Introduction to MOSFET Simulation using *Silvaco TCAD*

SILVACO

Dept. EEE&I

BITS Pilani
KK Birla Goa Campus

Sanket Deshpande
2014B5A3599G

1. Introduction

TCAD - Technology **C**omputer **A**ided **D**esign

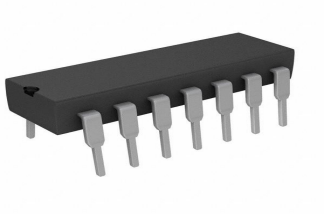
TCAD is a branch of electronic design automation that models semiconductor fabrication and semiconductor device operation.

- The modeling of the fabrication is termed as **Process TCAD.** [**ATHENA** in *Silvaco TCAD*]
- The modeling of the device operation is termed as **Device TCAD.** [**ATLAS** in *Silvaco TCAD*]

TCAD is a virtual device fabrication laboratory:

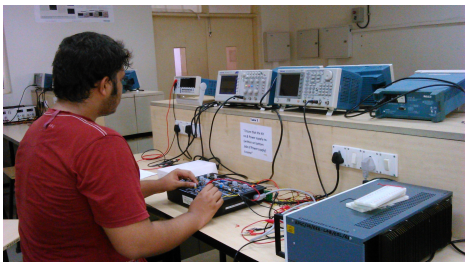
Source: Dept. of Physics and Astronomy, The University of Sheffield

ATHENA fabricates the device by various processes:



Source: Digikey Electronics

ATLAS analyzes the device characteristics:



Source: Dept. EEE&I, BITS Pilani, Goa Campus

2. Simulate the fabrication of an NMOS device

An example is used to demonstrate how this software can be used for various device simulations. This example illustrates how the *Silvaco TCAD* software can be used to simulate the fabrication of an NMOS device. The software is installed on a Linux based system. For a Windows based system, the UI and the steps may slightly vary from the ones illustrated in this manual.
To ensure better understanding, the device parameters have been selected to be the same as in the example available with the software. Each step is described with it's associated process in *DeckBuild* and the corresponding ASCII statement.

Step 1: Opening *DeckBuild*
Navigate to the installation folder of the sofware and click on *DeckBuild.* A window similar to the following image should open:

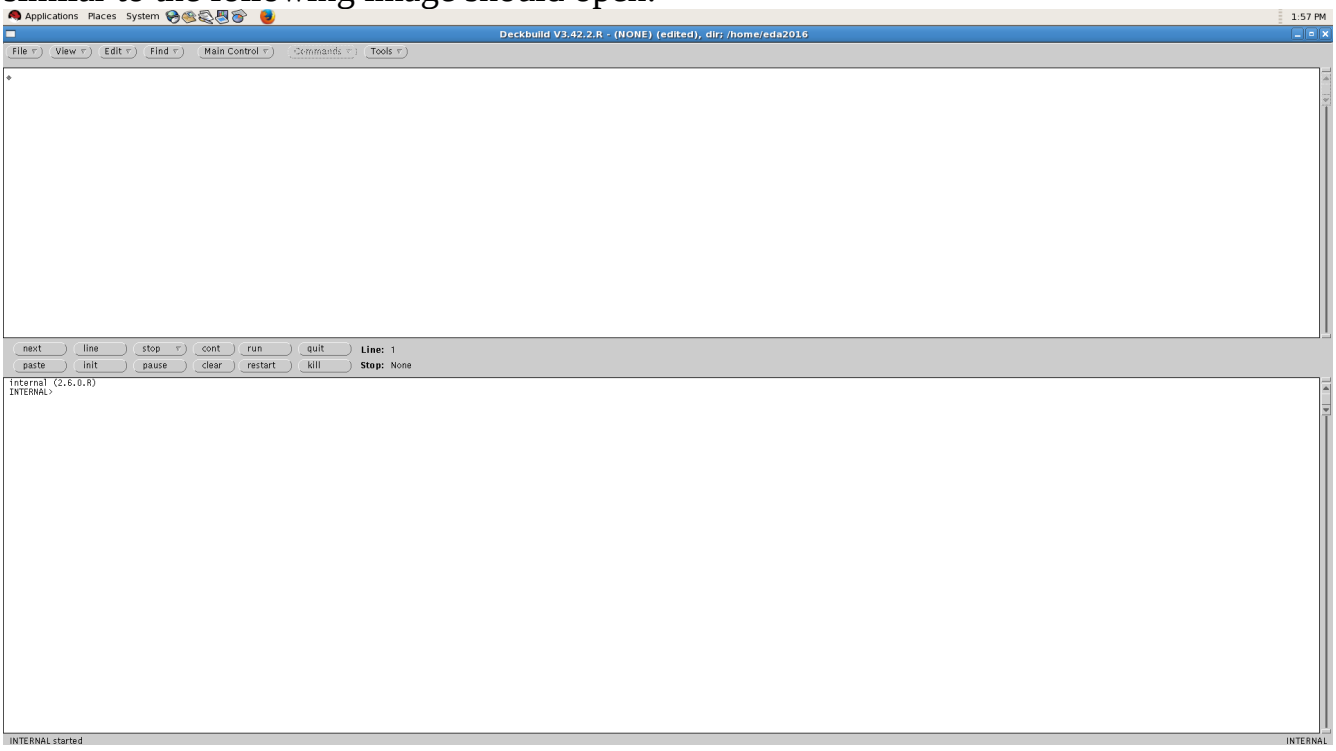


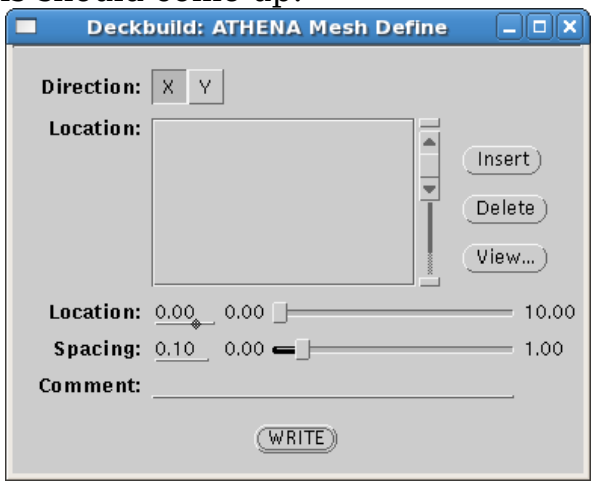
Fig. 1

Step 2: Start ATHENA
Type "`go athena`" in the top window and click on 'run'

