MÜHENDİSLİK FAKÜLTESİ
FACULTY OF ENGINEERING
ELEKTRİK-ELEKTRONİK MÜHENDİSLİĞİ BÖLÜMÜ
DEPARTMENT OF ELECTRICAL AND
ELECTRONICS ENGINEERING

ORTA DOĞU TEKNİK ÜNİVERSİTESİ
MIDDLE EAST TECHNICAL UNIVERSITY

DUMLUPINAR BULVARI 06800
ÇANKAYA ANKARA/TURKEY
T: +90 312 210 23 02
F: +90 312 210 23 04
ee@metu.edu.tr
www.eee.metu.edu.tr

EXPERIMENT 2. NMOS AND BJT INVERTING CIRCUITS

I. Introduction

I.I Objectives

In this experiment, you will analyze and compare the voltage transfer characteristics (VTC) and the dynamic response of the NMOS and BJT inverters. You will learn how to use Visual Engineering Environment (VEE) for controlling the laboratory instruments

- DC power supply (Agilent E3631A) and
- Digital multimeter (Agilent 34401A)

  for sending and receiving data. You will manipulate this data to display the VTC of NMOS inverter using PC.

II. Preliminary Work

1) Read Chapter 1 (Properties and Definitions of Digital ICs) and sections 4.1 (Analysis of BJT Circuits with Known States) and 4.2 (BJT Inverter) of **Digital Integrated Circuits** by T. A. Demassa and Z. Ciccone.

2) Read sections 7.1 (Design of the MOS Inverter with Resistive Load), 7.7 (Power Dissipation) and 7.9 (Dynamic Response of the NMOS Inverter with a Resistive Load) of **Microelectronic Circuit Design (International Edition)** by R. C. Jaeger.

3) Find **V_{OH}, V_{OL}, V_{IL}, V_{IH}, Noise Margin High** and **Noise Margin Low** of the inverter which has the VTC shown in Fig. 1.

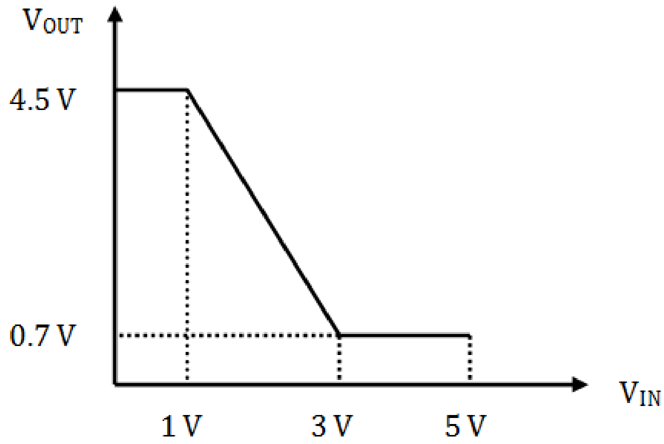

Fig. 1: The VTC of the inverter

4) Determine the **rise time (t_r)**, **fall time (t_f)**, **low to high propagation delay time (t_{LHP}), high to low propagation delay time (t_{HLP})** in terms of t_1, t_2, t_3, t_4, t_5, t_6, t_7 and t_8 for the inverter which has the following input and output waveforms.

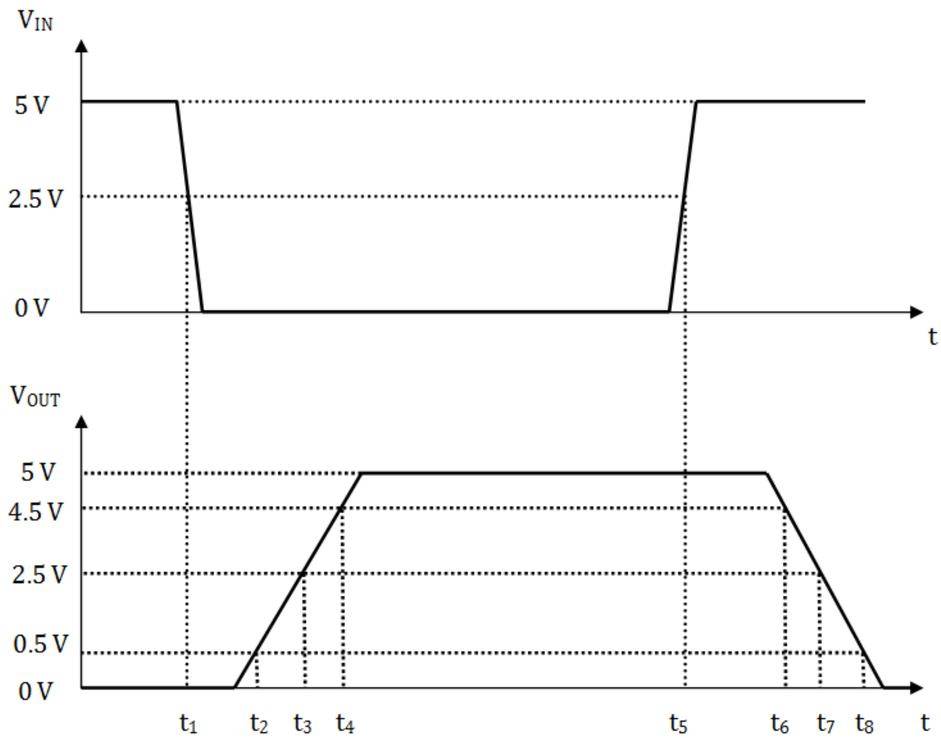

Fig. 2: The input and output waveforms for the inverter

5) Sketch the **VTC** of the circuit in Fig. 3 by finding the critical voltages, $\mathbf{V_{OH}}$, $\mathbf{V_{OL}}$, $\mathbf{V_{IL}}$, $\mathbf{V_{IH}}$. In addition, find **noise margins**.

V_{EE}= -5 V, R_B=10 kΩ, R_C=40 kΩ, V_{BE}(FA)=0.7 V, V_{BC} (RA)=0.6 V, V_{BE}(SAT)=0.8 V, V_{CE}(SAT)=0.2 V, β_F=60.

Hint: Sweep V_{IN} from -5 V to 0 V.

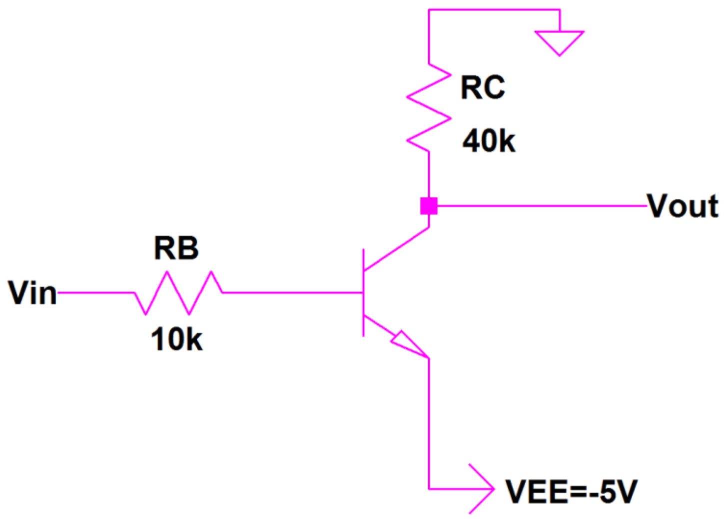

Fig. 3: BJT inverter

6) Consider BJT inverter circuits in Fig. 4 and estimate the circuit parameters that will affect rise and fall times of a basic BJT inverter. If the BJT is turned on in any of the following stages, you can assume that it stays in forward-active region throughout the transitions.

Also, assume that the voltage changes will be roughly linear during the transitions. While this is a gross assumption and the actual rise & fall times will be different from estimated ones; it is nonetheless an instructive exercise to learn how and why rise and fall times will change.

i) What is the operating state of the BJT during rise time? Assume that the input signal changes instantly to low value, right before the transition.

ii) Continuing from the first step, what will the rise time depend on?

iii) Now, determine the operating state of the BJT during fall time. Again, assume that the input switches instantaneously to high value, right before transition.

iv) From part iii, determine the parameters that fall time will depend on.

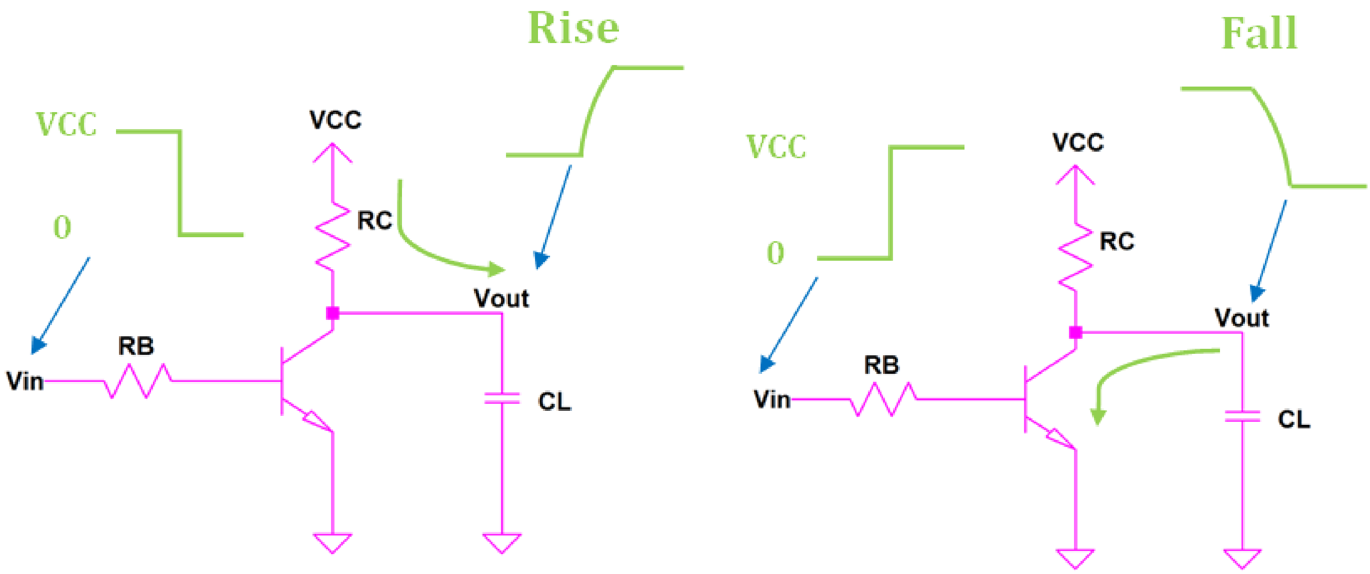

Fig. 4: Schematic showing the rise and fall of the output voltage for a basic BJT inverter

7) Consider the NMOS inverter circuit in Fig. 5, where K=1 mA/V^2, V_t=1 V for the NMOS transistor. Sketch the expected **VTC** by calculating the output voltage V_{OUT} for V_{IN}=0, 1, 2, 3, 4, and 5 V. Find also **noise margins**.

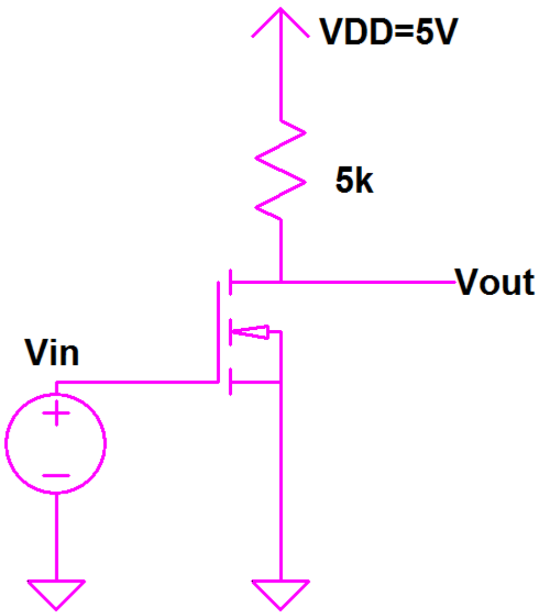

Fig. 5: NMOS inverter circuit schematic

8) Now, consider the circuit in Fig. 6 and think about the rise and fall times of this inverter circuit when a square wave signal is applied at the input. Try to answer the following questions qualitatively by explaining your reasoning.

- **i)** Do you think that the rise time of the circuit depends on the magnitude of the supply voltage, V_{DD}?
- **ii)** Does the rise time depend on the values of the drain resistor and the capacitance at the output?
- **iii)** Does the rise time increase or decrease, if a resistor is connected in parallel to the capacitance at the output?
- **iv)** What are the parameters likely to determine the fall time?

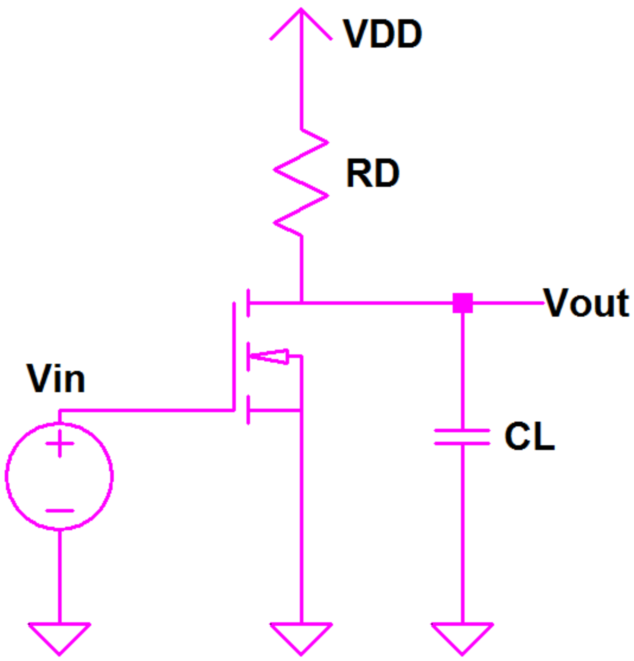

Fig. 6: NMOS inverter circuit schematic

9) Read the document **Introduction to Agilent VEE** and proceed to the next question.

10) Using VEE, generate a **1kHz sine** wave with **0 DC offset**, **0 phase angle**, **2 milliseconds time span** and **256 sample points** using **Function Generator** object. Then use this generator to display three sine waveforms with 0, 2 and 4 V DC offset in the same figure in 0-2 milliseconds time range. When you run the program, you should observe the waveforms as in Fig. 7. **Include the screenshots to show that you fulfilled this task fully and completely.**

Hint: You can use Function Generator, Formula, For Range, Real 64 Allocate Array, and X vs Y Plot objects.

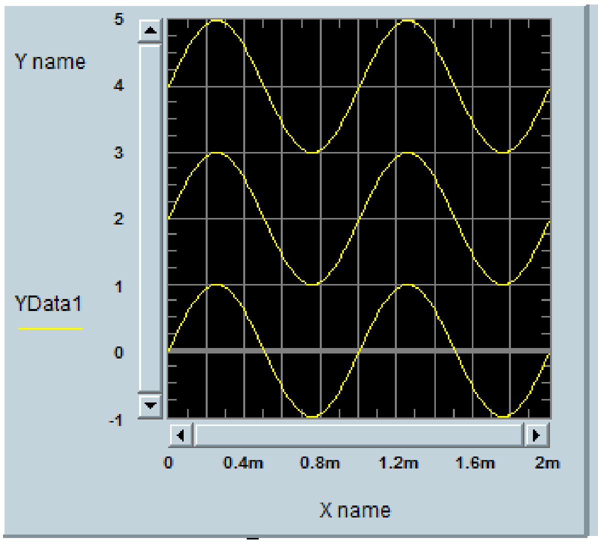

Fig. 7: The output of the VEE program

III. Experimental Work

a) Resistor Loaded NMOS Inverter

In this part, we will characterize the inverter circuits constructed with the n-channel MOSFETs in the CD4007 MOS array chip. The chip includes 3 n-channel and 3 p-channel enhancement mode MOSFETs as shown in Fig. 8. The K and V_t values for the n-channel MOSFETs are measured to be approximately 0.5 mA/V^2 and 1.5 V, respectively under zero substrate reverse bias.

