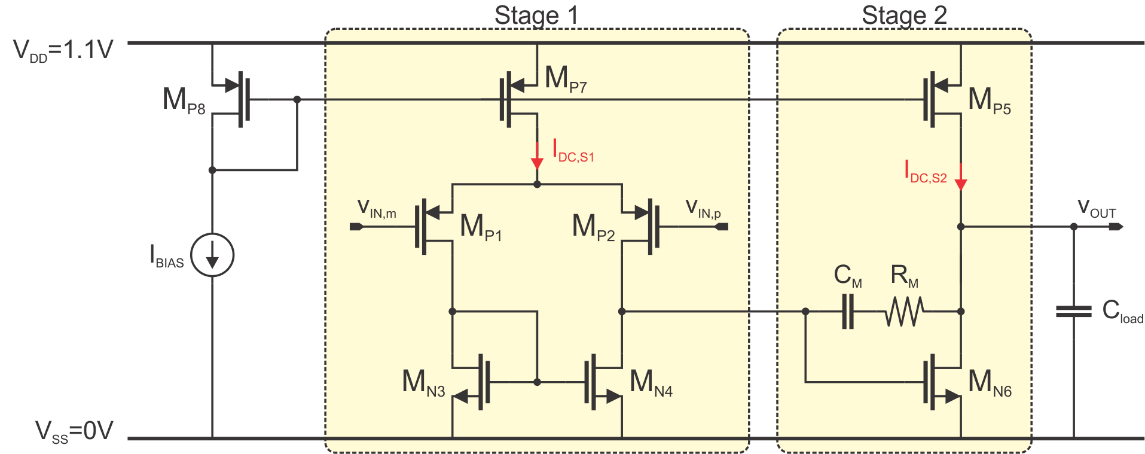
**Analog Electronic Circuits – 2020-2021**

**Design Project Report**

# Group data

|  |  |
| --- | --- |
| Group number | 09 |
| Name – Student 1 | Leander Hemelhof |
| Name – Student 2 | Vlad-Eusebiu Baciu |

# Goal:



Design of the 2stage OpAmp such that it passes the given specifications (insert your specs below):

|  |  |
| --- | --- |
|  | 30 |
| DC gain [dB] | 49 |
| [MHz] | 28 |
| Phase margin (PM) [deg] | > 70 |
| Output swing [V] | > 0.7 |

# Plan: Design of the 2-stage OpAmp on paper (10 points)

### Calculate the and L of each transistor (see exceptions under “Remark”) and the required and in the OpAmp circuit. For that, insert all your calculations as well as your -, -plots you used for your handcalculations below:

Hints:

* First, plot and across and gatelength L and do it again across , as presented in the 1stsession. Think about whether to create the plots for a PMOS or for a NMOS device.
* Then, based on those plots, start your calculations.
* Furthermore, you can assume the following:   
  (1) the OpAmp is designed in triple-well-technology (i.e. ; (2) and across ; (3)

Remarks:

* For Mp5, Mp7 and Mp8 you are not required to calculate the

Plots used for the handcalculations:

|  |
| --- |
| D:\Facultate\VUB_sem1\Analog Electronics\Learn\Final_project\plots\nmos\gm_over_gds_VOV.png  D:\Facultate\VUB_sem1\Analog Electronics\Learn\Final_project\plots\pmos\gm_over_gds_VOV.png |
| **Figure 1:** vs across gatelength |

|  |
| --- |
| D:\Facultate\VUB_sem1\Analog Electronics\Learn\Final_project\plots\nmos\gm_over_gds_VGS.png  D:\Facultate\VUB_sem1\Analog Electronics\Learn\Final_project\plots\pmos\gm_over_gds_VGS.png |
| **Figure 2:** vs across gatelength |

|  |
| --- |
| D:\Facultate\VUB_sem1\Analog Electronics\Learn\Final_project\plots\nmos\gm_over_IDS_VOV.png  D:\Facultate\VUB_sem1\Analog Electronics\Learn\Final_project\plots\pmos\gm_over_IDS_VOV.png |
| **Figure 3:** vs across gatelength |

Handcalculations:

Assume that Mp8 and Mp7 are matched: => I\_BIAS = I\_DC\_S1

I\_DC\_S2 should be the same as I\_BIAS if Mp5 is the same as Mp8,Mp7.

Then, choosing a reference current (e.g 25 uA) => Id\_1,2,3,4 = 12.5 uA

Id\_6 = 25 uA.

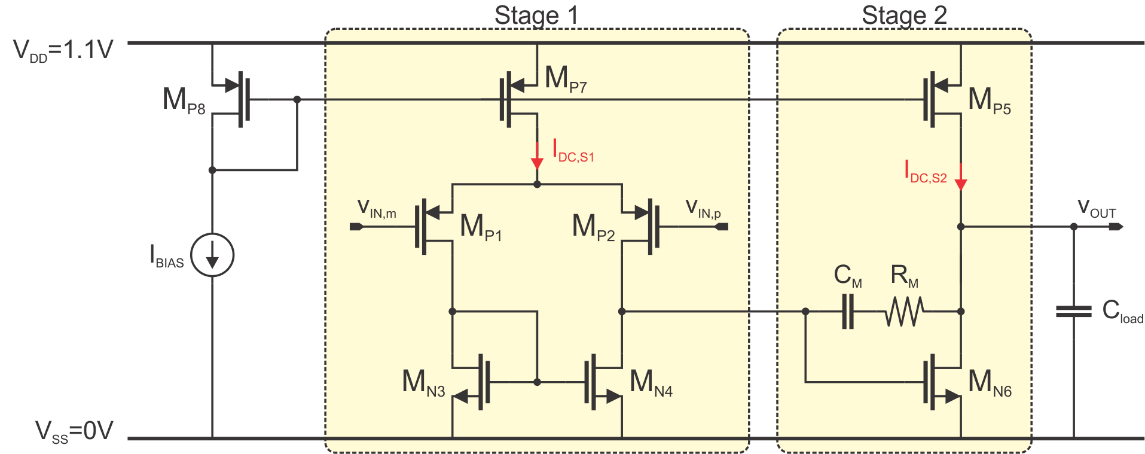
If the input stage is perfectly balanced, then VDS4 = VDS3 = VGS4. Then VGS4 = VGS6

