Introduction to Visual TCAD

The tool is design by the **Cogenda semiconductors, Singapore**. As per the tool is used for Device Modeling, Simulation and Characterization of the Device (transistors and diodes) are done using different methods. Analytical method is used for modeling which consists of closed form equations obtained using spice. Experimental method is required for characterization. Numerical methods which are based on the concept of 2D/3D meshing can be adopted for conducting simulation studies which requires Technology CAD. Different TCAD device simulators are MEDICI, SILVACO (ATLAS), SYNOPSIS (SENTAURAS), and **COGENDA (Visual TCAD)**. Among this 2D/3D simulation is possible using COGENDA TCAD.

Cogenda TCAD consists of Genius powerful simulator and Visual TCAD interactive GUI in which 2D structures and 3D structures can be created. Circuit simulation is also available in this tool. Core features are provision for Advanced device modeling, Circuit schematic capturing, easy device simulation control, Versatile device visualization, Syntax highlighted text editor, Spreadsheet and X-Y Plots for 2D along with Z direction if used in 3D. Multiple analysis modes such as DC / Transient / AC simulation and optical simulation are there in the tool itself. Advanced setup such as boundary conditions, material parameters, physical models are also available. By using Visual TCAD, One can gain clearer image of understanding semiconductor devices physics and Microelectronics

2D device structures can be drawn and there are 2 methods for 3D device drawing. Either 2D structure can be drawn and extended to Z Axis or direct 3D structure can be formed using deck (code) format. In extended 3D method, meshes in the Z direction is automatically generated whereas there is provision for defining meshes in X,Y, and Z direction in deck scheme which makes it more accurate and interactive .

Various steps followed for device simulation are:

- Define X,Y, and Z mesh
- Define regions and corresponding materials
- Define doping Profile
- Apply models
- Apply bias
- Export the output file

Using the Graphical User Interface

Visual TCAD is the integrated graphical user interface of the Genius device simulation package.

1. To start Visual TCAD in Linux.
    - Open the terminal or Press **[ctrl+alt+t]**
    - Type Visual TCAD or the corresponding alias name

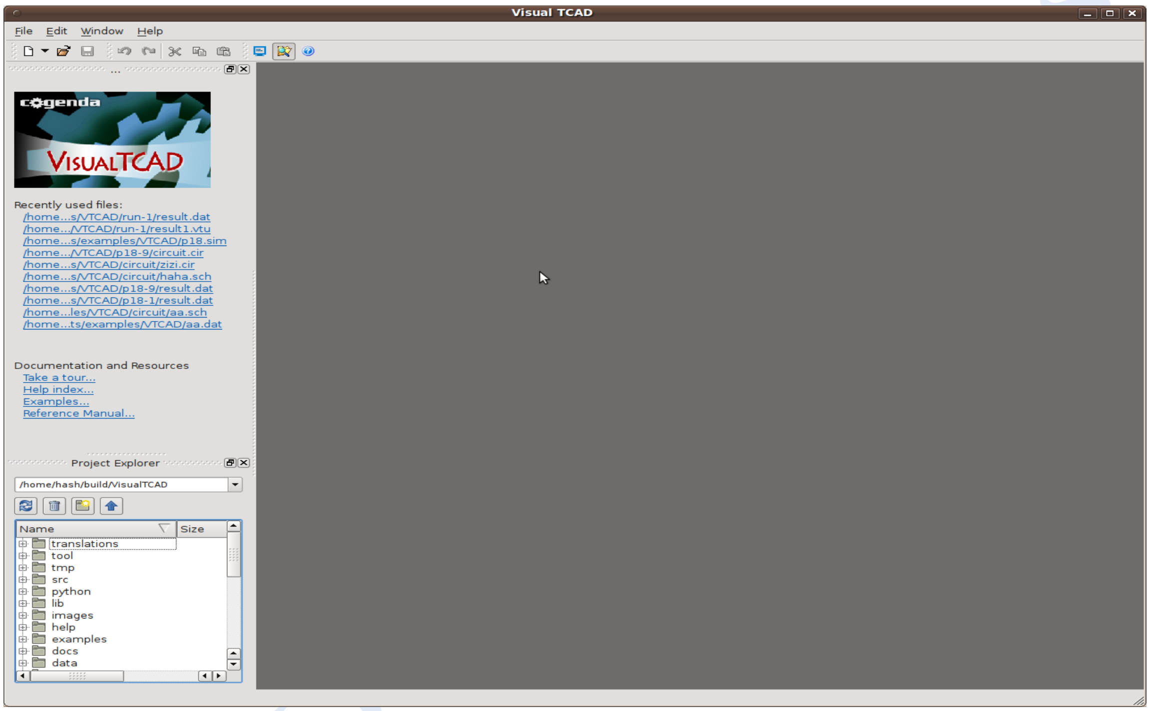


Fig 1: Visual TCAD GUI window

2. To draw new structure.
    - Click on file
    - New file
    - Device drawing
3. To write Deck file (code)
    - Click on file
    - New file
    - Text document

4. To do Device simulation
    - Click on file
    - New file
    - Device simulation
5. To Draw circuit schematic
    - Click file
    - New file
    - Circuit schematic

The version number of the release can be checked by clicking the > menu > Help > about. The version number will be coming in a dialog box as shown below

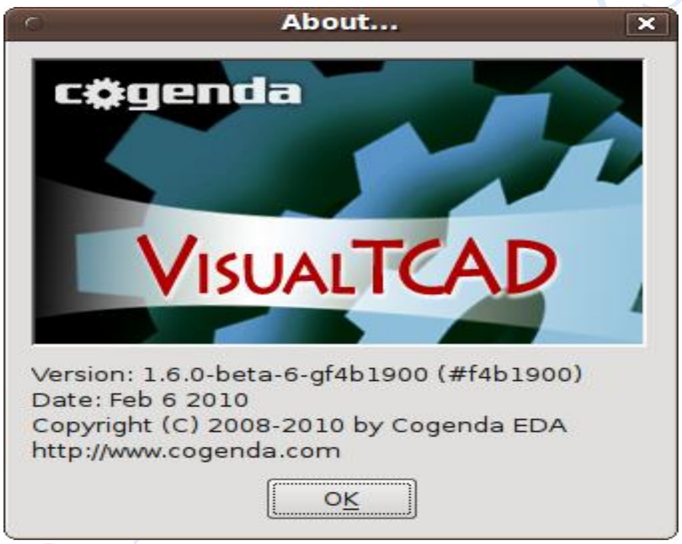


Fig 2: dialogue box showing version number

Features of Visual TCAD

1. Advanced Device Drawing Tools

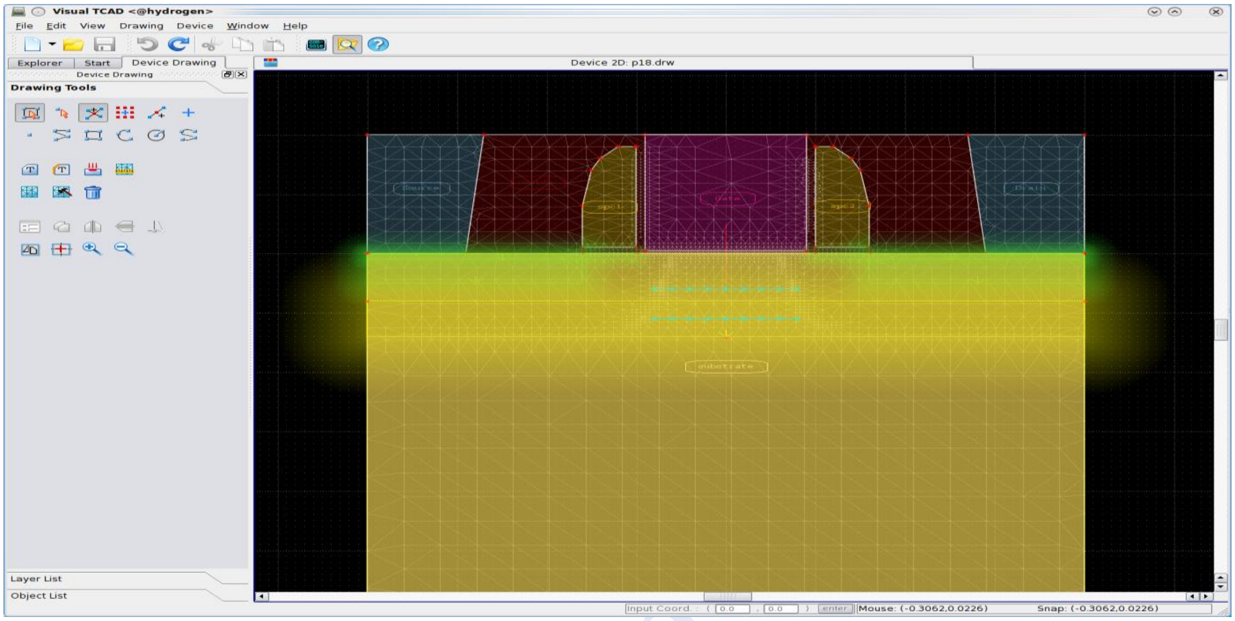


Fig 3. A demo NMOS device with meshed network

2. Easy Simulation Setup

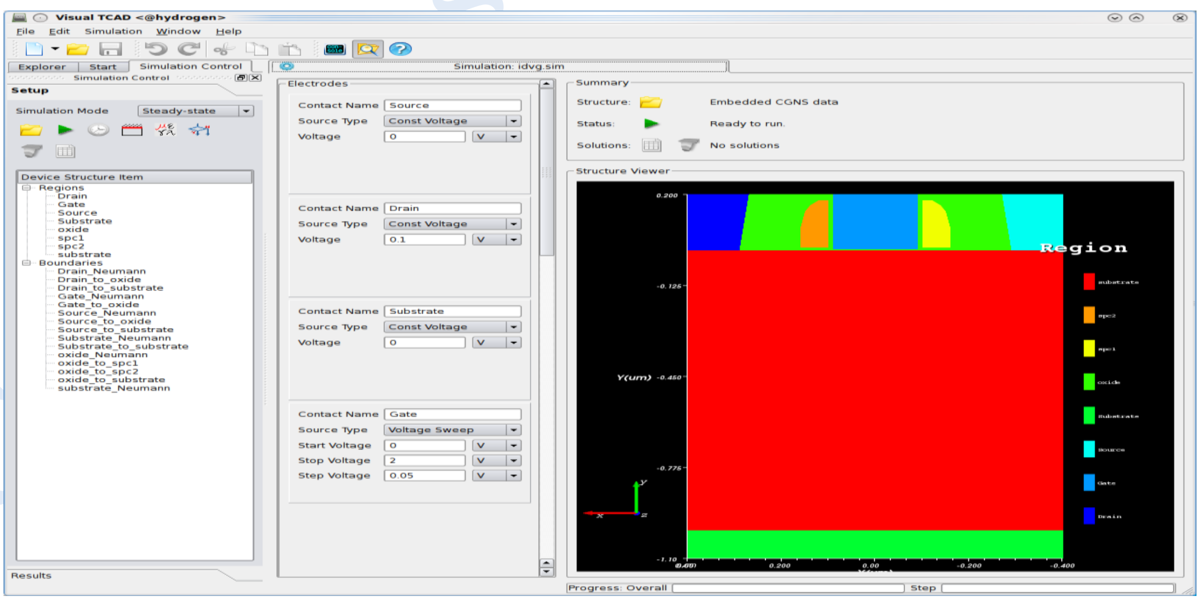


Fig 4. A demo NMOS device with simulation workbench

3. Versatility in Device Visualization

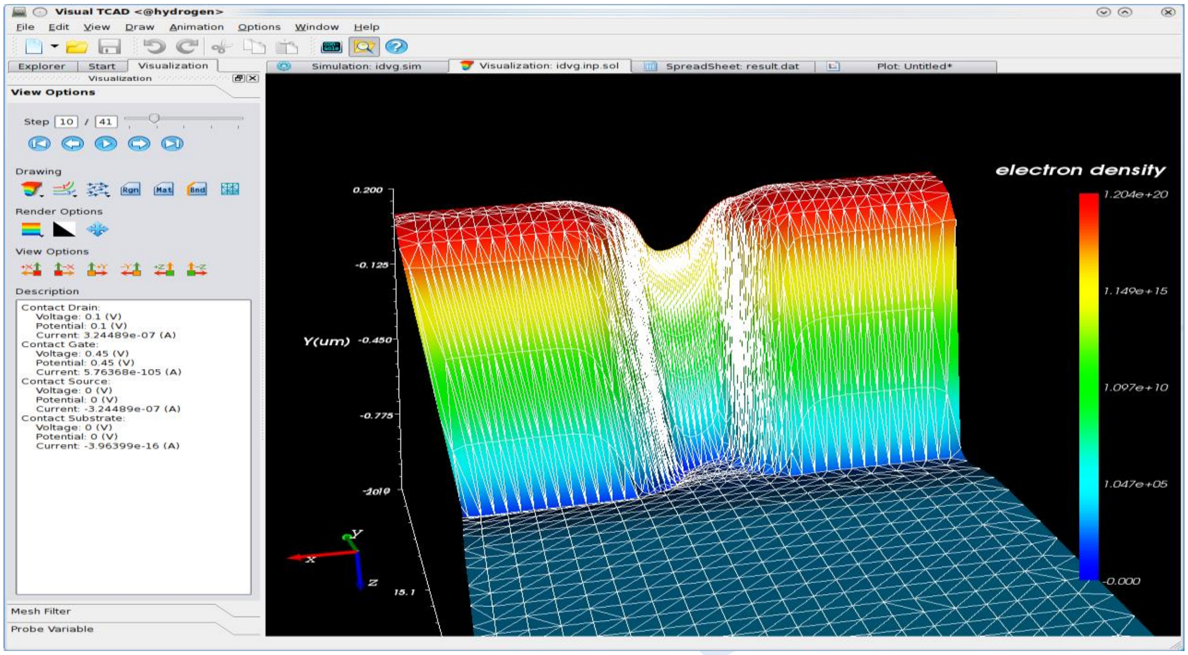


Fig 5.visualization of NMOS demo after simulation

4. Keywords Highlighted Text Editor

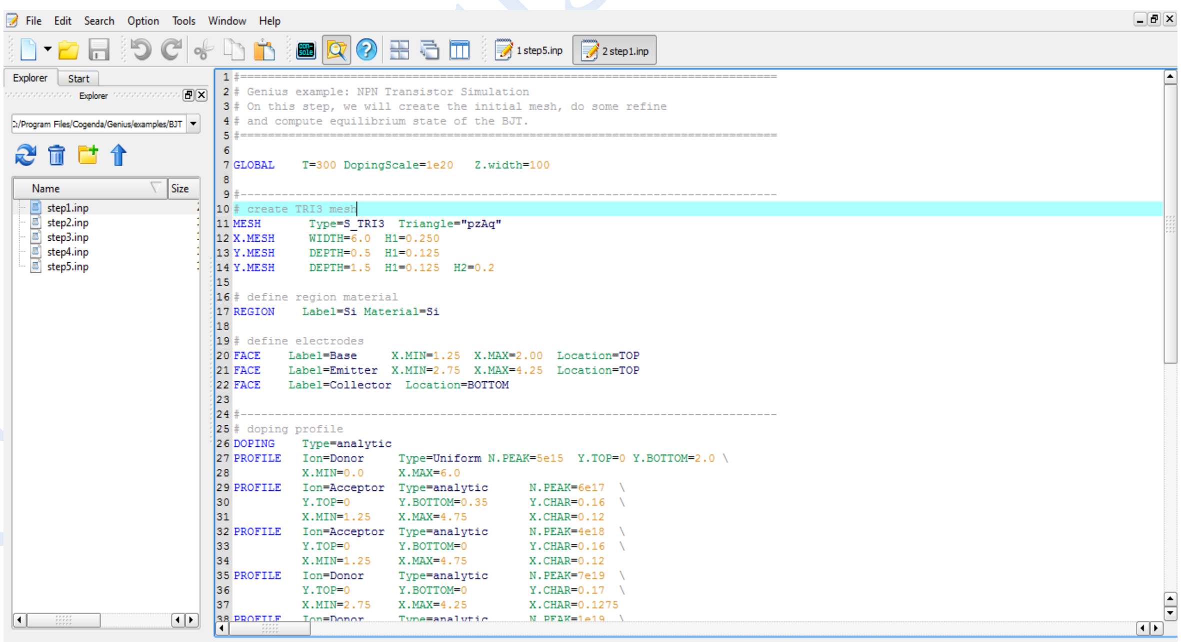


Fig 6.kewords enlightened in green color coding

5. Data Analysis: Spreadsheet and Plotting

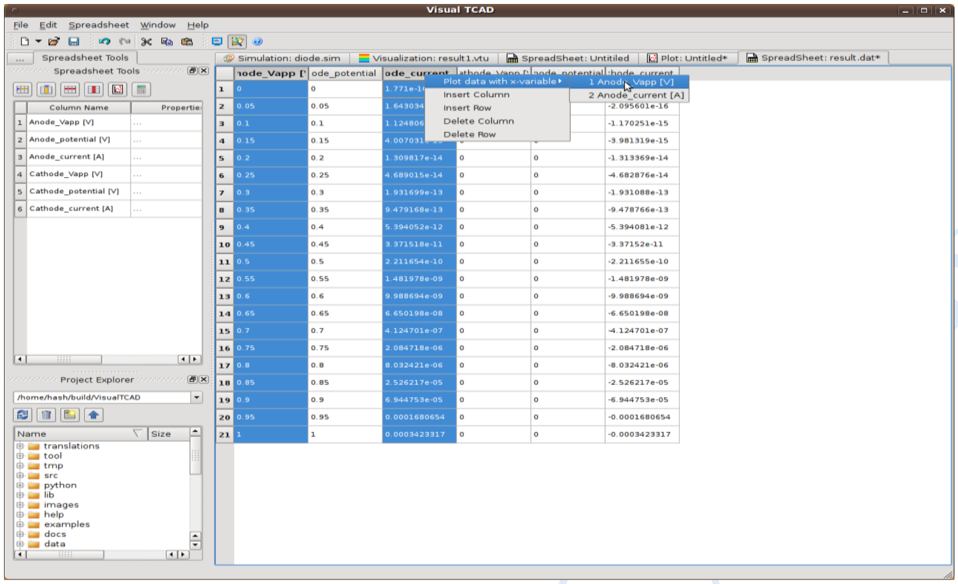


Fig 7.Illustration of a spreadsheet

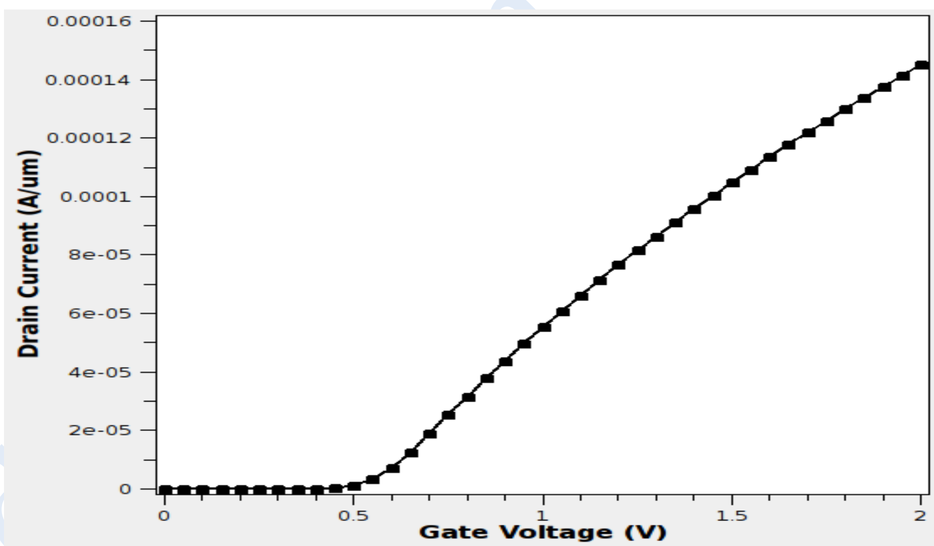


Fig 8. Illustration of a standard result of a NMOS device (input characteristics)*

2D Structures

1. Diode
2. NMOS Transistor (FET)
3. PMOS Transistor(FET)
4. CMOS(FET)
5. DAOUBLE GATE (FET)
6. Bipolar Junction Transistor(BJT)

1. DIODE

AIM: To study the diode and perform I-V characteristics curve with the help of simulations

Tool used: Some study about DIODE and Cogenda Visual TCAD tool

Theory: Diode is a 2 terminal device is also known as P-N junction diode and worked in both forward and reverse bias

- when the Positive voltage is given to the anode or P type junction and the negative voltage is given to the cathode or N type junction, the diode is in forward biased
- when the Negative voltage is given to the anode or P type junction and the Positive voltage is given to the cathode or N type junction, the diode is in reverse biased
- Diode is also used as a rectifier circuit
- Below in figure are the diode standard characteristics

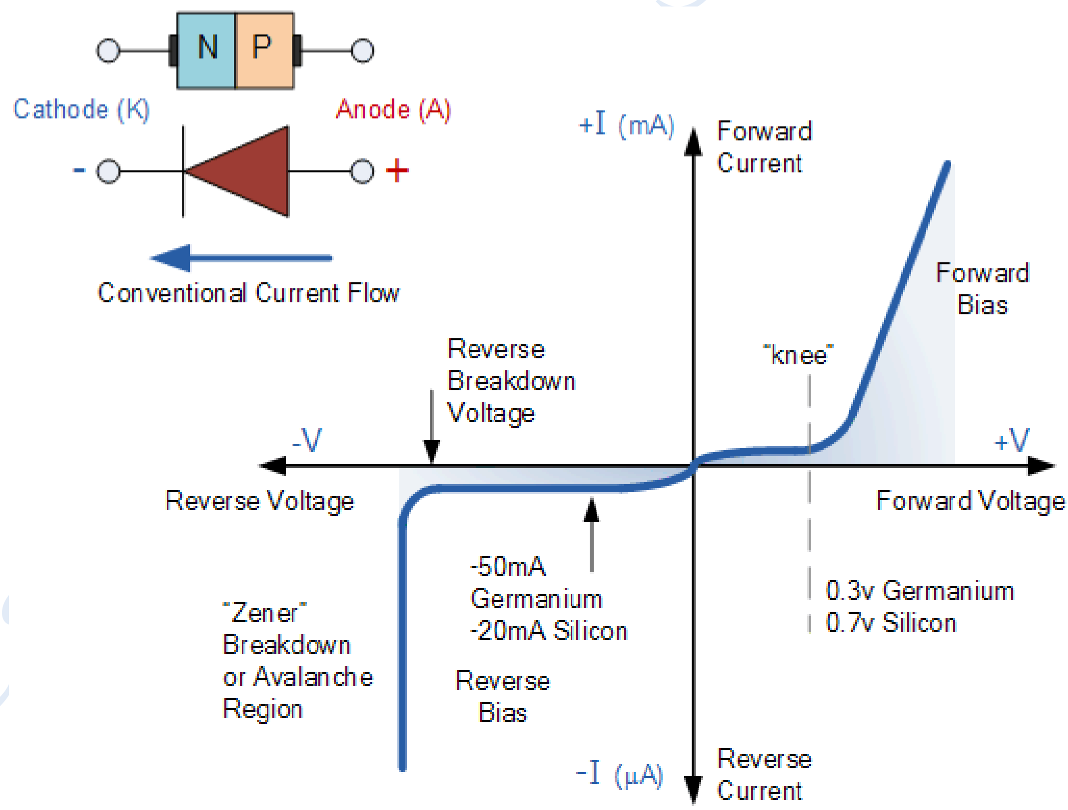


Fig. 1(a) standard characteristics of PN Jn diode with symbol

Procedure:

1. Open the Visual TCAD tool by clicking on the icon in start menu for windows user and for Linux(ubuntu or rhel) open the terminal [ctrl+alt+t]
2. Go to the new file and choose the Device Drawing as an option
3. A window with black background will open up with default grid spacing width of 0.01um
4. We are preparing a diode of (L=4.4um x H=4um) with the anode region (L=2um x H=0.2um) and cathode block(L=4um x H=0.2um)
5. With the help of the shapes given on the left top use rectangular shape to design the device
6. The actual shape is designed below using the rectangular shape

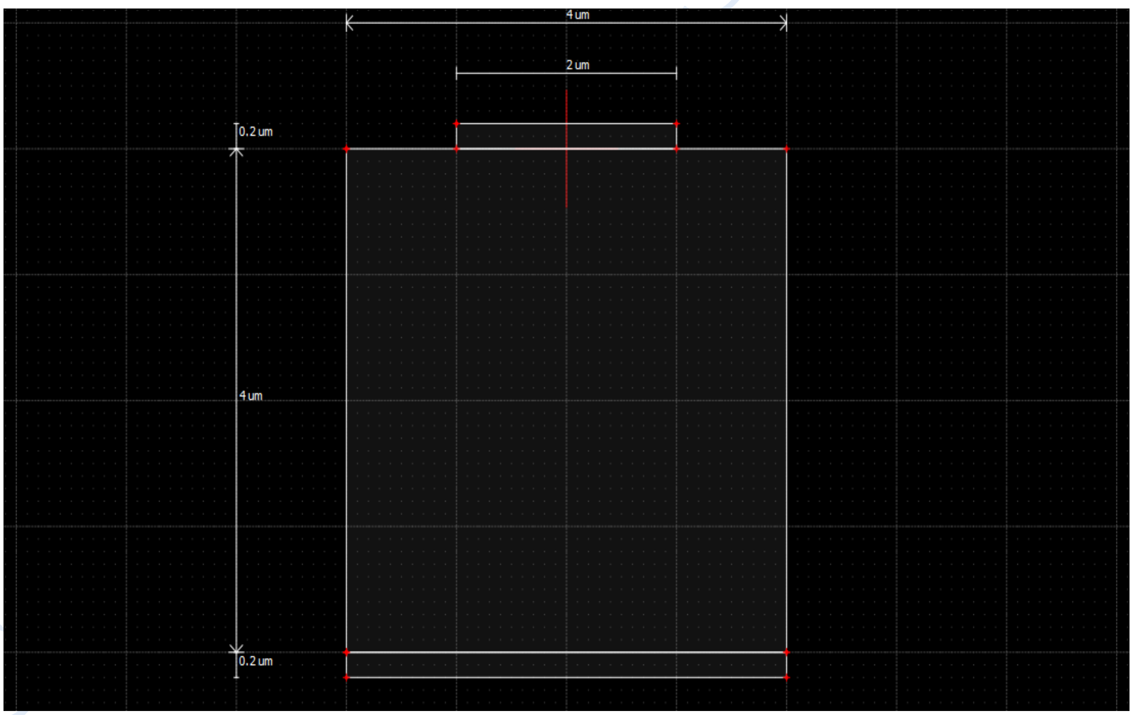


Fig. 1(b) Diode Structure

7. Add the region label substrate (4um x 4um) from the silicon material with the mesh size assumed as 1/10th of the block size e.g. as it is of 4um block length so max. meshing size will be 0.4um
8. Add the region label of the two blocks of (2um x 0.2um) & (4um x 0.2um) with the aluminum (Al) region and keep the max mesh size same as by default given as we don't require to calculate on the metal-semiconductor junctions

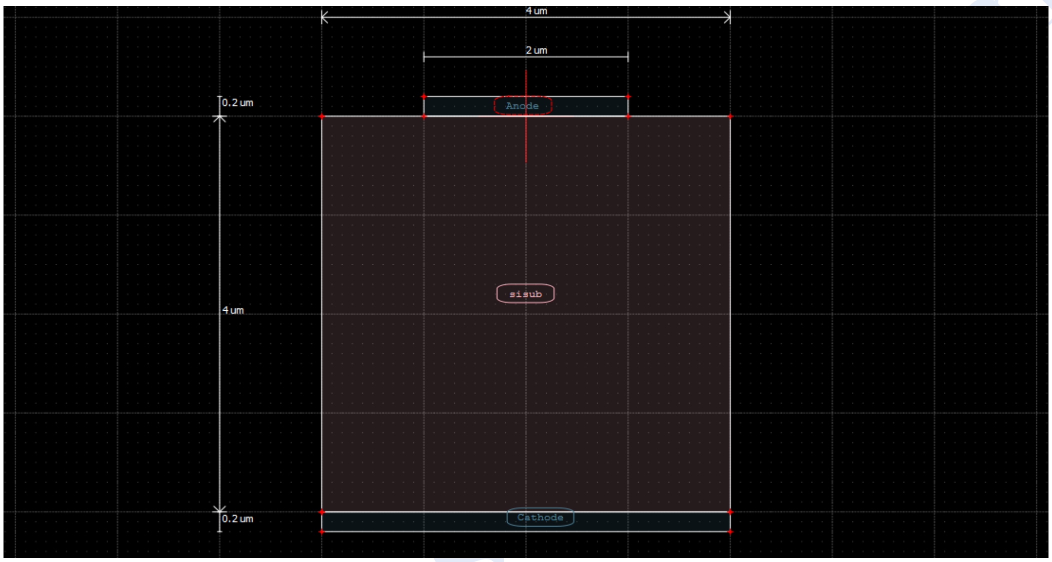
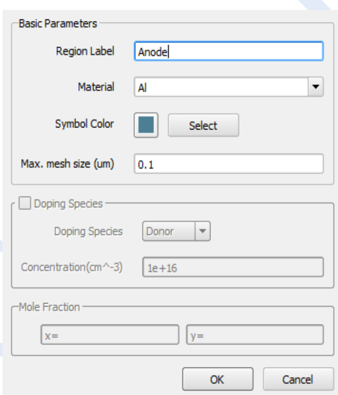
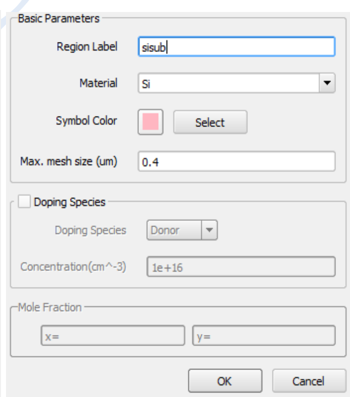


Fig. 1(c) Diode Regions with meshing, material and selected name





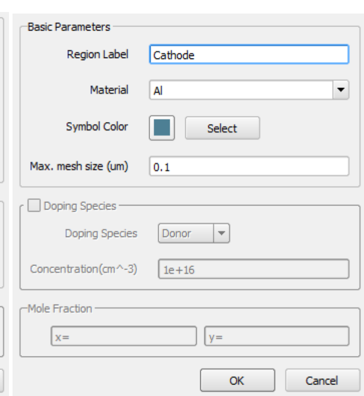


Fig. 1(d) Anode **Fig. 1(e) Si-Substrate** **Fig. 1(f) Cathode**

9. Now with the option of Add Doping Profile we will make the two regions
    - First doping profile is uniform doping profile **of N type / Donor** with the doping concentration of **(1e+16)/cm^3** across the whole region from top to bottom uniformly or we can say (4um x 4um) above the silicon region from top to bottom
    - Second doping profile is of P **type / Acceptor** from below the Anode from **top to bottom** with the dimensions (2um x 0.1um) along with the doping conc.(**1e+19) cm^3** and the **Y-characteristics** length will be **0.25um** with **X-Y ratio** of **1**

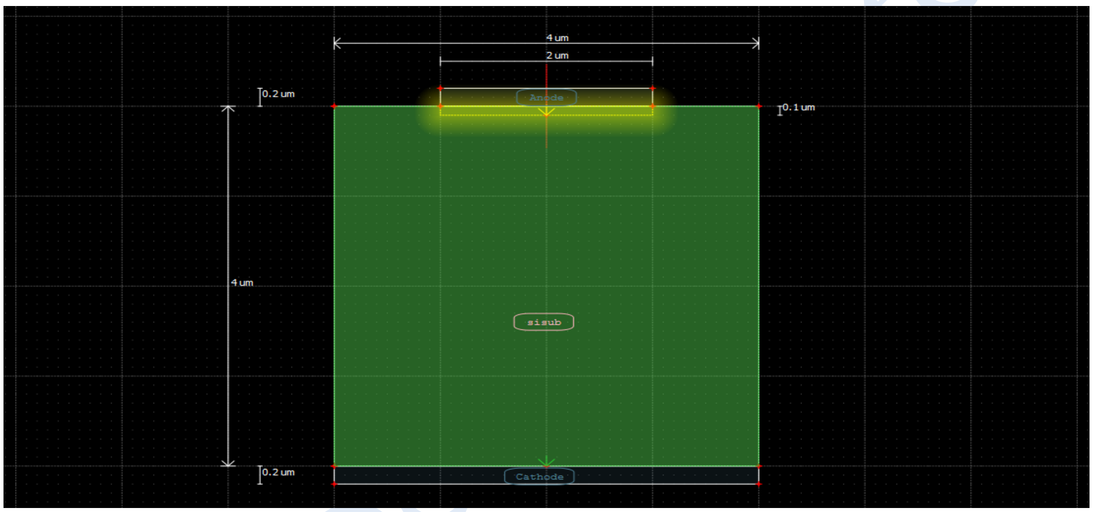
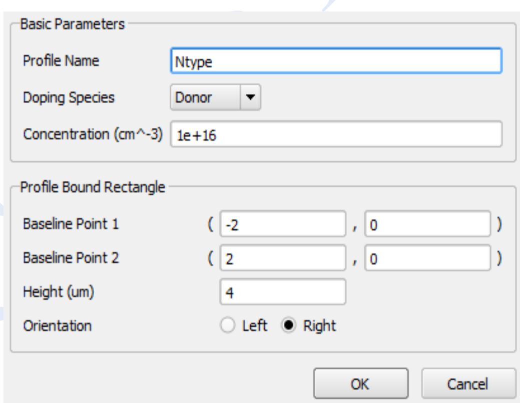


Fig. 1(g) Diode Doping Profile



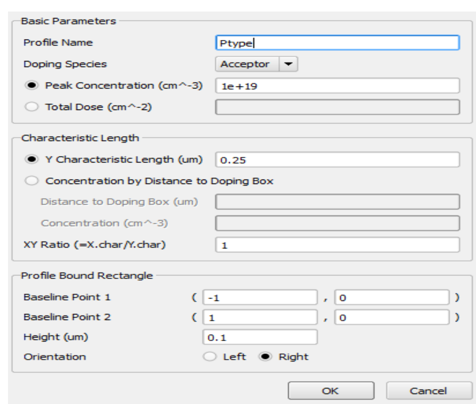


Fig. 1(h) N-type Uniform Doping Profile **Fig. 1(i) P-type Gaussian Doping Profile**

10. Now with the option **Do Mesh** we can meshed the device and refine the mesh with the option **Refine Existing Mesh** and we can also do the mesh by **spring method** ,the area of junction will be so denser after refining it 2-3 times ,the tool automatically detect the Junction of material and used to do the denser mesh at junctions automatically

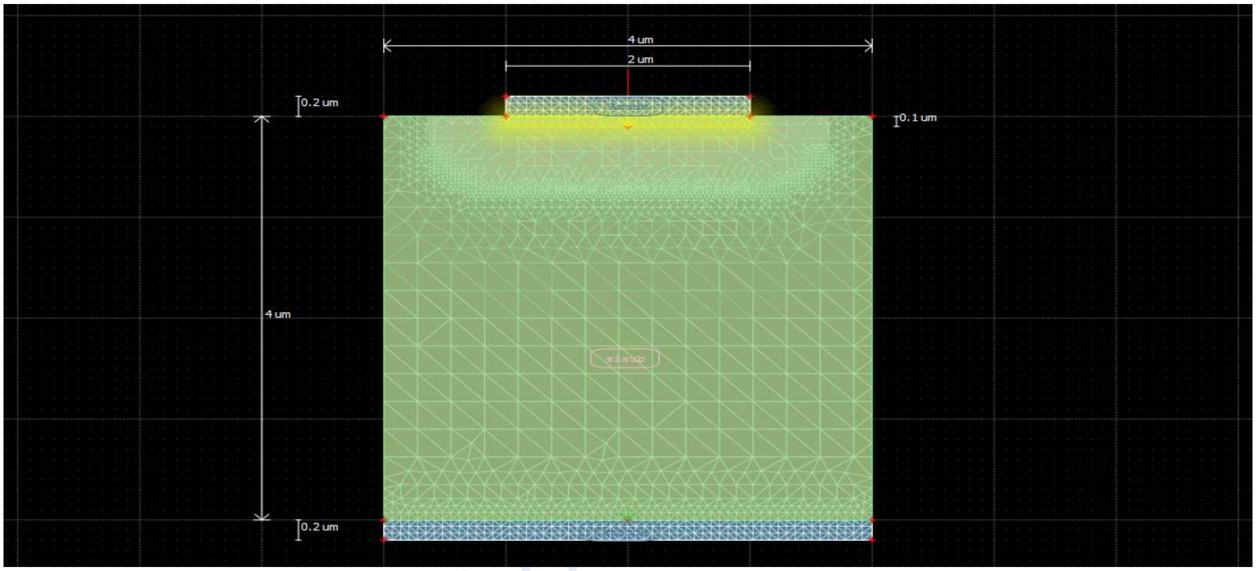


Fig. 1(j) Diode Meshing

11. After the meshing being done we will save this file to **.tif** file using **Device** option in menu bar above we will get an option below "**Save mesh to file**"
12. After the mesh file saved to .tif file **go to the file option above** and open the **Device simulation**
13. From the **folder below Setup** with the location we can open the .tif file and it will take some time or earlier load the structure from .tif file along with the contacts shown on the middle electrode work area e.g. Anode and Cathode respectively or vice versa
14. We will apply biasing as shown in fig below

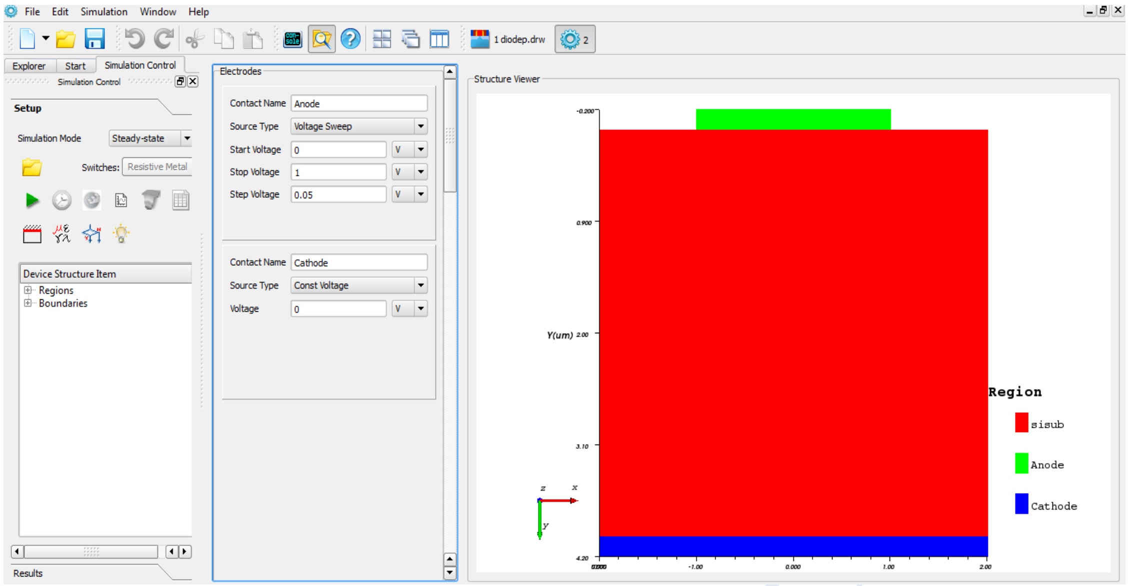
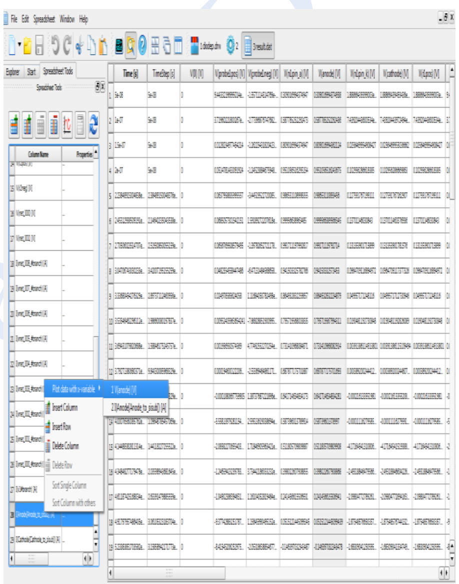


Fig. 1(k) Diode Simulation

15. Now with the help of Run button we can simulate our device by firstly save it and then **create a folder** named as results or run or as per user wants to give name to it, after choosing the folder for simulation our simulation will be submitted and will give the results or we can open the file **result.dat** from the folder we have selected to simulate in.
16. Now we can **Plot the data** with the help of results or spreadsheets **between I(anode)** and **V app(anode)**
17. **Results with Spreadsheet and Plot are given below:**



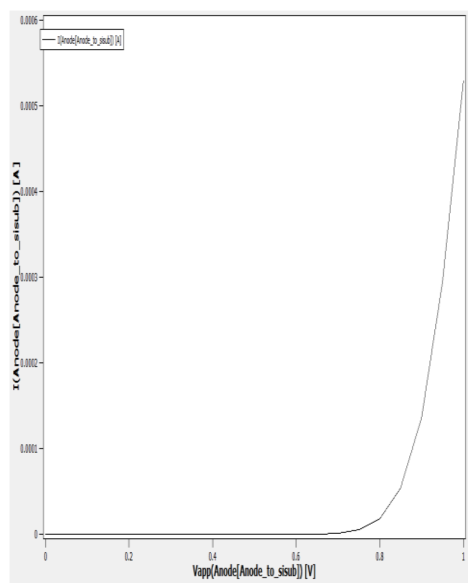
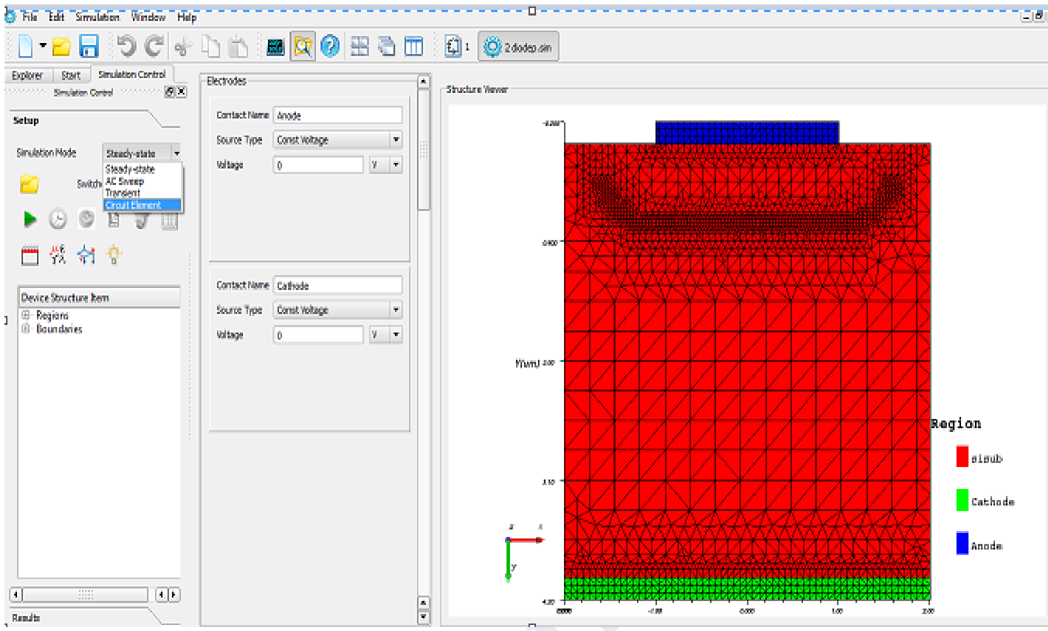
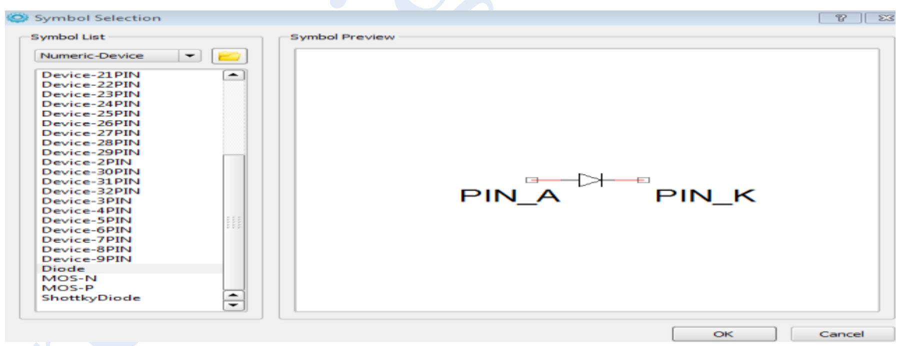


Fig. 1(l) Diode Simulation Results **Fig. 1(m) Diode Simulation Curve**

18. We will also make circuit symbol by loading the **.sim** file and selectthe circuit element like given below and select diode from the list after scrowling down in the device lists change symbol button to diode as shown





19. For half wave rectifier we will go to the File > New > Circuit Schematic and with the option numerical device[ ] as we have created **.sim** the circuit element of diode we will load and draw a below circuit with the other elements like voltage probe [ ], ground [ ],wire[ ],component [ ] etc as shown below

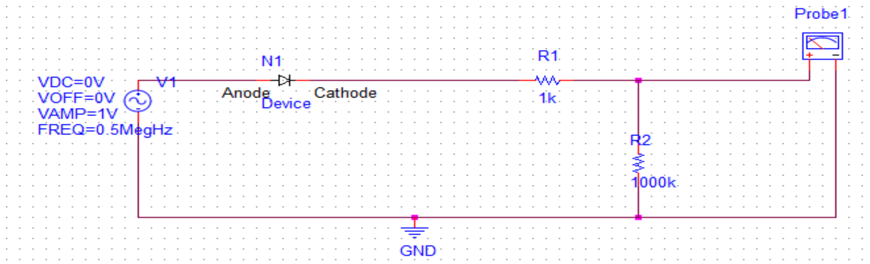


Fig. 1(n) Schematic circuit

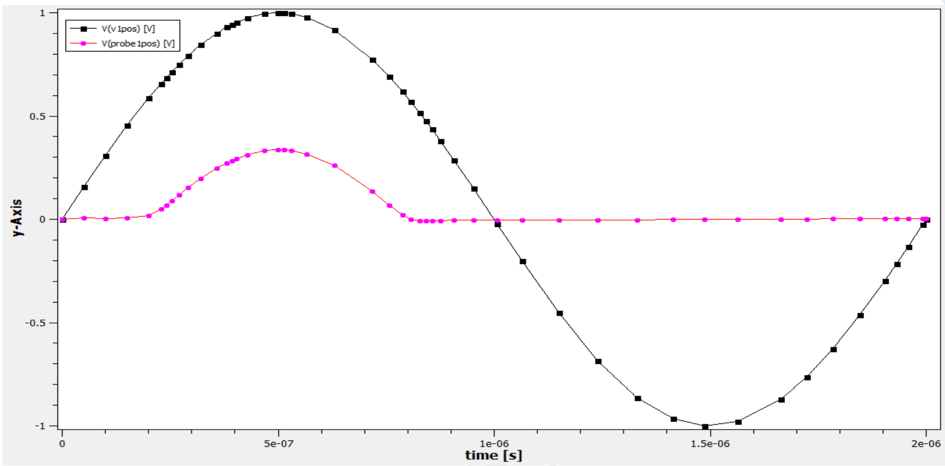


Fig. 1(o) Rectifier action characteristics