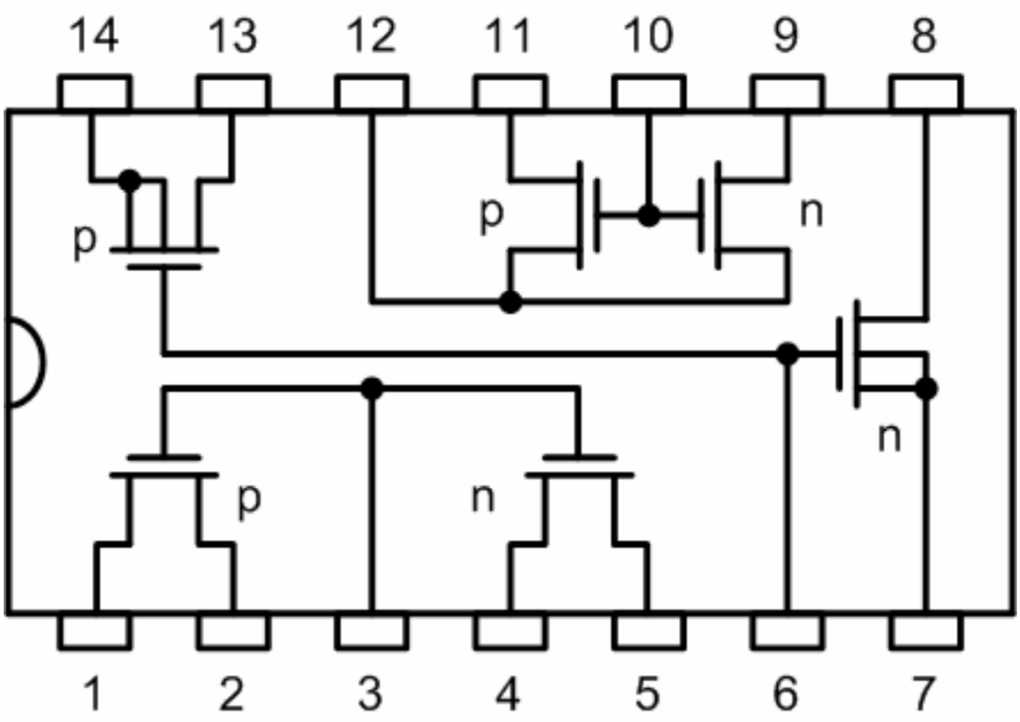

Fig. 8: Pin diagram for CD4007 MOS array chip

a) i) Displaying Voltage Transfer Characteristics with VEE

1) Construct the circuit shown in Fig. 9. Note that **pin 14** of **CD4007** is connected to **+25 V output** of the power supply and **pin 7** is connected to **ground**.

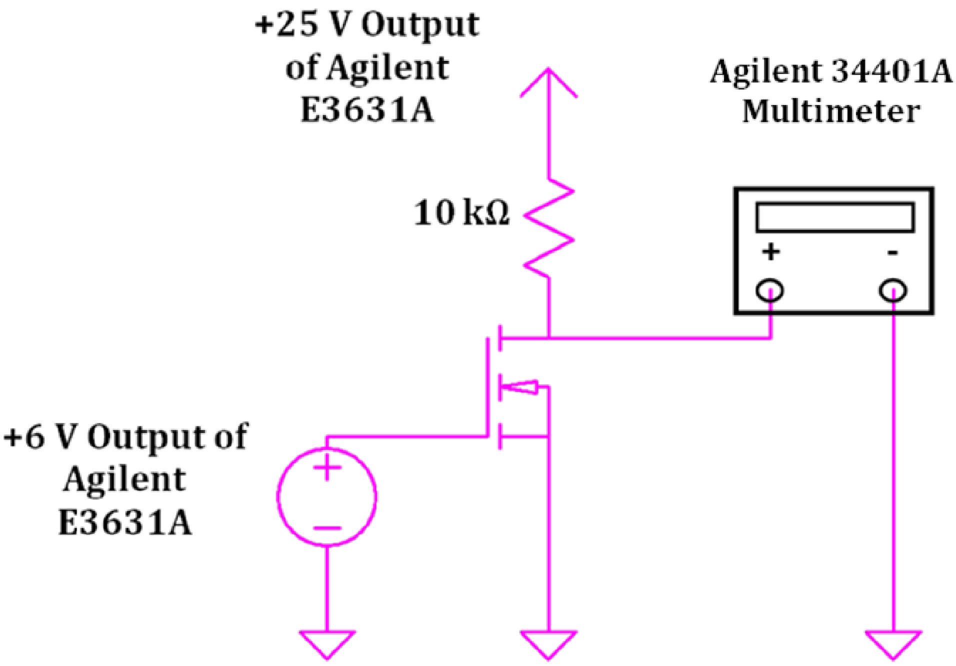

Fig. 9: Circuit Schematic for the resistor loaded NMOS inverter

2) Create a directory on your desktop with the name of Exp2.

3) Double click on the VEE Pro 9.3 icon on your desktop.

4) Move the mouse cursor on **Instrument Manager** at the right side of the work area. You will see an opened window. Click **Find** button to see the connected instruments as shown in Fig. 10. Click **OK** in all opened windows.

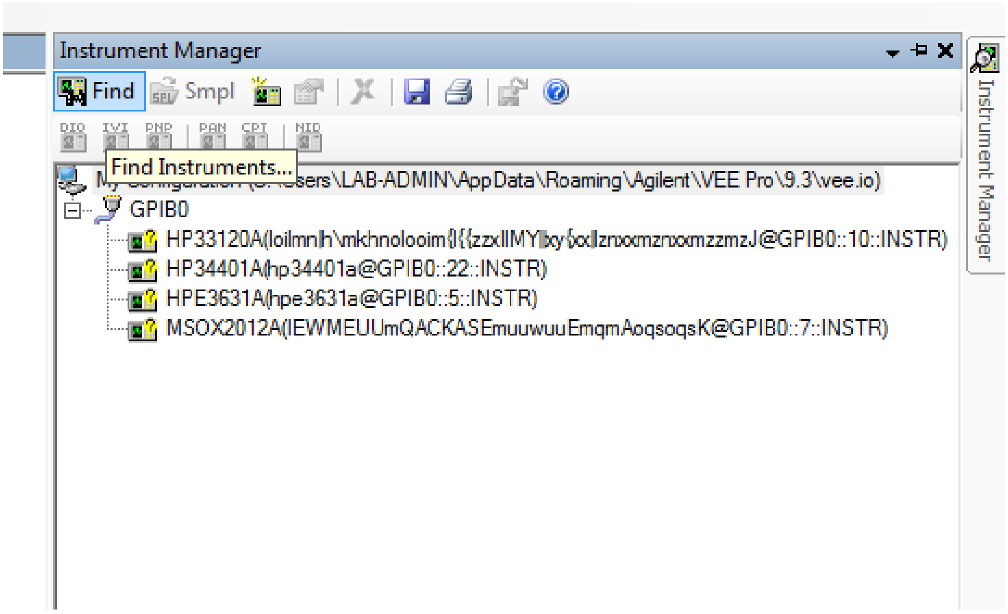

Fig. 10: Finding connected instruments

5) Click on **HPE3631A** (DC Power Supply) and **IVI** button to create IVI-COM Driver Object as shown in Fig. 11. Place the driver in the work area.

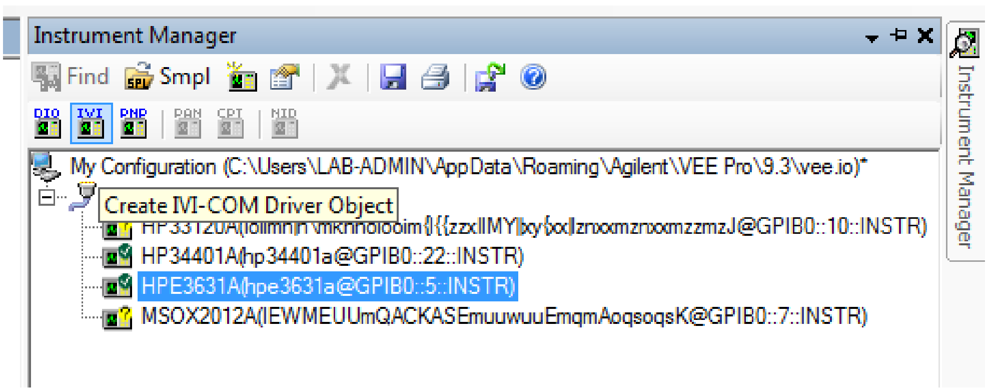

Fig. 11: Creating IVI-COM driver object for Agilent E3631A

6) The IVI driver object for Agilent E3631A is shown in Fig. 12. This driver will control the **+25 V (V_{DD})** output of the power supply.

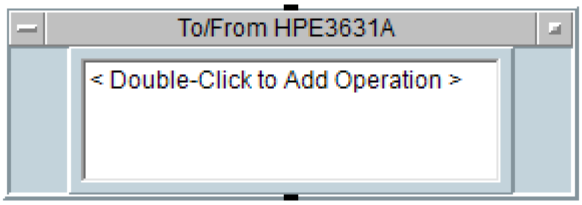

Fig. 12: IVI driver object for Agilent E3631A

7) Click on the driver and change its title as **"VDD"** from the **Properties** menu at the left side as shown in Fig. 13.

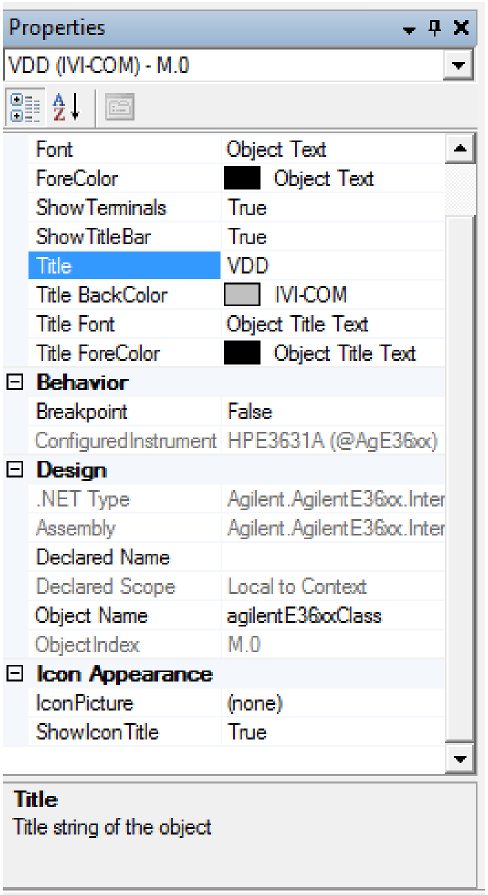

Fig. 13: Changing the title of the IVI driver object

8) Double click on the **<Double-Click to Add Operation>** bar of the **HPE3631A IVI Driver**. Now you get the dialog box which you will use to perform the desired task with the instrument. Select **CreateInstance** and Click **OK**. Click **OK** in the opened **Edit "CreateInstance"** window.

9) Double click on the **<Double-Click to Add Operation>** bar of the **HPE3631A IVI Driver**. Select **Initialize** and Click **OK**. Set **Reset** value to **False** from the drop-down menu as shown in Fig. 14 and click **OK**.

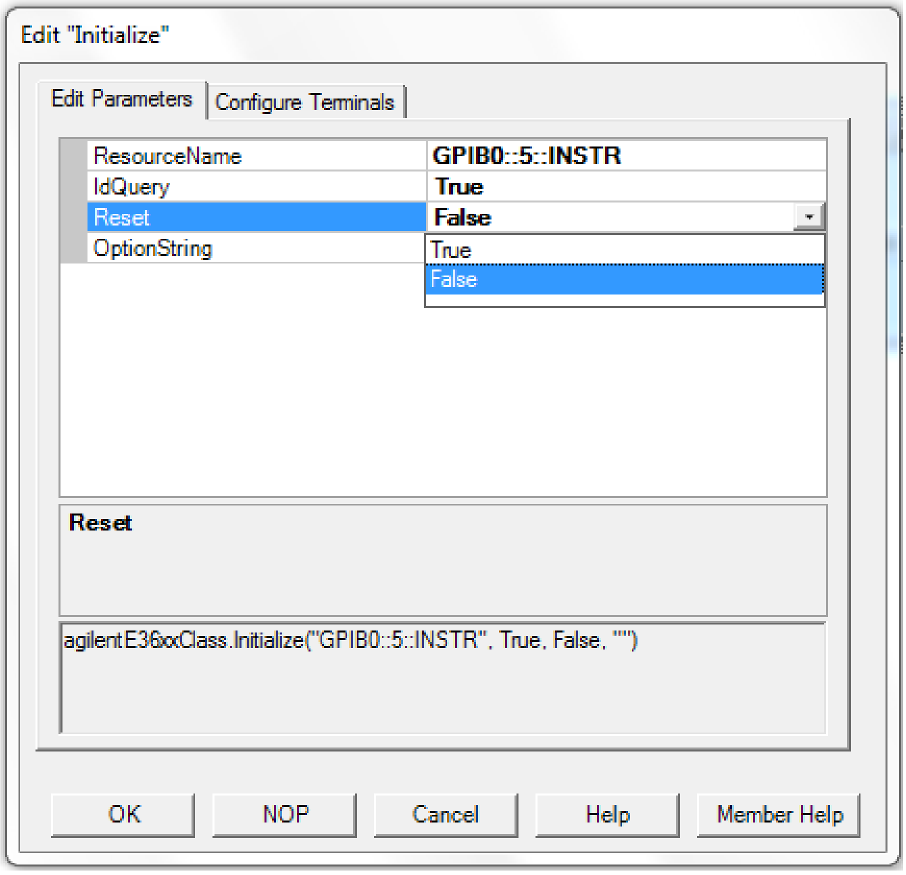

Fig. 14: Initializing IVI driver object for Agilent E3631A

10) Double click on the **<Double-Click to Add Operation>** bar of the **HPE3631A IVI Driver**. Select **Outputs → Item(Name) → ApplyVoltageCurrent** as shown in Fig. 15 and click **OK**.

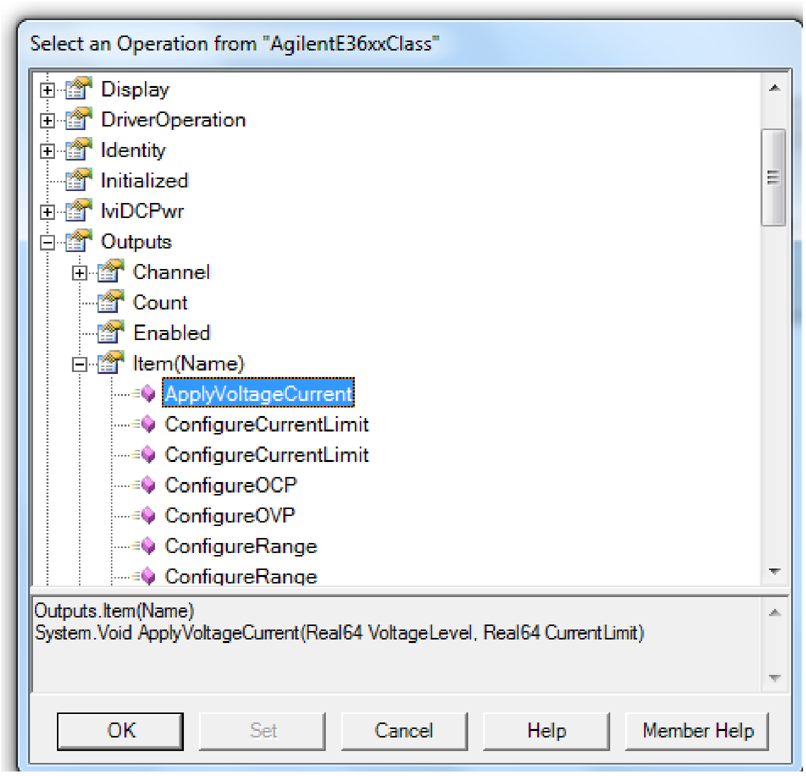

Fig. 15: Selecting ApplyVoltageCurrent for IVI driver object

11) In the opened window, under **Edit Parameters,** enter **output2** into **Name** field as shown in Fig. 16. Note that **output1**, **output2** and **output3** are the reserved names for 6 V, +25 V, -25 V outputs of the DC power supply, respectively.