A1 = −gm1(ro2 || ro4)

A2 = −gm6(ro6 || ro5)

### Place all the calculated voltages (in the blackbox) and all currents (in the red box) on the circuit depicted below and calculate also:

|  |  |
| --- | --- |
|  |  |
|  |  |
|  |  |
|  |  |
|  |  |



# Design: Implementation of the OpAmp in MATLAB and LTspice (10 points):

### Based on your handcalculations, create your OpAmp design in MATLAB. After completion, please insert your final and entire MATLAB code after the appendix (i.e. at the end of this report-document).

### Fill out both tables depicted below. All values are to be determined in MATLAB.

Device sizes and bias point parameters according to MATLAB

|  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| **Device** | **W**  **[μm]** | **L**  **[nm]** | **Ids**  **[μA]** | **VOV**  **[V]** | **gm**  **[S]** | **gds**  **[S]** | **gm/gds**  **[-]** | **Vds,sat**  **[V]** | **Vds**  **[V]** |
| Mp1 |  |  |  |  |  |  |  |  |  |
| Mp2 |  |  |  |  |  |  |  |  |  |
| Mn3 |  |  |  |  |  |  |  |  |  |
| Mn4 |  |  |  |  |  |  |  |  |  |
| Mp5 |  |  |  |  |  |  |  |  |  |
| Mn6 |  |  |  |  |  |  |  |  |  |
| Mp7 |  |  |  |  |  |  |  |  |  |
| Mp8 |  |  |  |  |  |  |  |  |  |

|  |  |  |
| --- | --- | --- |
| **Device** | **Units** | **Value** |
| CM | pF |  |
| RM | Ω |  |
| IBIAS | A |  |

### Based on the parameters filled in the table above, design your OpAmp in LTspice. After completion, please insert your final and entire LTspice-netlist after the appendix (i.e. at the end of this report-document).

# Experiment (10 points):

In this section you will simulate the following:

1. Frequency Response in MATLAB and LTspice
2. Noise contribution in LTspice
3. Linearity in LTspice

Note on how to present graphs in general:

* When you make a graph, put labels on every axis to make clear what you are showing. Also, show the units!
* When you plot multiple curves on one graph, add a legend.

1. Frequency Response in MATLAB and LTspice

Note on how to present graphs in this sub-section:

* For magnitude plots use dB (linear scale) vs. Hz (logarithmic scale)
* for phase plots use ° (linear scale) vs. Hz (logarithmic scale).

### Simulate (small voltage gain) with and in MATLAB and LTspice. Then, paste both -curves in one plot below such that the difference between MATLAB and LTspice can be clearly seen. In addition to that, indicate the resulting phase margin (PM) in both plots.

**Plots:**

|  |  |
| --- | --- |
| **Simulator** | **PM (deg)** |
| MATLAB |  |
| LTspice |  |

### Explain, analyse and interpret the results in “6)”. Furthermore, if you observe significant difference between MATLAB and LTspice, explain why.

### Simulate in LTspice for the following cases:

### No compensation network ().

### With compensation capacitor but no compensation resistor ().

### With both the compensation capacitor and the compensation resistor.

### Then, show all 3 cases in one plot such that the differences are clearly visible. Furthermore, indicate on this plot the resulting PM for each case.

**Plot:**

|  |  |
| --- | --- |
| **Case** | **PM [deg]** |
| No compensation network |  |
| No compensation resistor |  |
| With both the compensation |  |

### Explain, analyse and interpret the results in “8)”.

1. Noise contribution in LTspice

### Simulate the output-referred noise voltage power density over an appropriate frequency range in LTspice. Then, insert the -plot below.

**Plots:**

### Explain, analyse and interpret the results in “10)”.

### Simulate the total input-referred noise voltage by using the .NOISE option in LTspice and taking the values of the contribution of integrated noise at the ouput for each element. Then, copy the list of the top 10 noise contributors using "View->SPICE log error" option and insert that list below.

### Explain, analyse and interpret the results in “12)”.

1. Linearity in LTspice

### Simulate the output voltage amplitude and voltage gain as a function of the input voltage amplitude in LTspice. Then, insert the plots below and indicate the 1-dB compression point.

**Plots:**

### Explain, analyse and interpret the results in “14)”.

# Conclusion (10 points):

### Conclude your experiment by filling the editable fields in the performance table depicted below

|  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- |
| **Metric** | **Units** | **Specification** | **from hand-calculations** | **MATLAB** | **LTspice** |
| DC gain | magnitude |  |  |  |  |
| DC gain | dB |  |  |  |  |
| Gain-Bandwidth frequency | MHz |  |  |  |  |
| Dominant pole frequency | kHz |  |  |  |  |
| PM | ° | > 70 |  |  |  |
|  | V |  |  |  |  |
|  | V | > 0.7 |  |  |  |
|  | mW |  |  |  |  |
| (output-referred noise) | Vrms |  |  |  |  |

### Comment about the deviations between hand calculation, MATLAB and LTspice values found above. Conclude, what the causes of such deviations are.

### If you have results which are not passing the specifications: conclude for each of those result what the cause is and what you need to do in order to make it pass, if you were repeating this experiment.

# Appendix:

Insert your MATLAB code here:

Insert your LTspice netlist here: