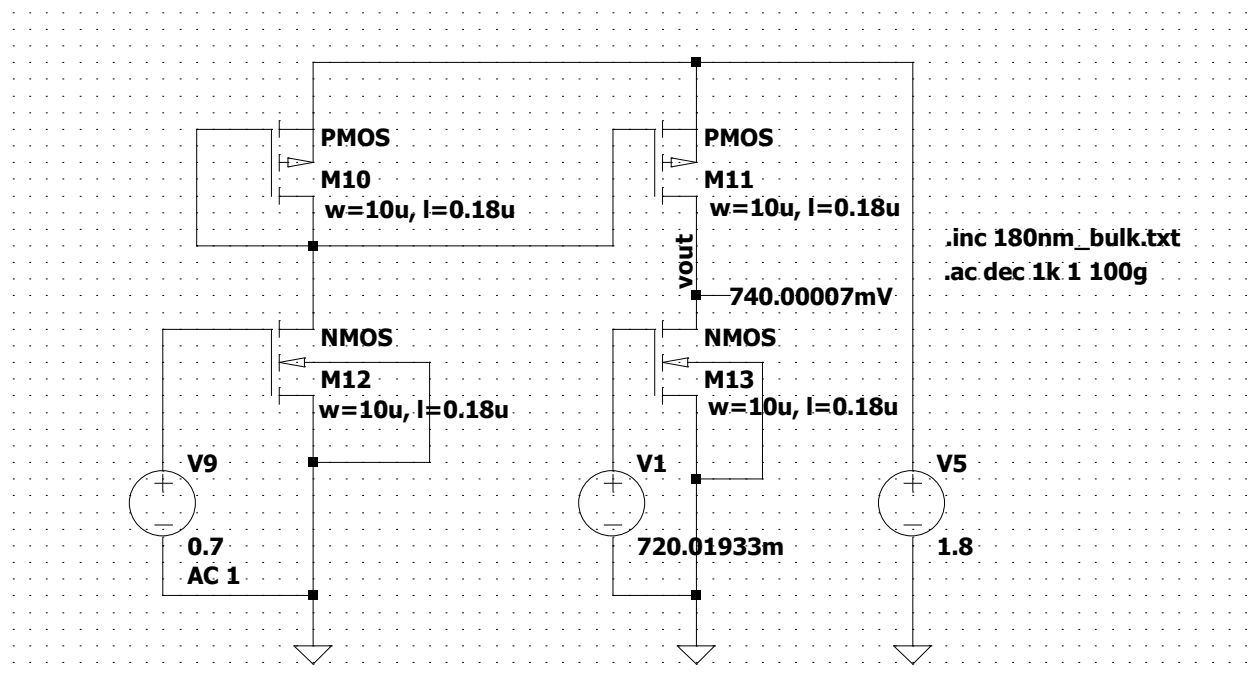


Then we can determine the V1 voltage that corresponds to 0.74V of output voltage from the figure. And to be precise let's zoom in the considered area. We can see the V1 voltage from the figure:

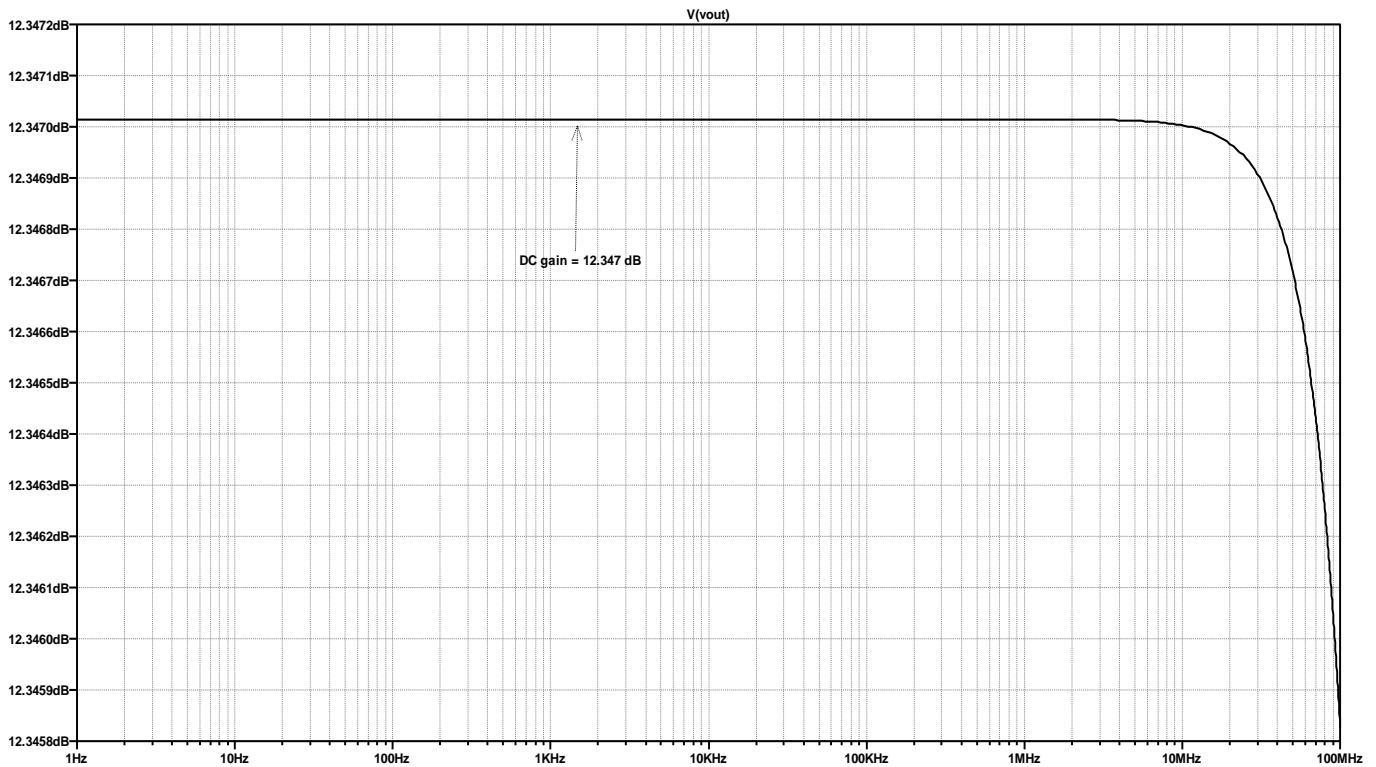


As it can be clearly seen $V1 = 720.01933\text{mV}$ corresponds to $Vout = 740\text{mV}$

Then by selecting $V1 = 720.01933$, analyze the gain of this circuit structure. We are using ac analysis for 1k points from 1 Hz to 100Ghz. The code for this is shown in the figure.

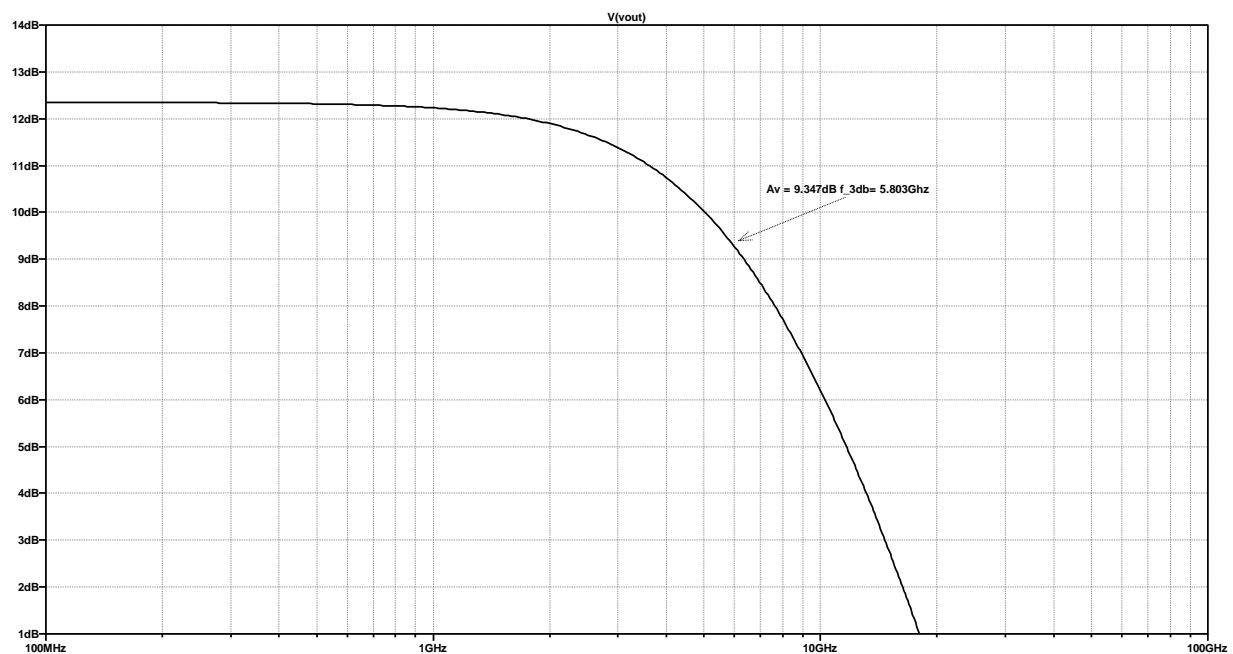


If we run the circuit above, we get a frequency response as shown:



From this figure we find the DC gain of this circuit as $A_v = 12.3470 \text{ dB}$

Then let's zoom in to see at which frequency the circuit power halves. In other words, the gain drops by 3dB. The zoomed figure as shown:



Now from this figure we can say that $A_v = 12.3470 \text{ dB}$ and $f_{3-dB} = 5.803 \text{ GHz}$