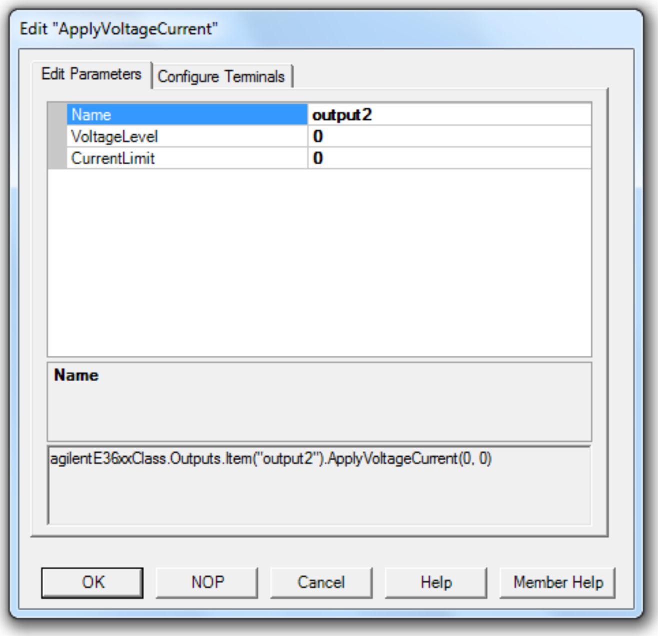

Fig. 16: Editing parameters of IVI driver object for Agilent E3631A

12) In order to create input terminals, click on **Configure Terminals** tab and then on **variable** and **Create input terminal** under **VoltageLeve**l and **CurrentLimit** as shown in Fig. 17. Click on **OK**.

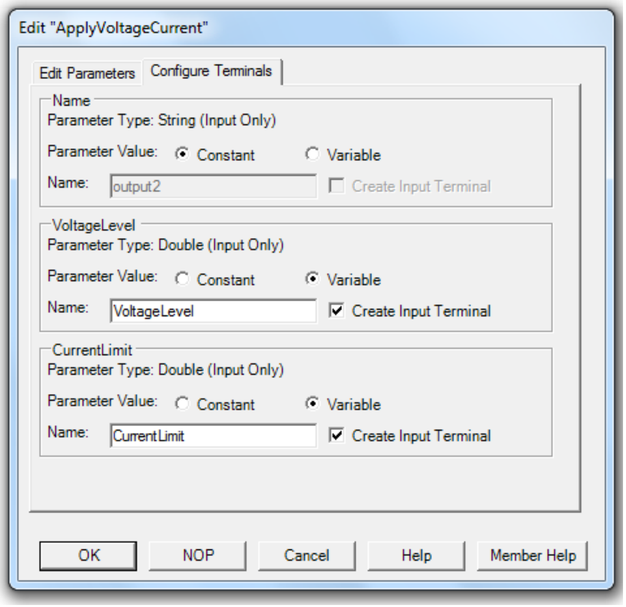

Fig. 17: Creating input terminals for IVI driver object

13) Now, input terminals (**VoltageLevel** and **CurrentLimit**) are created on the driver. Double click on the **<Double-Click to Add Operation>** bar of the **HPE3631A IVI Driver**. Select **Close** and click **OK** in the opened window.

14) Select **Menu Bar → Data → Constant → Real64** and place it near to the **CurrentLimit** data input of VDD. Enter its value as **100m** and connect the data output pin of it to **CurrentLimit input** pin of the VDD.

15) Select **Flow → Repeat → For Range** from the menu bar and place it to the left of **VDD**. Connect the **output pin** of the **For Range** object to the **VoltageLevel input** of the **HP E3631A Driver**.

16) Fill the text boxes in For Range object as shown in Fig. 18. (From=5, Thru=10, Step=2.5). We will obtain VTC of the NMOS inverter for V_{DD}=5, 7.5 and 10 V.

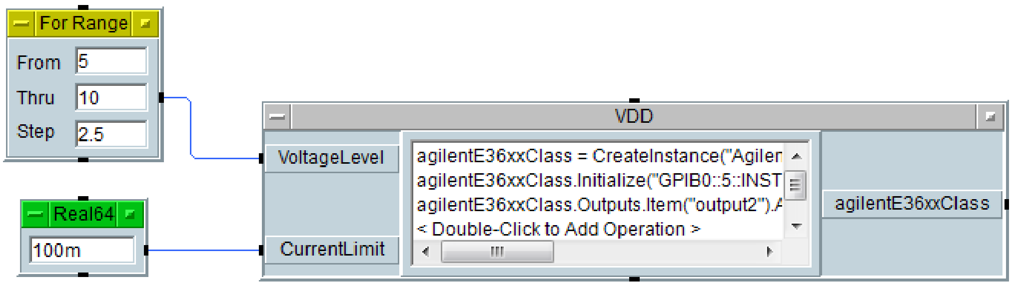

Fig. 18: Connecting For Range and Real64 Constant objects to IVI driver object

17) We will use another IVI driver for **Agilent E3631A** to control the **+6V (VGS)** output of the power supply. Place another IVI driver object for Agilent E3631A following the same steps before.

18) Change the title of the new driver object as **VGS.**

19) Double click on the **<Double-Click to Add Operation>** bar of the **VGS.** Select **CreateInstance** and Click **OK.** Click **OK** in the opened **Edit "CreateInstance"** window.

20) Double click on the **<Double-Click to Add Operation>** bar of the **VGS.** Select **Initialize** and Click OK. Set **Reset** value to **False** from the drop-down menu and click **OK.**

21) Double click on the **<Double-Click to Add Operation>** bar of the **VGS.** Select **Outputs → Item(Name) → ApplyVoltageCurrent** and click **OK.**

22) In the opened window, under **Edit Parameters,** enter **output1** into **Name** field.

23) In order to create input terminals, click on **Configure Terminals** tab and then on **variable** and **Create input terminal** under **VoltageLeve**l and **CurrentLimit**. Click on **OK**.

24) Double click on the **<Double-Click to Add Operation>** bar of the **VGS.** Select **Close** and click **OK** in the opened window.

25) Select **Menu Bar → Data → Constant → Real64** and place it near to the CurrentLimit data input of VGS. Enter its value as **10m** and connect the data output pin of it to the **CurrentLimit input** pin of the VGS.

26) Select **Flow** → **Repeat** → **For Range** from the menu bar and place it to the left of **VGS**. Connect the **output pin** of the **For Range** object to the **VoltageLevel input** of the **VGS**.

27) Fill the text boxes in For Range object as shown in Fig. 19. (From=0, Thru=6, Step=0.2).

28) We will sweep V_{GS} from 0 to 6 V for each V_{DD} value (5 V, 7.5 V, and 10 V) in order to obtain VTC. For this purpose, connect the **data output pin** of the **For Range** object controlling **VDD** to the **sequence input pin** of the **For Range** object controlling **VGS** as shown in Fig. 19.

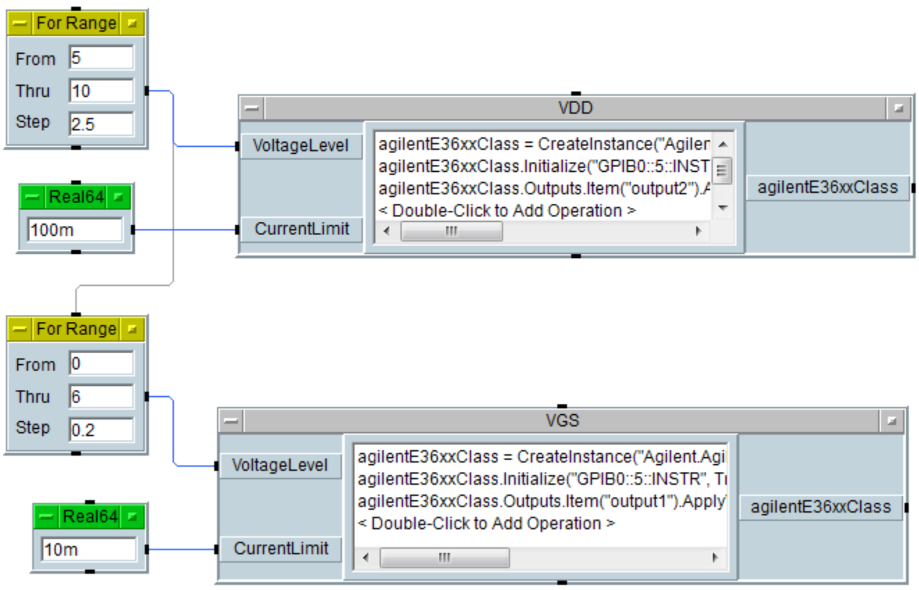

Fig. 19: Connecting For Range objects

29) In order to measure the output voltage for each V_{GS} value, we need an IVI driver object for **Agilent 34401A** (digital multimeter). From **Instrument Manager**, click on **HP34401A** and **IVI** button to create IVI-COM Driver Object. Place the driver in the work area.

30) Change the title of the driver object as **voltmeter**.

31) Double click on the **<Double-Click to Add Operation>** bar of the **voltmeter**. Now you get the dialog box which you will use to perform the desired task with the instrument. Select **CreateInstance** and Click **OK**. Click **OK** in the opened **Edit "CreateInstance"** window.

32) Double click on the **<Double-Click to Add Operation>** bar of the **voltmeter**. Select **Initialize** and Click **OK**. Enter **simulate=false** into the **OptionString** field as shown in Fig. 20 and click **OK**.

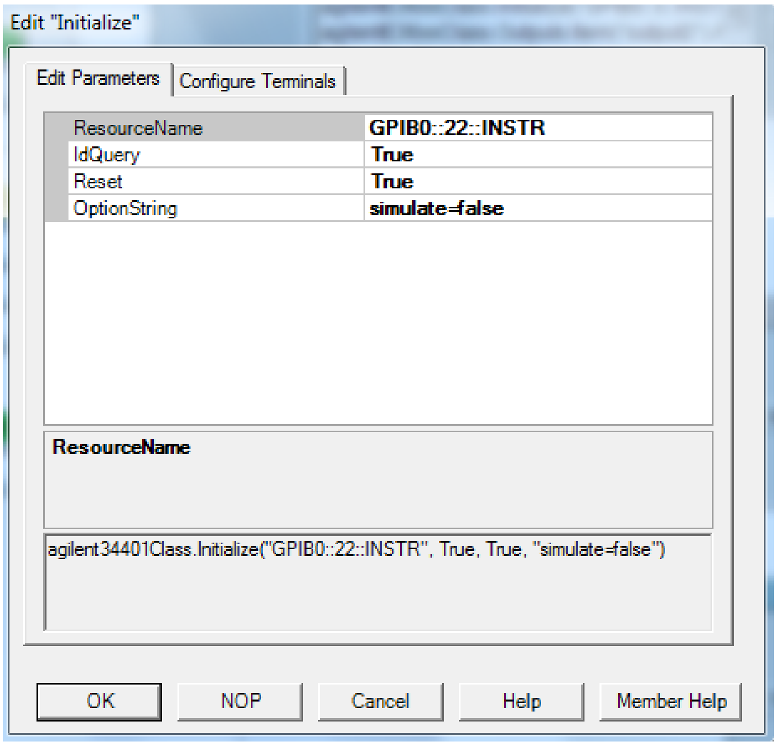

Fig. 20: Initializing IVI driver object for Agilent 34401A

33) Double click on the **<Double-Click to Add Operation>** bar of the **voltmeter**. Select **Measurement → Read** and click **OK**.

34) Enter **100** into the **MaxTimeMilliseconds** area as shown in Fig. 21 and click **OK.**

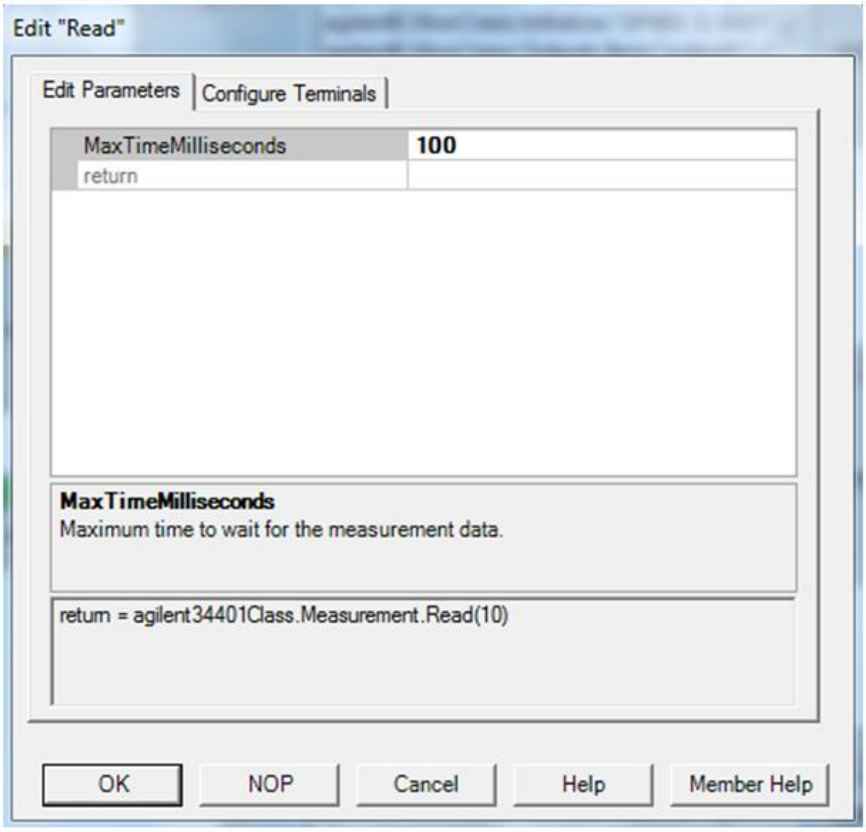

Fig. 21: Adjusting maximum time to wait for the measurement data

35) Double click on the **<Double-Click to Add Operation>** bar of the **voltmeter**. Select **Close** and click **OK** in the opened window.

36) Connect the **sequence output pin** of **VGS** to the **sequence input pin** of the **voltmeter** as shown in Fig. 22. This will ensure that the multimeter will read the output voltage after V_{GS} is set.

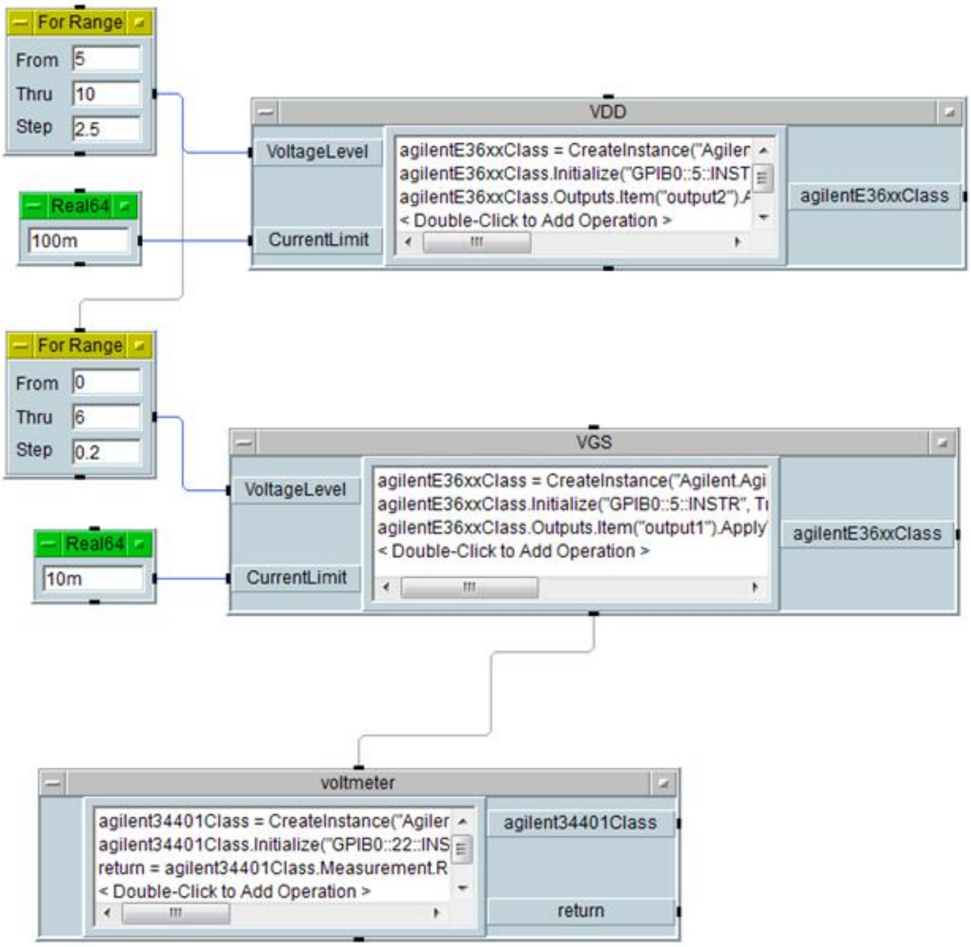

Fig. 22: Connecting Delay object

37) Select **Menu Bar → Display** → **X vs Y Plot**. Place the **X vs Y Plot** object in the work area. Click on the object, and then change the value of **ShowTerminals** under **Appearance** (located @ **Properties** window) to **True**.

38) Double click on **XData** in the input terminal area. Name the terminal **Vin**. Double click on **YData1** and name the terminal **Vout**.

39) Connect the **data output** of the **For Range** object controlling **VGS** to the **Vin input** of the **X vs Y Plot** object.

40) Connect the **return** output of the **voltmeter** to the **Vout input** of the **X vs Y Plot** object.

41) Click on the **X vs Y Plot** object with **right mouse** button and choose **Add Terminal** → **Control Input** → **Autoscale** → **OK**. Connect the **sequence output pin** of the **For Range** object controlling **VDD** to the **Auto Scale input** of the **X vs Y Plot** object. This will ensure that the plot is autoscaled when the program ends.

42) Click on the **X vs Y Plot** object. Set **NumMarkers** as **Delta** under **Markers** (located @ **Properties** window).

43) Click on **Vout** on the **X vs Y Plot** object. Select color, line type and point type as **Green, Point** and **Diamond**, respectively. At the end, the VEE configuration should look like as in Fig. 23.

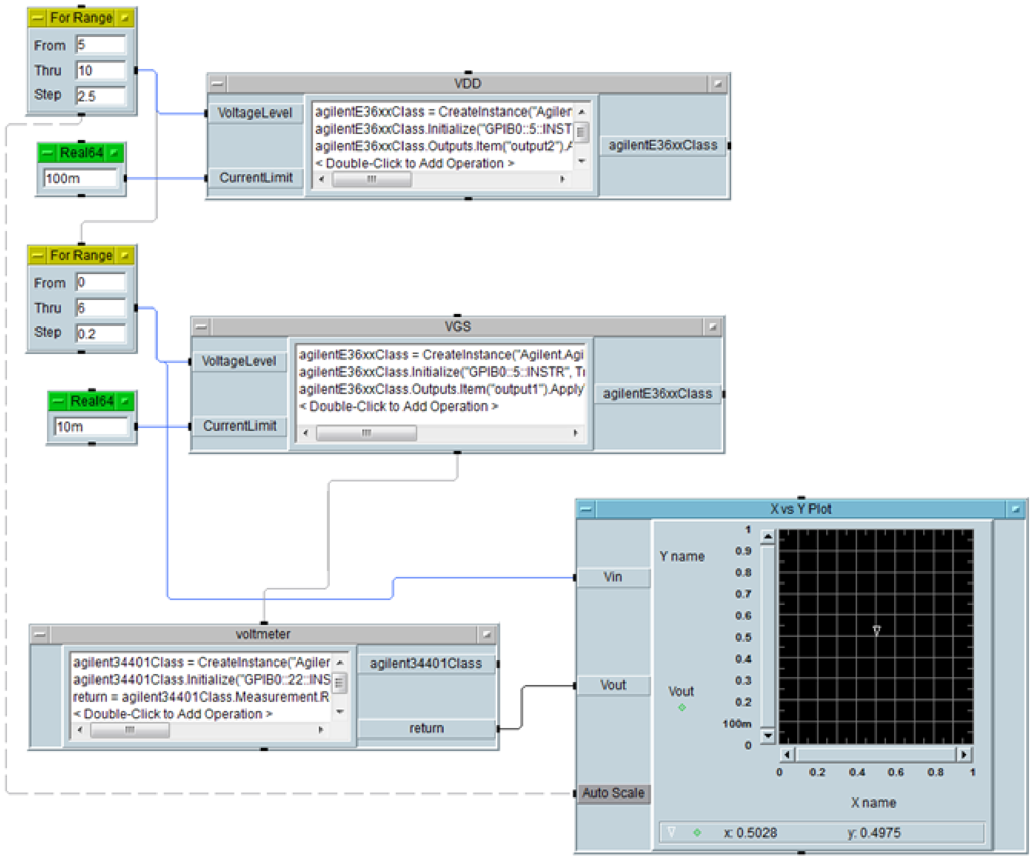

Fig. 23: Final VEE configuration

44) Please be sure that **DC power supply** is turned **ON** and the **voltage limits** of +6 V and +25 V outputs are larger than +5 V and +10 V, respectively. In addition, the multimeter should be in **DC voltage measuring mode**.

45) Now, you can run your program by clicking **Run** button below Menu Bar. Wait until the program is completed and observe the transfer characteristics on X vs Y Plot.

46) Find the following voltage levels when V_{DD}=5 V, 7.5 and 10 V (Use the markers and the zoom facility).

- V_{OH}= output high voltage
- V_{OL}=output low voltage
- V_{IH}= minimum input voltage which provides a low output voltage
- V_{IL}=maximum input voltage which provides a high output voltage

47) Calculate the noise margin of the inverter for V_{DD}=5 V, 7.5 and 10 V. Which supply voltage is better for protection from extraneous noise voltages that may alter the output level?

48) Estimate the resistance of the MOSFET when it is ON (the output is at the low level).

49) When V_{OL} values for two different supply voltages are compared under the same gate voltage, V_{OL} seems to be larger for higher V_{DD}. Why?

50) Now connect a 10 kΩ resistor (R_L) between the drain of the MOSFET and ground. Run the Agilent VEE program and observe the transfer characteristics. Compare the noise margins with those obtained without R_L. Comment on the results.

a) ii) Dynamic Response

In this part of the experiment, we will investigate the dependence of the dynamic response of the inverter on load capacitance, load resistance and supply voltage.

