DEVELOPING EDUCATIVE CURRICULUM MATERIAL FOR PHYSICAL SCIENCES USING EDA SOFTWARE TO ENHANCE STUDENTS LEARNING

A.J.Chaudhari*

Abstract:

The purpose of this paper is to develop the curriculum material on Basic Electronics. This material support an active learning process, creative participation, and learner engagement. One of the challenges of designing learning materials is that of engaging students. The aim of a learning material is to provide students with challenges related to the main task with prior experience if possible.

In this paper author tried to develop a study material on JFET-characteristics experiment of North Maharashtra University, Jalgaon, as a sample case, using 5Spice software. Further the study is extended to two JFETs connected in parallel. Almost all experiments of UG-Electronics Laboratory can be simulated and carried /send to students across the country. This will be very useful resource material for UG –Students.

Keywords: Learning material, Simulation, JFET, Analysis, 5Spice.

* Dept. of Physics and Electronics, M.J.College , JALGAON- INDIA-425002.

Introduction:

While learning the semiconductors characteristics, we start from diode characteristics and then proceeds to zener diode, bipolar junction transistor, field effect transistor and so on. But author thinks differently. He want to study the characteristics if a silicon diode is connected in series with the germanium diode and then analyze it mathematically and graphically by determining certain important parameters. In this work such innovative situation is considered. Here drain characteristics of two JFETs connected in parallel is studied and analyzed by using 5Spice simulation software. Computer simulations have become a useful part of mathematical modeling of many natural systems in physicals sciences and. Simulations can be used to explore and gain new insights into new technology, and to estimate the performance of systems too complex for analytical solutions. A successful simulation is any simulation which characterize and predict the behavior of real systems. Successfulness depends on the quality of the model used and the computational power available.

5Spice is a general-purpose electronic circuit simulation program with a full graphical user interface. It wraps around a traditional Spice simulation engine, presenting a single application to the user. 5Spice provides Spice specific schematic entry, the ability to define and save an unlimited number of analyses and graph setups, and integrated graphing of simulation results[1]. Other software are LT Spice, Pspice, Tina etc. In this paper drain characteristics of two identical JFETs connected in parallel is studied and compared with single JFET characteristics using 5Spice software. Also JFET parameters are calculated.

5Spice-Computer simulation software is a tool that produces outputs based on particular inputs. It is very useful in circuit design/analysis. Circuit design is a creative and analytical process.5Spice provides the analysis abilities needed by experienced circuit designers while remaining easy-to-use - a big step up from student oriented Spice programs.

Features of 5Spice:

1. Visual representation of concepts visually.
2. Default simulation wizard

3. Lager numbers of analyses and graph plotting ability.
4. Library contains many parts and sub circuits.
5. Compatible with Pspice..
6. Sweep ability for components and temperature.
7. Node numbering is not required to consider while drawing or analyze.
8. Support for analysis of alternative designs
9. Ability of both analog and digital simulations
10. Help menu works as an additional manual.
11. Behavioral modeling
12. Schematic symbol with user defined equation (equation may contain IF-THEN-ELSE structures, math functions, etc.)
13. A notes section in the schematic and one in each analysis allows users to document their work.Theoritical things can be stored in Note(Source-5Spice-manual)

Methodology: The methodology used is shown bellow in Fig.1

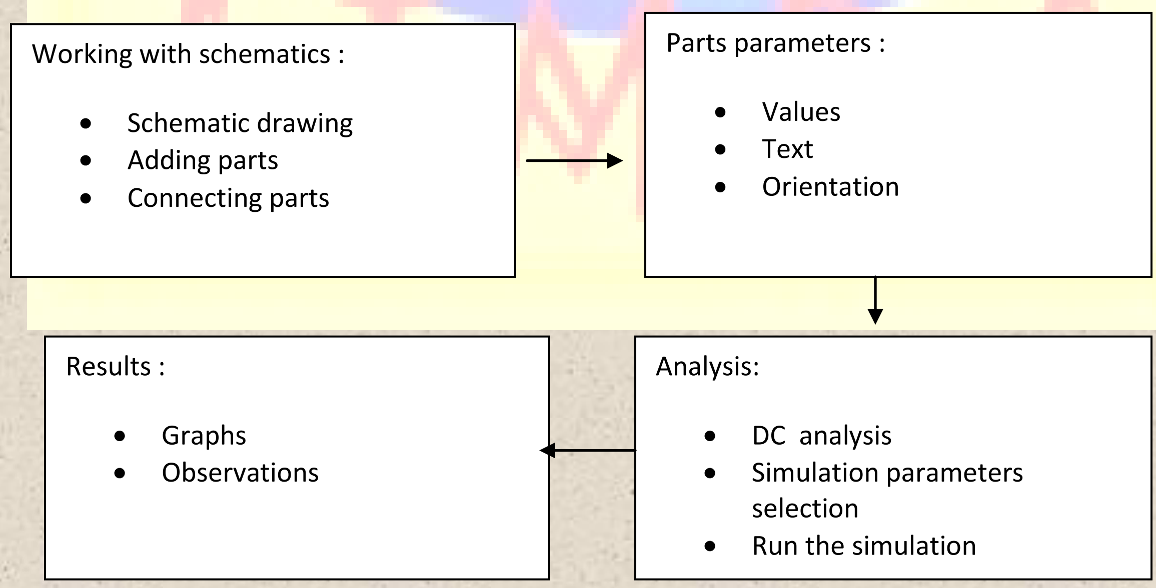

Fig 1-Methodological steps

The simulation, in general, involves following sequential steps[2]

=.> New Schematic----Opens clear area/canvas

=.> Drag parts

=.> Drag test point/s-----Test points in the circuit

=.> Connect all ……. Rig out the circuit

=.> Change attributes , if any

=.> Analyze

=.> Select/Edit

=.> Analysis dialog----select analysis

=.> Graph/Table

=.> Plots/Axis………..selects plot and axis with test point/s

=.> Legend……….names the X-axis and Y-axis

=.> Apply changes

=.> Apply and Run

=.> Output

In this work DC-Analysis of JFETs was studied and graphs were plotted using the software. The software window is shown bellow in Fig.2

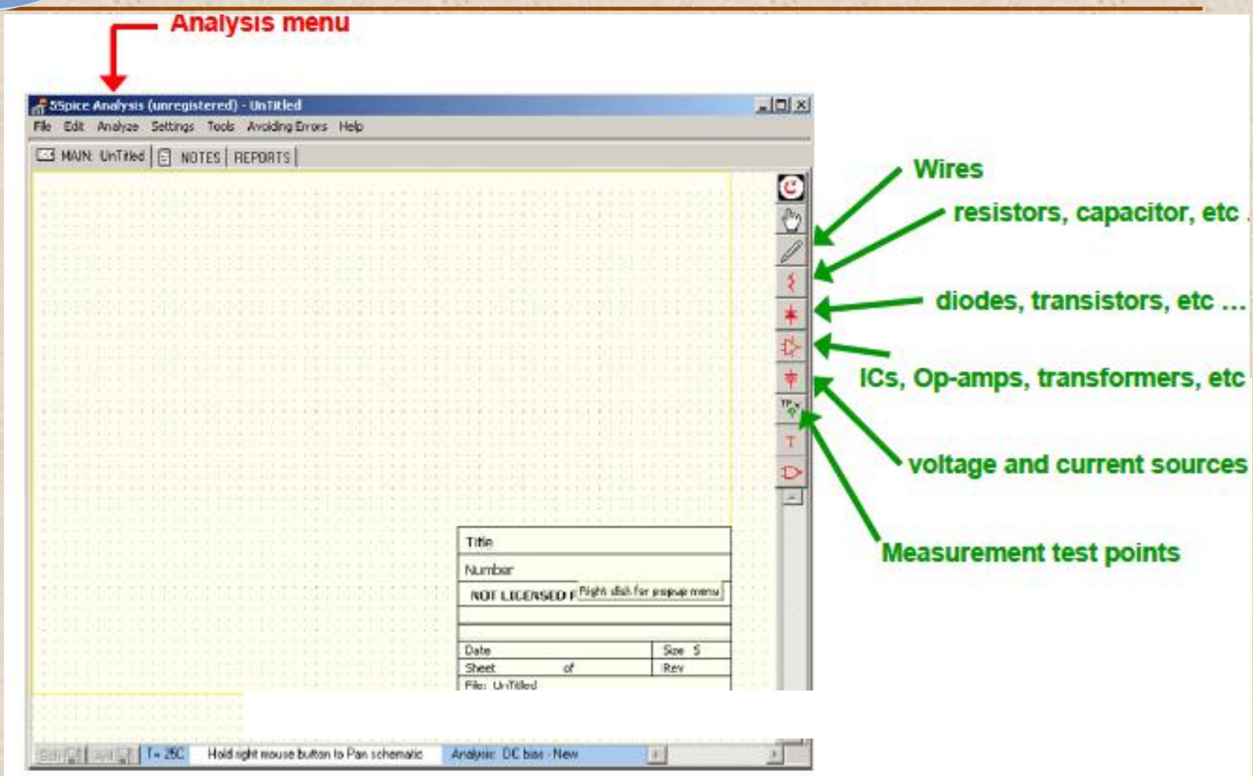

Fig 2. (Window of 5spice)

Proper DC conditions were selected and graphs were plotted using Graphs option of the analysis window[3].

1.Drain Characteristics for a single JFET:----The spice model is shown in Fig.3 bellow:

Schematic:

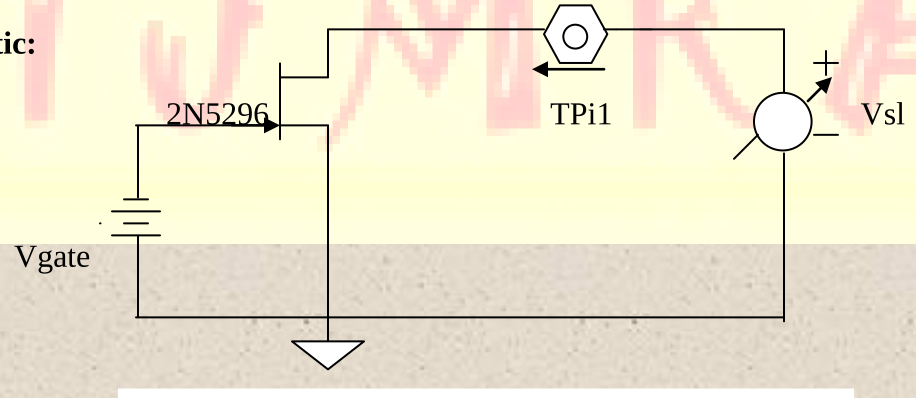

Fig 3- (Spice model for single JFET)

For single FET simulation, following steps were performed.

=.> New Schematic----Opens clear area/canvas

=.> Drag parts…………B1,Q1,Vs1,G

=.> Drag test point/s-----TPi1

=.> Connect all ……. Rig out the circuit

=.> Change attributes ….B1=3,Q1=2N5196,Vs1=0 to 5 ,step=0.25

=.> Analyze

=.> Select/Edit

=.> Analysis dialog----………DC New

=.> Graph/Table…….Graph

=.> Plots/Axis ………Plot-Axis-Left-TPi1

=.> Legend………..Optional

=.> Apply changes

=.> Apply and Run

=.> Graph of Drain current Vs Drain source voltage.

The simulated circuit is shown in Fig.4

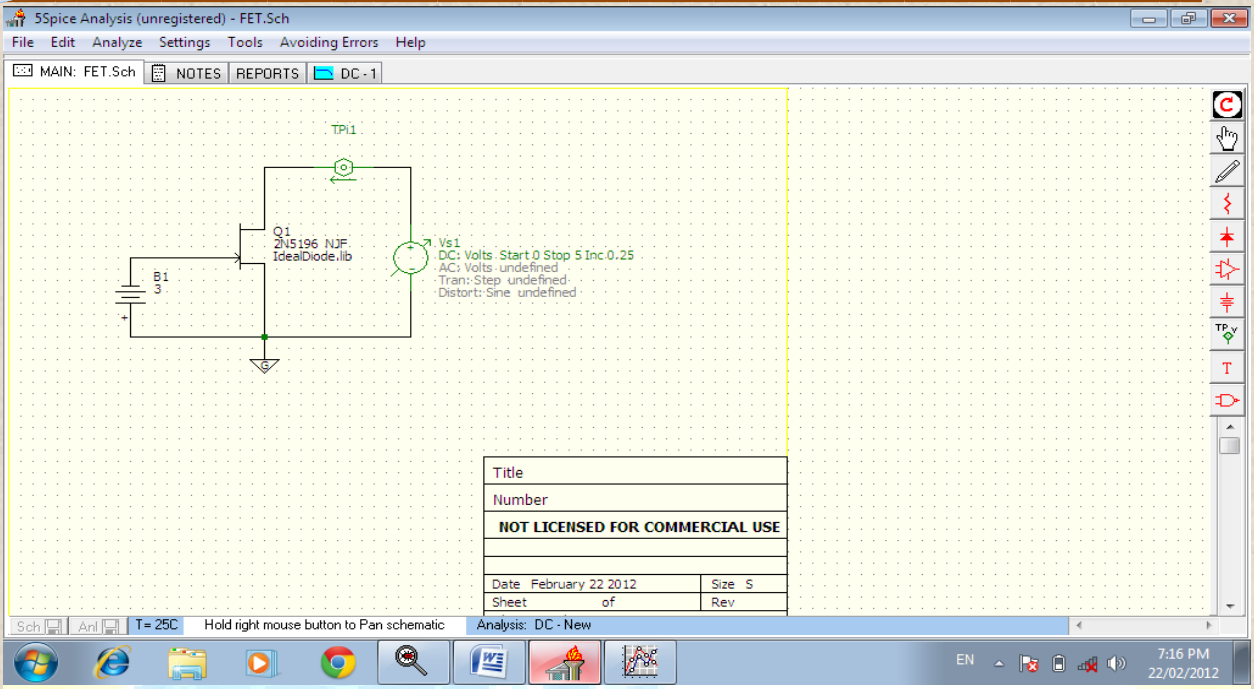

Fig. 4 (Single FET-Circuit)

Analysis setup and graph setup is shown in Fig.5 and Fig. 6 respectively.

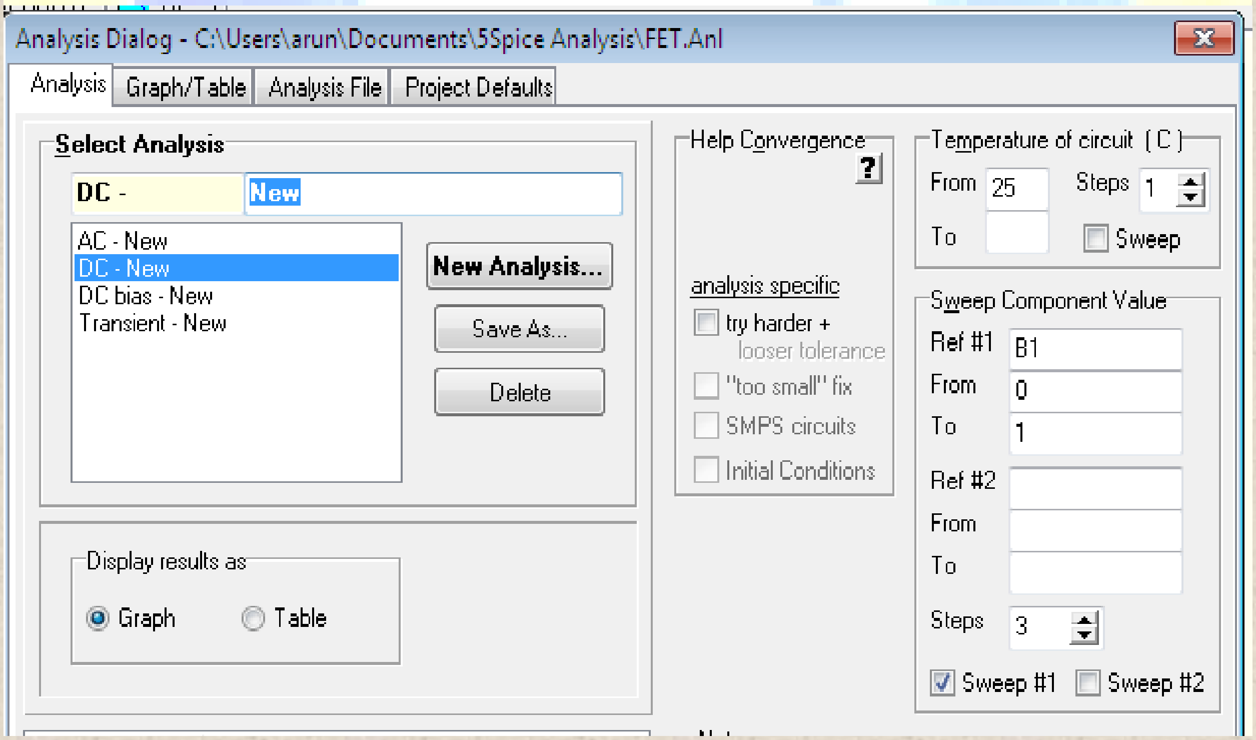

Fig. 5 (Analysis dialog box)

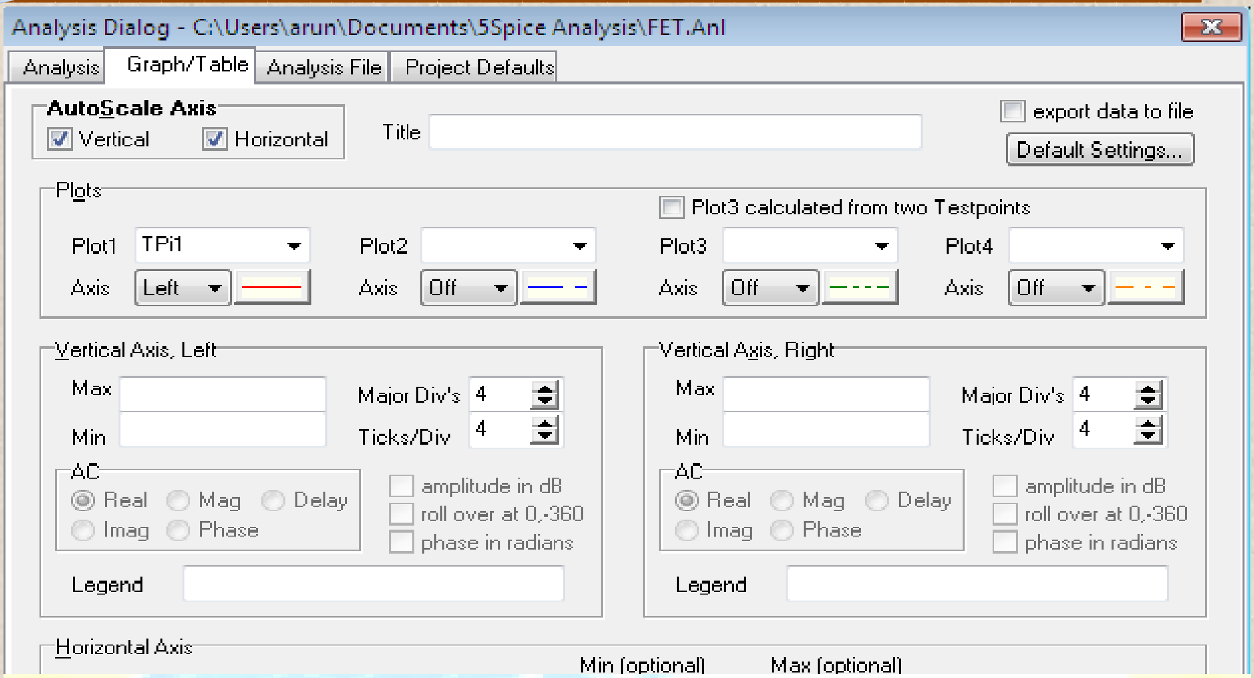

Fig. 6 (Graph/Table option box)

The IV-Characteristics of single FET after simulation is shown bellow in Fig.7

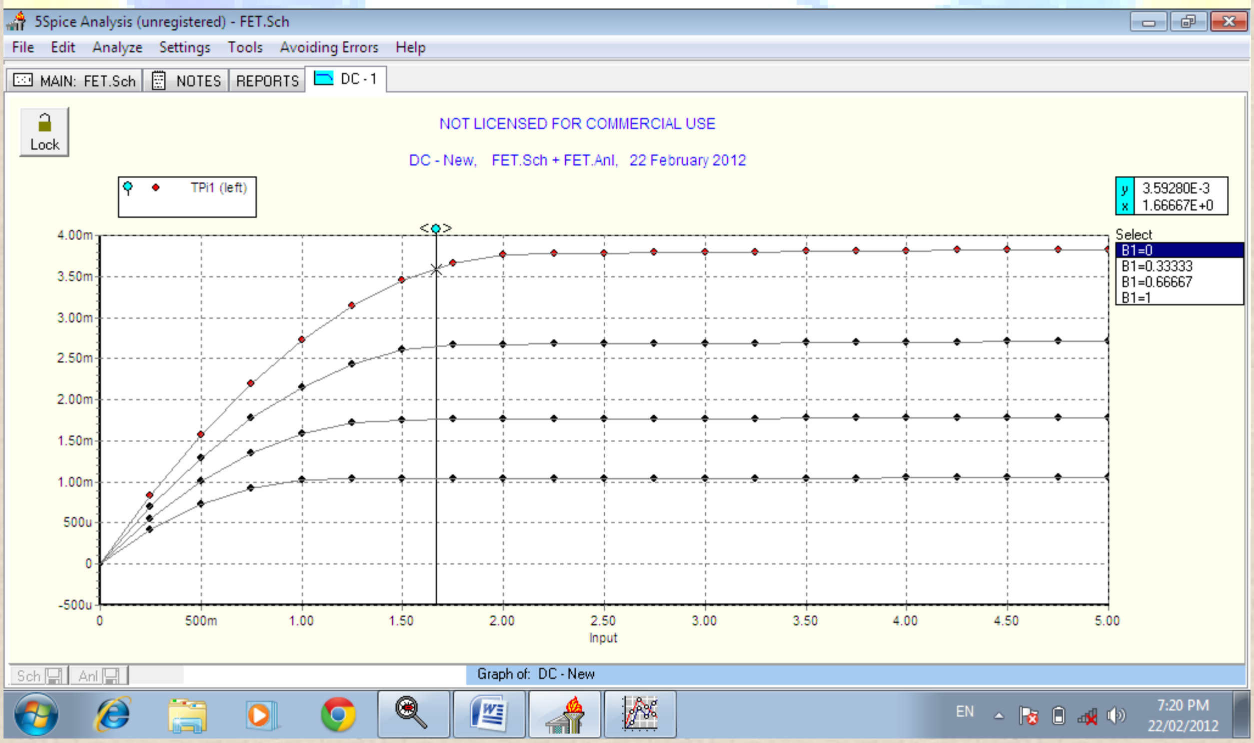

Fig. 7 (IV-Charactersics for single FET)

This is the drain characteristics for Vgs = 0 , 0.33 , 0.66 and 1 volt as obtained by using 5Spice simulation software for single JFET.Clearly the FET current was 4mA maximum.This simulation was performed on 22February2012 and was indicated by sofware at the top.

2. Drain Characteristics for two JFETs connected in parallel--:----The spice model is shown in Fig.8 bellow:

Schematic:

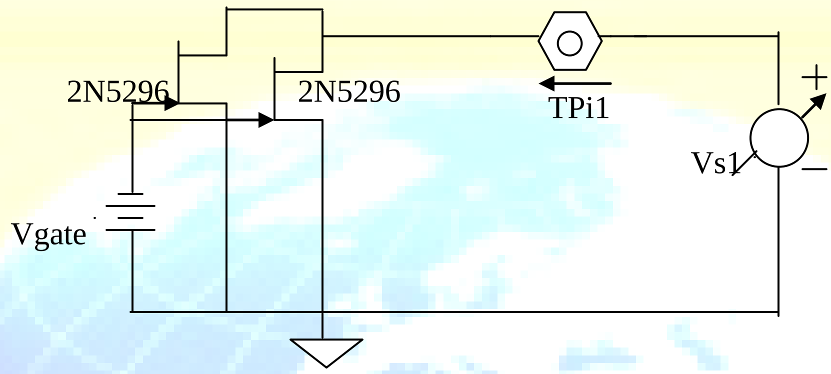

Fig 8-Spice model for characteristics of two JFETs in parallel

The simulated circuit for two FETs connected in parallel is shown in Fig. 9

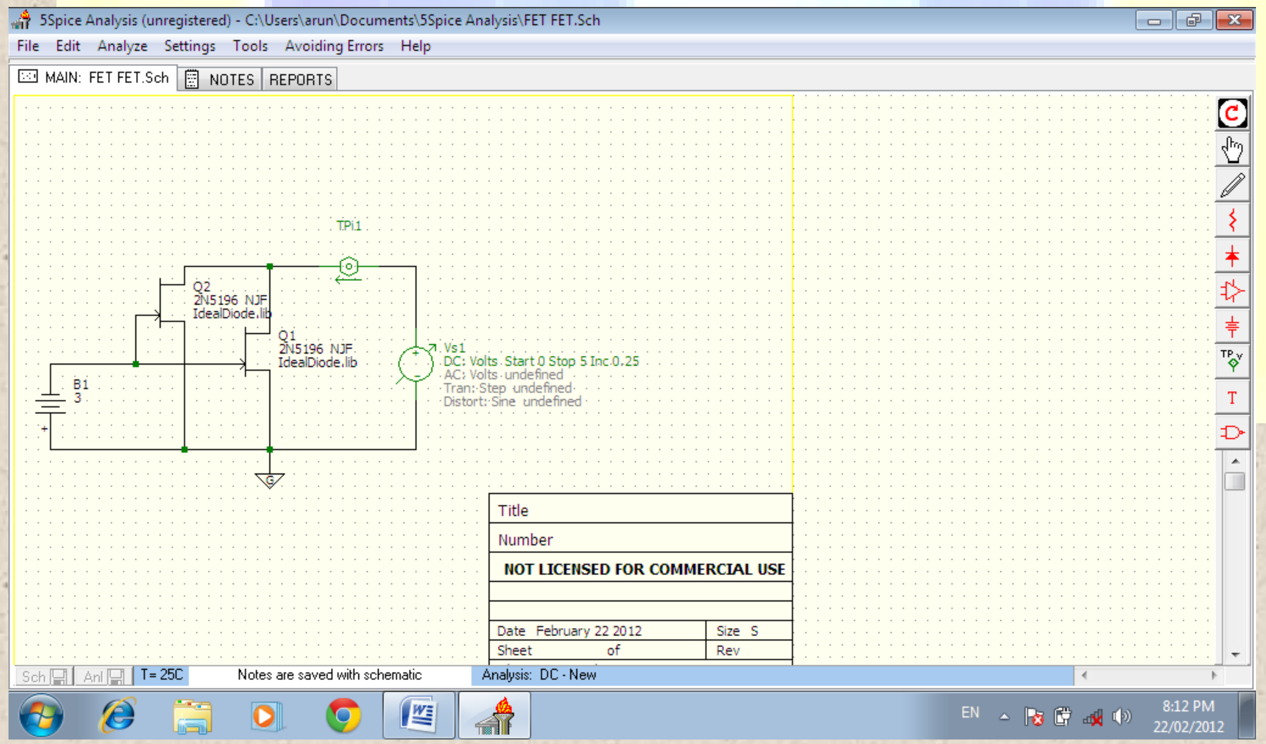

Fig.9 (Two FET-Circuit)

The drain characteristics(for the same gate voltages) as obtained by using Spice simulation software for two JFETs connected in parallel is shown bellow in Fig. 10

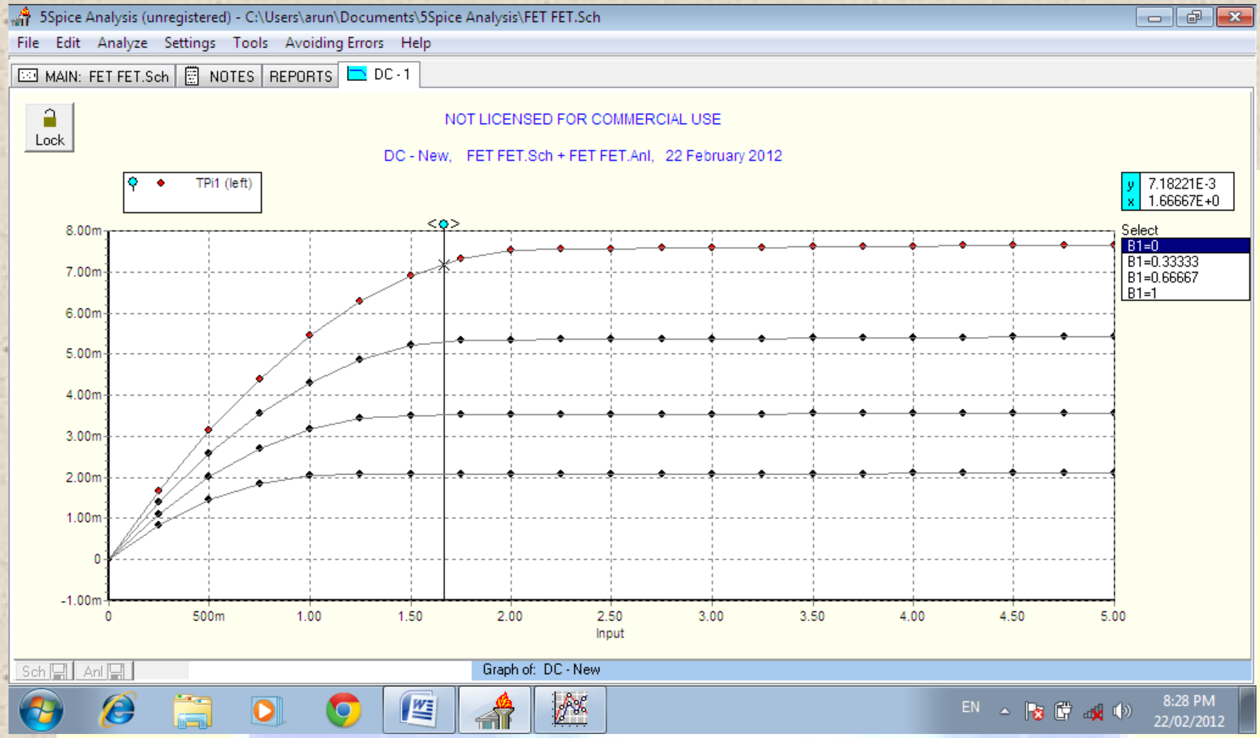

Fig.10 (IV-Charactersics for two FETs)

Clearly the drain current increased two times to approximately 8mA.

Values of **rd , gm** and μ are calculated from the graphs and tabulated as follows :

For Single JFET (2N5196)

The calculated values of rd , μ and gm are given below in tables 1 to 3

TABLE 1

Constant Parameter(value)	ΔId mA	ΔVds Volt	rd=ΔVds/ΔId KΩ
Vgs = 0 V	0.05	2	40

TABLE 2

Constant Parameter(value)	ΔVds Volt	ΔVgs Volt	μ=ΔVds/ΔVgs
Id = 2 mA	0.23	0.33	0.696

TABLE3

Constant Parameter(value)	ΔId mA	ΔVgs Volt	gm=ΔId/ΔVgs mA/V
Vds = 2 Volt	1.09	0.33	3.3

For two identical JFETs(2N5196) - in parallel:

The calculated values of rd , μ and gm are given below in tables 4 to 6

TABLE 4

Constant Parameter(value)	ΔId mA	ΔVds Volt	rd=ΔVds/ΔId KΩ
Vgs = 0 V	0.10	2	20

TABLE 5

Constant Parameter(value)	ΔVds Volt	ΔVgs Volt	μ=ΔVds/ΔVgs
Id = 4 mA	0.22	0.33	0.666

TABLE 6

Constant Parameter(value)	ΔId mA	ΔVgs Volt	gm=ΔId/ΔVgs mA/V
Vds = 2 Volt	2.18	0.33	6.6

Analysis :

The drain current is expressed as

Id = gm Vgs + (1/rd)Vds …………………………..(i)

where gm = ΔId/ΔVgs = tranceconductance

rd = ΔVds/ΔId = drain or output resistance

Now consider two JFETs connected in parallel as shown bellow in Fig.11.

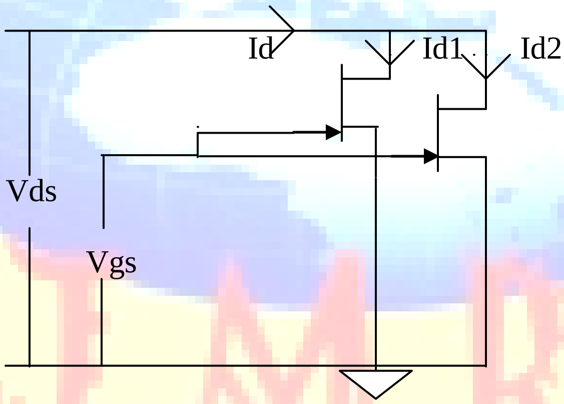

Fig. 11(Two JFETs in parallel)

Considering Vds = 0, applying equation (i) to Fig.11

Id = Id1 + Id2

= gm1 Vgs + gm2 Vgs

= (gm1 + gm2) Vgs

For identical JFETs (gm1 + gm2) = 2gm1 or 2gm2

Thus in this case total tranceconductance of combination is **two times** the individual tranceconductance [4].

Similarly considering Vgs = 0,

$$Id = Id1 + Id2$$

$$= 1/rd1 \times Vds + 1/rd2 \times Vds$$

$$= (1/rd1 + 1/rd2)Vds$$

$$\text{Hence, } Id/Vds = 1/rd = (1/rd1 + 1/rd2)$$

For identical JFETs , total drain resistance of combination is given by

$$1/rd = (1/rd2 + 1/rd2)$$

Hence 1/rd = 2/ rd2 or rd = rd2/2 OR rd = rd1/2

Thus in this case, total drain resistance is **half** the individual drain resistance. The amplification factor ($\mu = rd \times gm$) remain unchanged.

Interactive learning offers following advantages [5, 6]:

1. Student's active role in learning process.
2. Multiplicity situations explored at the user's discretion.
3. The student is not a spectator, but an active participant.
4. Repetitive study at almost no extra cost.
5. Tool is powerful allowing for full animation, realistic three-dimensional images, etc.

Results:

Following things are observed by comparing the drain characteristics:

a) For identical JFETs connected in parallel (gm1 + gm2) = 2gm1 or 2gm2. Thus total tranceconductance of combination is two times the individual tranceconductance.

b) For identical JFETs connected in parallel, total drain resistance of combination is given by rd = rd1/2 or rd2/2. Thus total drain resistance is half the individual drain resistance.

c) The amplification factor for parallel combination (μ = rd × g) remain unchanged.

References:

- 5Spice Anderson Software manual.
- Robert F. Coughlin, F.F.Driscoll, Operational Amplifiers and Linear Integrated Circuits,6th edition, Pearson Education.
- A.J.Chaudhari, M.J.College, Jalgaon(M.S) , Simulations of Innovative Circuit Ideas Using 5Spice Software Research Link- (ISSN-0973-1628), Issue 50(A), Vol: VII(3), May- 2008
- Dharam Raj Cheruku, Battula Tirumala Krishna, Electronic Devices and Circuit First Indian print , 2005 , Pearson Education.
- Reinventing teaching in the classroom of the future: The political science laboratory experience. New York: Penn State University. Retrieved March 28, 1999 from the World Wide Kennedy, C. (1998).

  Web: http://www.personal.psu.edu/faculty/c/l/clk8/reinventteach.html.
- Proceedings of National Seminar on Emerging Trends in Physics Education and Experimental Physics (NSPE-2006)-Tenali-27-28 Oct-2006.