1) Construct the circuit shown in Fig. 24.

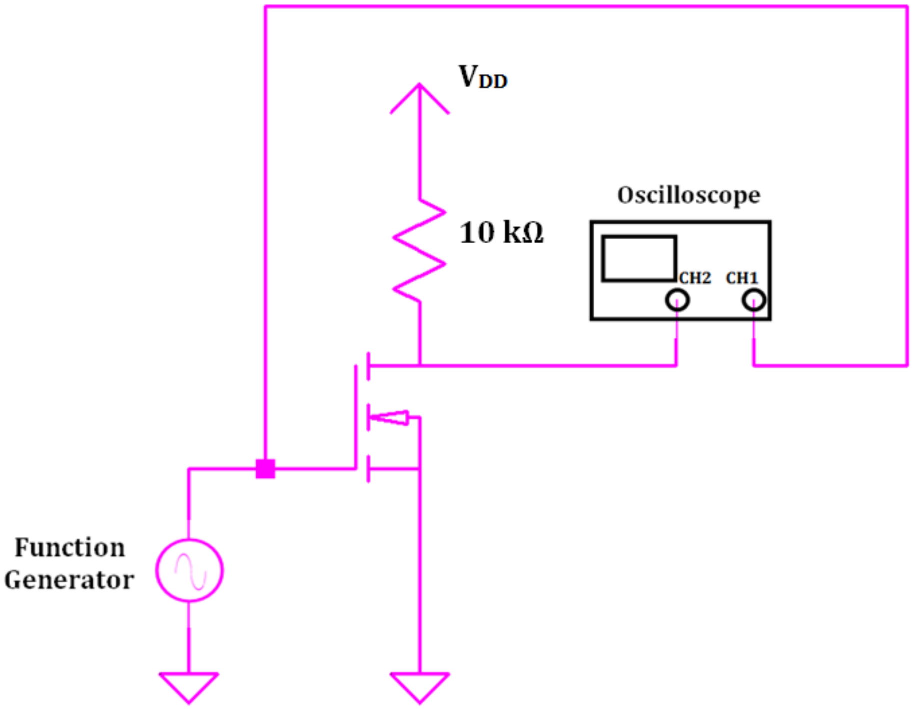

Fig. 24: Circuit schematic for the dynamic response

2) Set the output impedance of the function generator to high Z. Set the waveform to a square wave with 5 V_{PP} amplitude, 2.5 V offset and 1 kHz frequency. Set **V_{DD}=5 V**.

3) Measure and record the **rise** and **fall times** of the inverter.

4) Measure and record the **propagation delay times, t_{PHL}** and **t_{PLH}** for the inverter.

5) Connect a **100 pF capacitor between the drain of the MOSFET** and **ground**. Measure and record the **rise** and **fall times** and **compare** the results with those measured **without the capacitor**. **Comment** on the results.

6) Now, **disconnect** the **capacitor**. Measure and record the rise and fall times under **V_{DD}=7.5 V** and **V_{DD}=10 V**. **Compare** the results with those taken under **V_{DD}= 5 V**. **Comment** on the results.

7) The **rise time** does **not change significantly** with increasing **supply voltage**. **Why not?**

8) Decrease the **supply voltage** to **5 V**. **Connect** a **10 kΩ resistor between the drain of the MOSFET** and **ground.** Measure the **rise** and **fall times** of the inverter. **Compare** the results with those obtained **without the resistor**. **Why** do the **rise** and **fall times change** when the resistor is connected?

b) BJT Inverter

Construct the BJT inverter circuit in Fig. 25 by using an npn BJT in the **CA3046 BJT** array chip whose pin diagram is given in Fig. 26. Connect **pin 13** to **ground**. Set the output impedance of the function generator to **High Z**. Connect **input** to **channel A1** and **output** to **channel A2** of the scope.

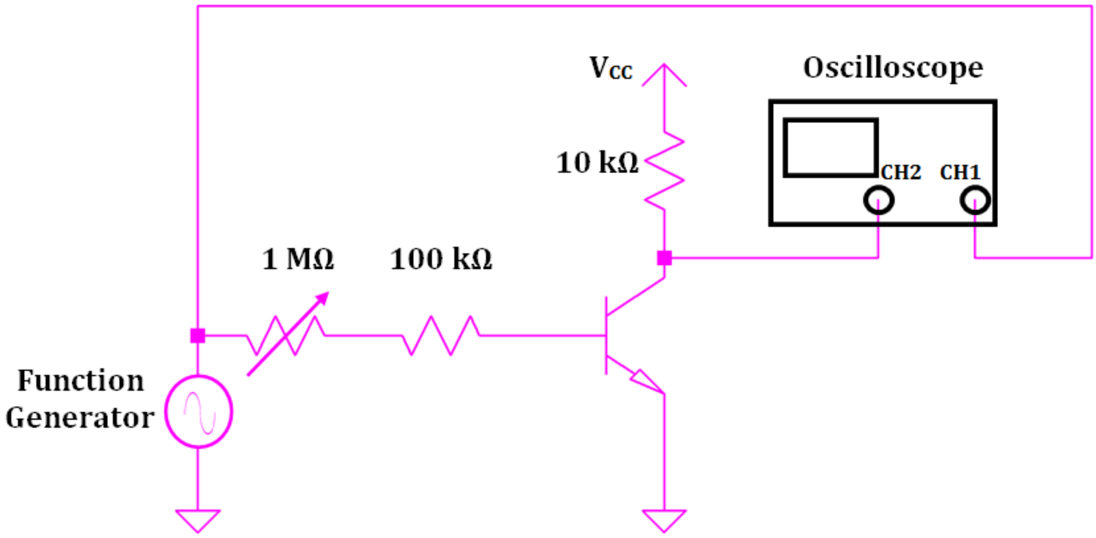

Fig. 25: Circuit schematic for BJT inverter

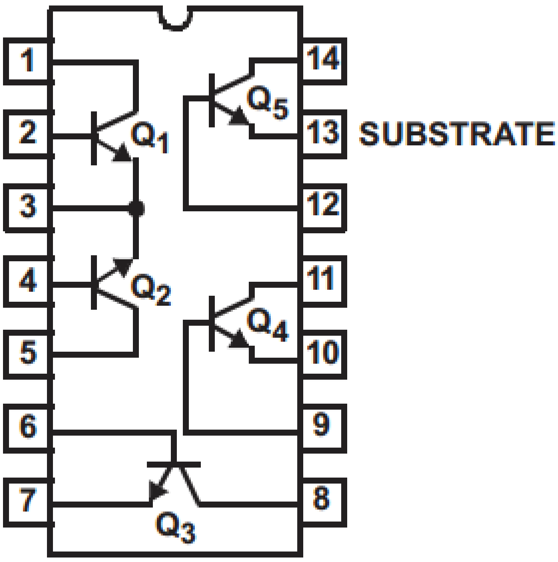

Fig. 26: The pin diagram for CA3046

b) i) Displaying Voltage Transfer Characteristics with the Scope

1) Set the waveform at the function generator output to a **triangle wave** with **5 V_{PP}** amplitude and **2.5 V offset** at a frequency of **1 kHz**. Set **V_{CC}=5 V**.

2) Observe the transfer characteristics of the inverter by using the scope in the **X-Y mode**. Use the potentiometer to **change the base resistance** and observe the change in the transfer characteristics by considering the **noise margins**. **Compare** the **noise margins** with those of the **NMOS inverter**. **Comment** on the results.

b) ii) Dynamic Response

1) Change the waveform at the output of the function generator to a **square wave** and observe the input and output signals on the scope. Set the **potentiometer** to **0 Ω**. Measure and record the **rise** and **fall times**. **Compare** the results with those of the **NMOS inverter**.

2) Now, **increase** the **potentiometer resistance** and observe the change in the **rise** and **fall times**. **Comment** on the results.

3) **Compare** the **voltage transfer characteristics** and the **rise time** of the BJT inverter with that of the NMOS inverter.