Note: The input information for ATHENA is usually provided in the form of input files. An input file is a text file that can be prepared by using an ASCII text editor (such as vi on any UNIX system) or by using the *DeckBuid* utility. *DeckBuid* allows the user to create the input file using a GUI.

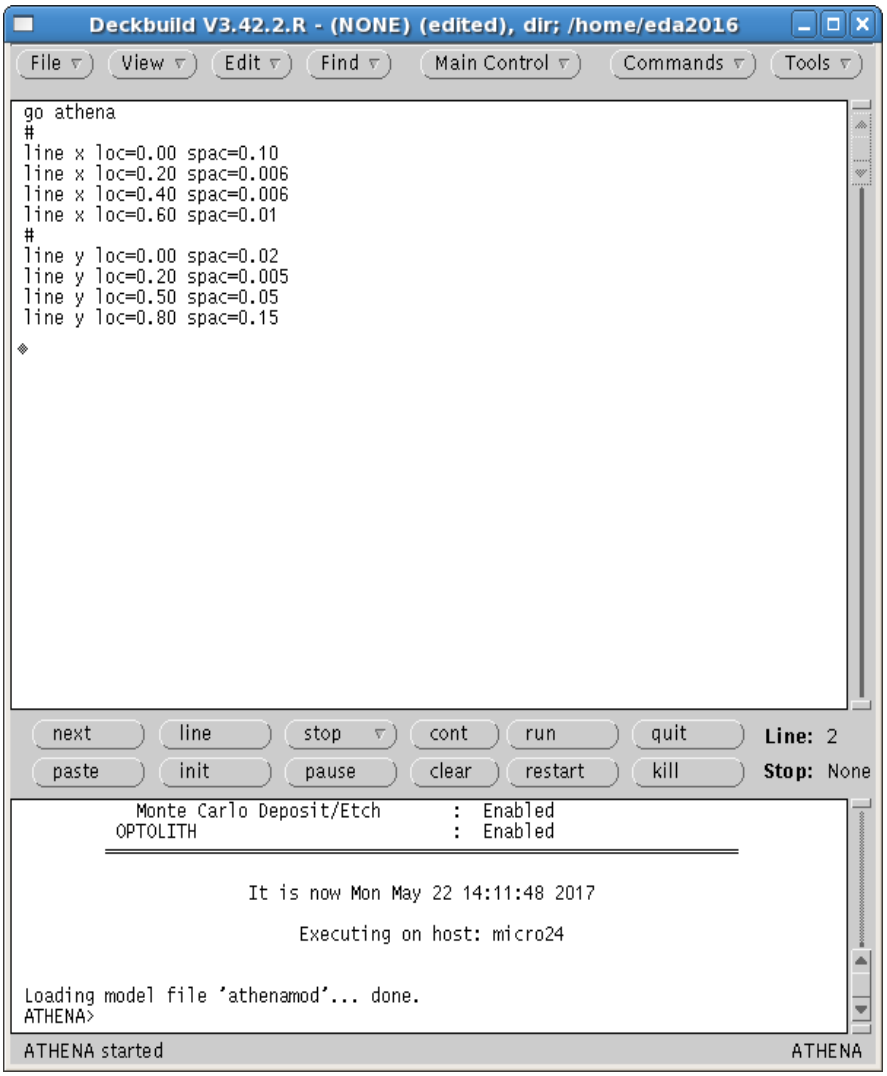
Step 3: Defining the initial rectangular grid

Right click on 'Commands'
Left click on 'Mesh Define...'
A window similar to this should come up:



1. Select Direction 'x'
2. Enter the Location: 0.0 ; Spacing: 0.1
3. Click on 'Insert'
4. Repeat 2 and 3 with:
   Location: 0.2 ; Spacing: 0.006
   Location: 0.4 ; Spacing: 0.006
   Location: 0.6 ; Spacing: 0.01
5. Select Direction 'y'
6. Repeat 2 and 3 with:
   Location: 0.0 ; Spacing: 0.002
   Location: 0.2 ; Spacing: 0.005
   Location: 0.5 ; Spacing: 0.05
   Location: 0.8 ; Spacing: 0.15
7. Click on 'WRITE'

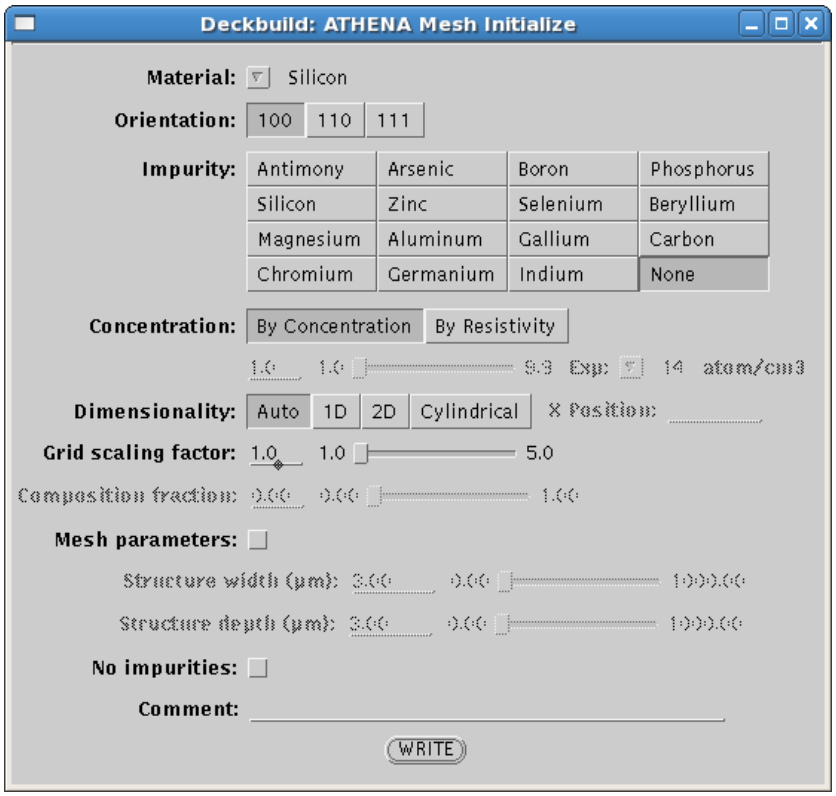
The DeckBuid top window show have the following statements:



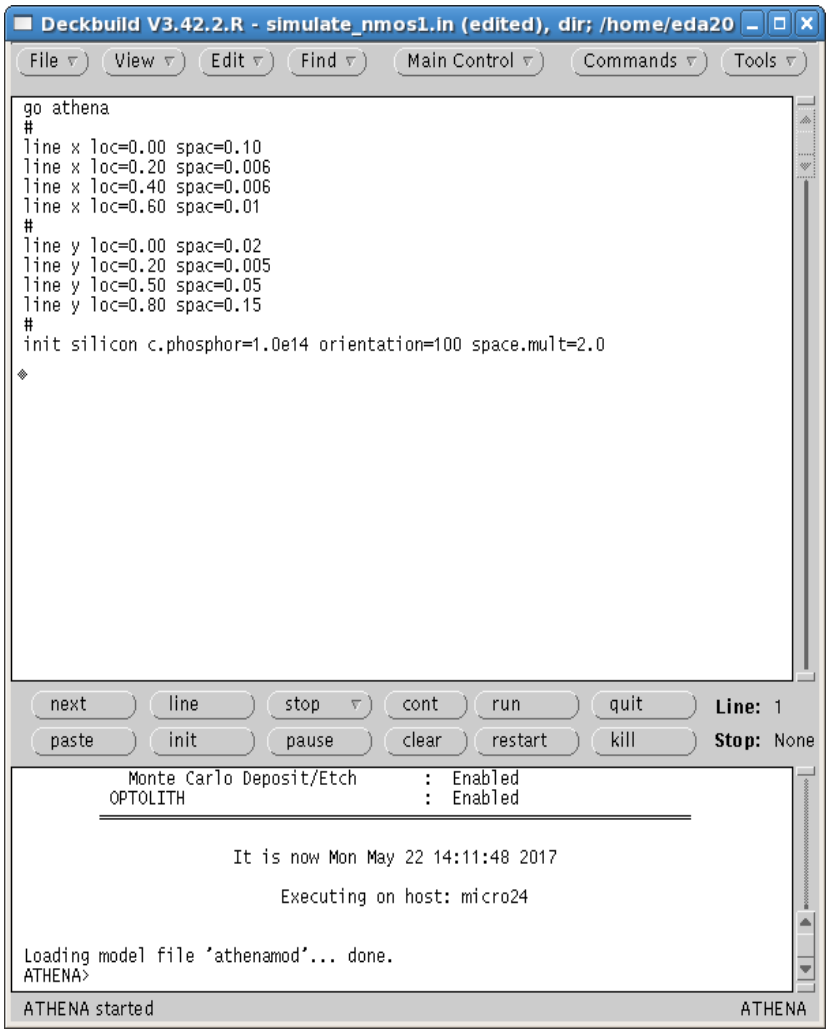
Step 4: Defining the initial substrate

1. Right click on 'Commands'
2. Left click on 'Mesh Initialize...'

3. A window similar to this should come up:



4. Select Orientation as '100'
5. Select Impurity as 'Phosphorus'
6. Select the concentration by entering '1.0' and then right click on the drop down menu of 'Exp' and select '14'
7. Enter Grid Scaling Factor as '2.0'
8. Click on 'WRITE'
9. The DeckBuild should have the following statements:

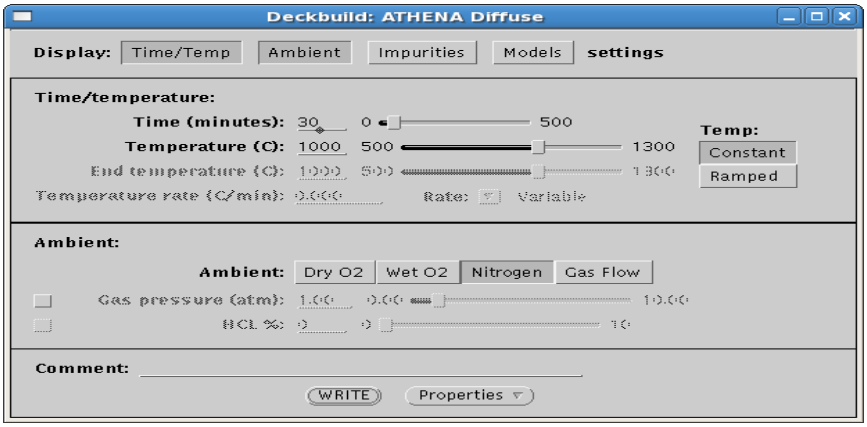


Step 5: Pwell formation including masking off of the Nwell

Enter as comment: #pwell formation including masking off of the nwell

Dry O2 Diffusion:

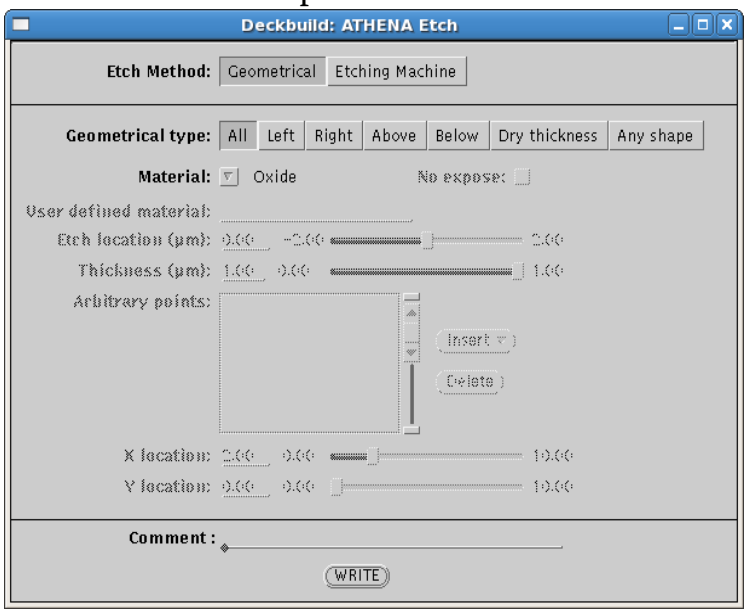
1. Right click on 'Commands'
2. Right click on 'Process'
3. Click on 'Diffuse'
4. A similar window should come up:



5. Enter time as 30 mins
6. Enter temperature as 1000 degrees Celsius
7. Select 'Dry O2' as 'Ambient'
8. Click on the checkbox of 'Gas Pressure' and enter as 1 atm
9. Click on the checkbox of 'HCL' and enter as 3%
10. Click on 'WRITE'

Etching

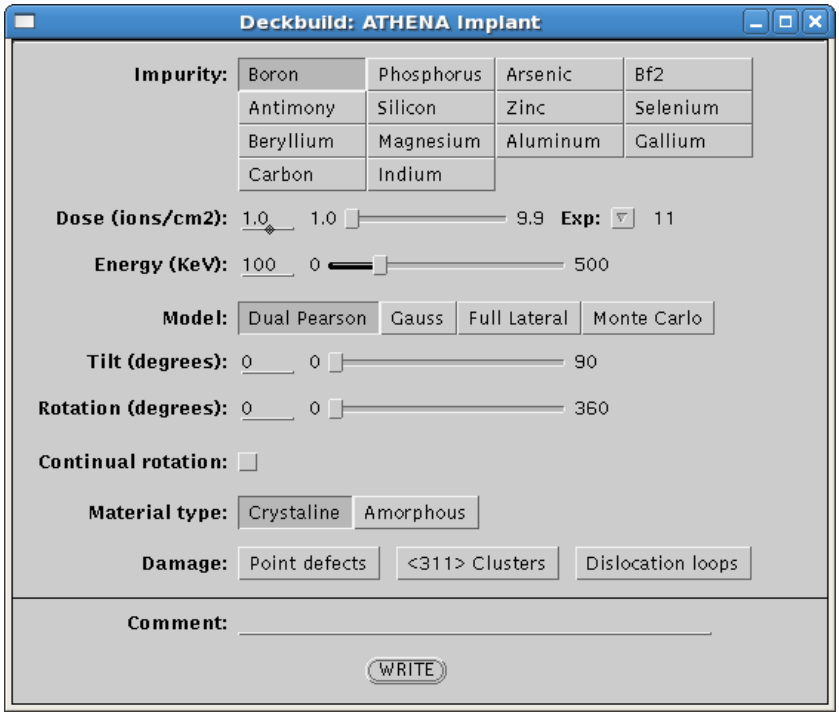
1. Right click on 'Commands'
2. Right click on 'Process'
3. Right click on 'Etch'
4. Left click on 'Etch...'
5. A similar window should come up:



6. Select 'Dry Thickness' as Geometrical type
7. Select 'Oxide' as the Material by right-click on the drop down menu
8. Enter thickness as 0.02 um
9. Click on 'WRITE'

P-well implant

1. From 'Commands', select 'Process' and then select 'Implant...'
2. A similar window should come up:

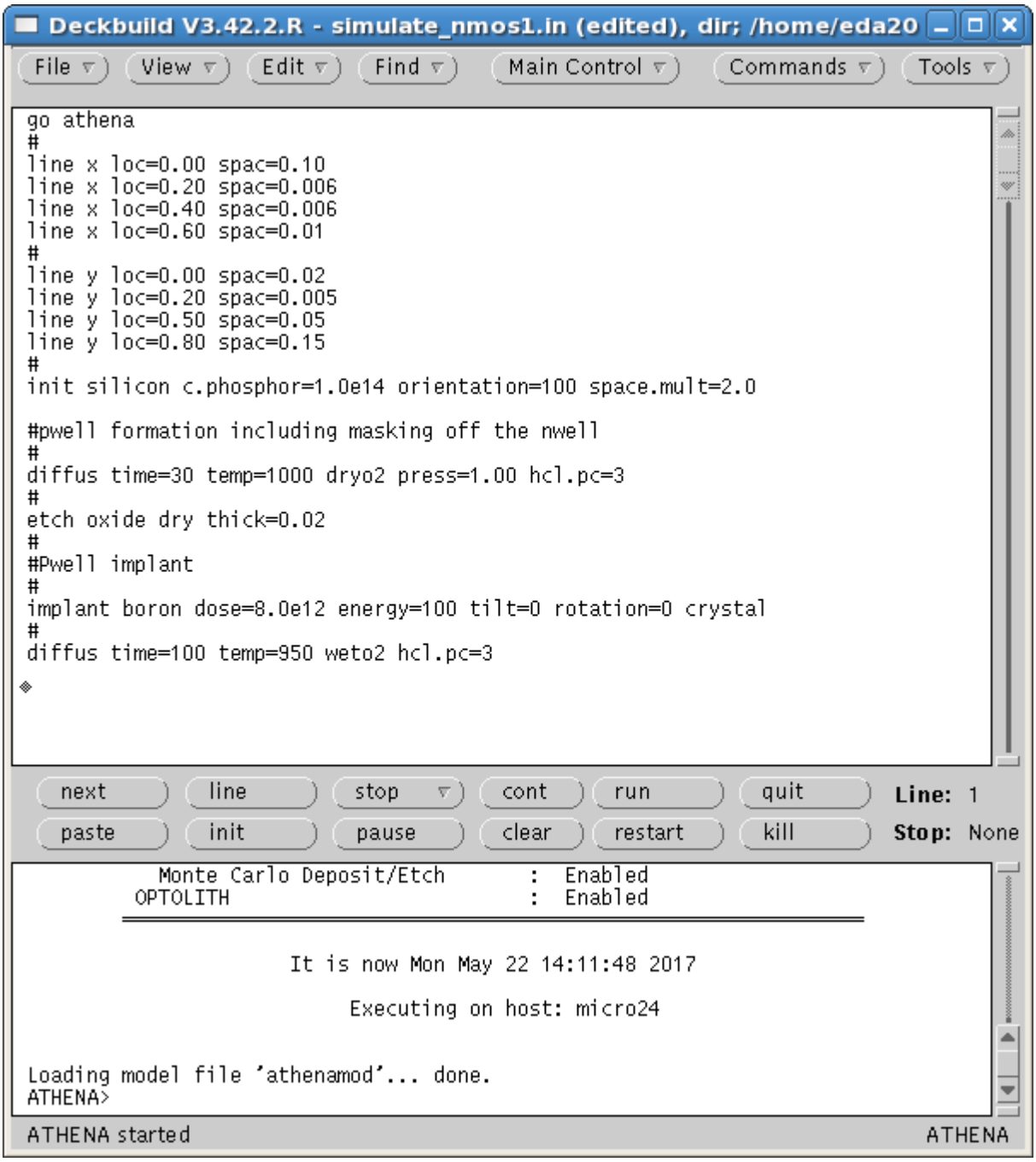


3. Select Impurity as 'Boron'
4. Input 'Dose' as 8e12 by entering 8.0. Then right click on the drop down menu of 'Exp' and select 12
5. Enter 'Energy' as 100KeV
6. Click on 'WRITE'

Wet O2 Diffusion

1. Open the diffusion window
2. Enter time as 100mins
3. Enter temperature as 950 degrees Celsius
4. Select 'Wet O2' as Ambient
5. Select only the HCL checkbox and enter 3%
6. Click on 'WRITE'

The DeckBuild window should now have the following statements:

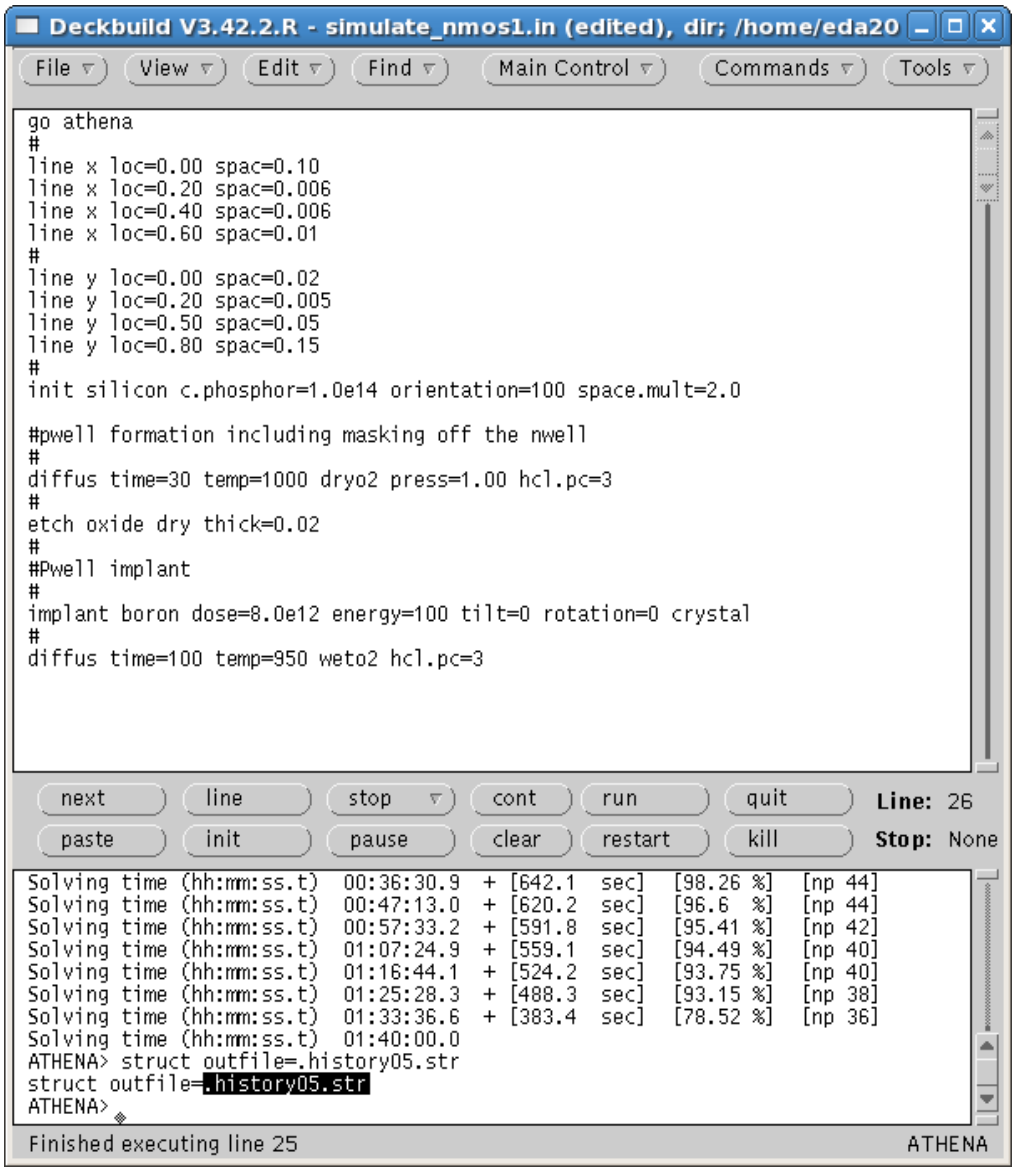


Step 6: What's done so far...

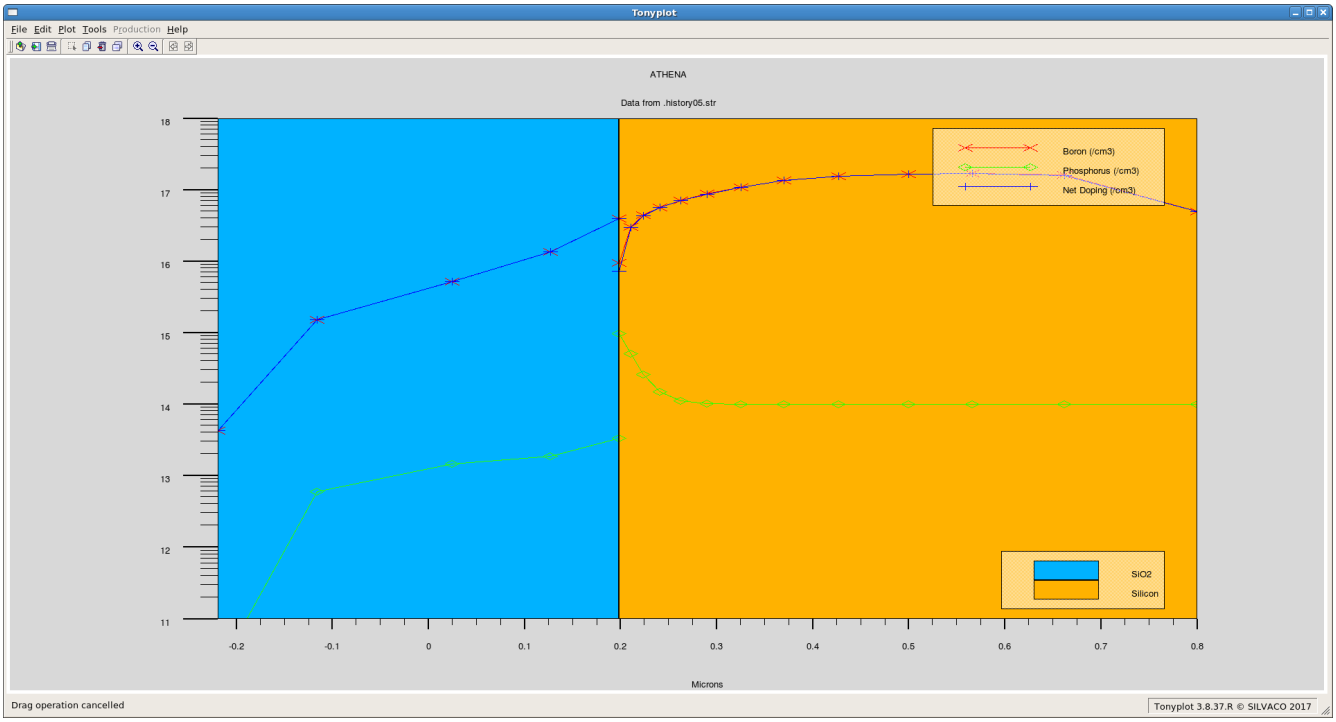
1. Save your file in an appropriate directory with the extension as ".in"
2. In this example, the file is named as "simulate_nmos1.in"
3. Now, click on 'run' in the DeckBuild window
4. After the process is completed in the bottom window will have struc outfile as a

history file.

5. Select the history file as shown in the following image:



6. Now, left click on 'Tools'
7. A new 'Tonyplot' window should open which illustrates the device fabrication completed so far.



Step 7: Well Drive

(Enter comment: #welldrive starts here)

DryO2 diffusion with Ramped Temperature:

1. Open the diffusion window
2. Select time as 50mins
3. Select Temp as 'Ramped'
4. Enter 'Temperature' as 1000 degrees Celsius
5. Enter 'End Temperature' as 1200 degrees Celsius
6. That gives us a 'Temperature rate' of 4 C/min
7. Select 'Dry O2' as 'Ambient'
8. Select the 'Gas pressure' checkbox and enter 0.1atm
9. Select the HCL% checkbox and enter 3
10. Click on 'WRITE'

Nitrogen diffusion at constant temperature

1. Open the diffusion window
2. Enter 'Time' as 220mins
3. Enter 'Temperature' as 1200 C
4. Select 'Nitrogen' as Ambient
5. Select 'Gas Pressure' as 1.0atm
6. Click on 'WRITE'

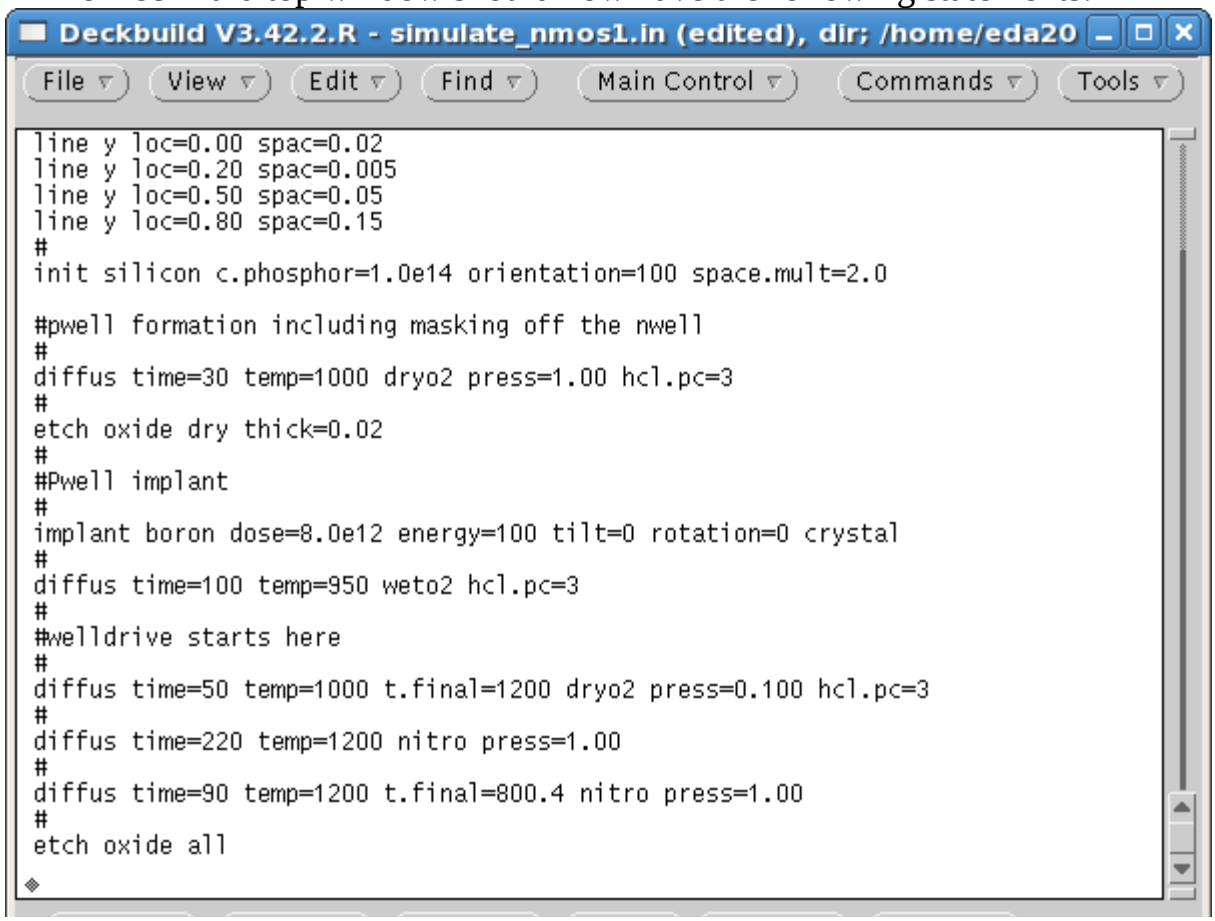
Nitrogen diffusion with ramped temperature

1. Open the diffusion window
2. Enter the 'Time' as 90mins
3. Select 'Temp' as 'Ramped'
4. Enter 'Temperature' as 1200 C
5. Enter 'End Temperature' as 800.4 C
6. This gives us a Temperature rate rate of -4.44 C/min
7. Select 'Nitrogen' as 'Ambient'
8. Enter 1.0 atm as 'Gas Pressure'
9. Click on 'WRITE'

Etch Oxide

1. Open the etching window
2. Select the 'Geometrical Type' as 'All'
3. Select the material as 'Oxide'
4. Click on 'WRITE'

The DeckBuid top window should now have the following statements:



Step 8: Cleaning the Oxide

(Enter comment: #cleaning oxide)

Diffusion with DryO2

1. Open the diffusion window
2. Enter 'Time' as 20mins
3. Enter 'Temperature' as 1000 C
4. Select 'Dry O2' as 'Ambient'
5. Select the 'Gas Pressure' checkbox and enter 1.0 atm
6. Select the 'HCL%' checkbox and enter 3
7. Click on 'WRITE'

Etch Oxide

1. Open the etching window
2. Select the 'Geometrical Type' as 'All'
3. Select the material as 'Oxide'
4. Click on 'WRITE'

Step 9: Growing the gate oxide

(Enter comment: #gate oxide grown here)

Diffusion with DryO2

1. Open the diffusion window
2. Enter 'Time' as 11mins
3. Enter 'Temperature' as 925 C
4. Select 'Dry O2' as 'Ambient'
5. Select the 'Gas Pressure' checkbox and enter 1.0 atm
6. Select the 'HCL%' checkbox and enter 3
7. Click on 'WRITE'

Note: The user can check the modifications in the device fabrication at any time by following Step 6.

Step 10: Extract a design parameter

(Enter comment: #extract a design parameter)

Any design parameter can be extracted by following this procedure.

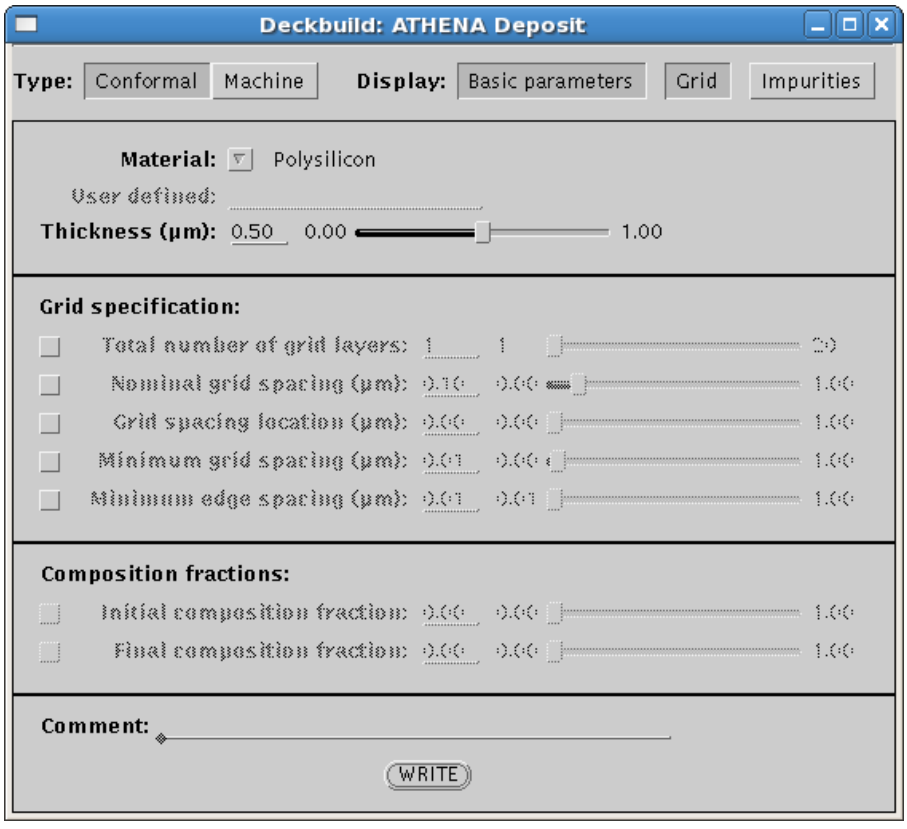
1. Right click on 'Commands'
2. Click on 'Extract...'
3. Select 'Material Thickness' from the drop down menu
4. Enter name as “gateox”
5. Select material as 'SiO2'

6. Enter 'Occurrence' as 1
7. Enter 'Exact location' of X coordinate by selecting 'X' and then entering 0.05
8. Click on 'WRITE'

Step 11: Implant Boron and deposit Polysilicon

1. Open the Implant window
2. Select 'Impurity' as 'Boron'
3. Enter 'Dose' as 9.5e11 ions/cm^2
4. Enter 'Energy' as 10 KeV
5. Click on 'WRITE'

1. Right click on 'Commands'
2. Right click on 'Process'
3. Right click on 'Deposit'
4. Click on 'Deposit...'
5. A Deposit window should come up:



6. Select 'Type' as 'Conformal'
7. Select 'Material' as 'Polysilicon'
8. Enter 'Thickness' as 0.2 um
9. Select the checkbox of 'Total number of grid layers' and enter 10
10. Click on 'WRITE'

From now on, we operate in 2 Dimensions
(Enter comment: #we operate in 2D)

Step 12: Etch polysilicon

1. Open the Etching window
2. Select 'Geometrical Type' as 'Left'
3. Select Material as 'Polysilicon' from the drop-down menu
4. Enter the 'Etch location' as 0.35um
5. Click on 'WRITE'

Step 13: Diffuse Wet O2 using 'Fermi Compress' Model

1. Open the Diffusion window
2. In the 'Display' menu, click on the 'Models' tab
3. Enter 'Time' as 3mins
4. Enter 'Temperature' as 900 C
5. Select 'Wet O2' as Ambient
6. Select only the 'Gas Pressure' checkbox and enter 1.0atm
7. In the 'Models' section, select the 'Diffusion' checkbox and select the option 'Fermi'
8. Select the 'Oxidation' checkbox and select 'Compressible'
9. Click on 'WRITE'

Step 14: Implant Phosphorus with dose = 3.0e13 ions/cm^2 and energy = 20 KeV

Step 15: Deposit oxide of thickness 0.12um and 8 grid layers

Step 16: Etch oxide of dry thickness = 0.12um

Step 17: Implant Arsenic of dose = 5.0e15 ions/cm^2 and energy 50 KeV

Step 18: Diffuse Nitrogen using 'Fermi Compress' Model with Time of 1min, Temperature as 900 C and Energy as 50 KeV

Step 19: Contact metal

Etch oxide

1. Open the Etching window
2. Select 'Etch Method' as 'Geometrical' and 'Geometrical type' as 'left'
3. Select 'Material' as 'Oxide'
4. Enter 'Etch location' as 0.2 um
5. Click on 'WRITE'

Deposit Aluminum

1. Open the 'Deposit' window
2. Select 'Material' as 'Aluminum'
3. Enter 'Thickness' as 0.03 um
4. In 'Grid specifications', select the 'Total number of grid layers' checkbox and enter 2.
5. Click on 'WRITE'

Etch Aluminum

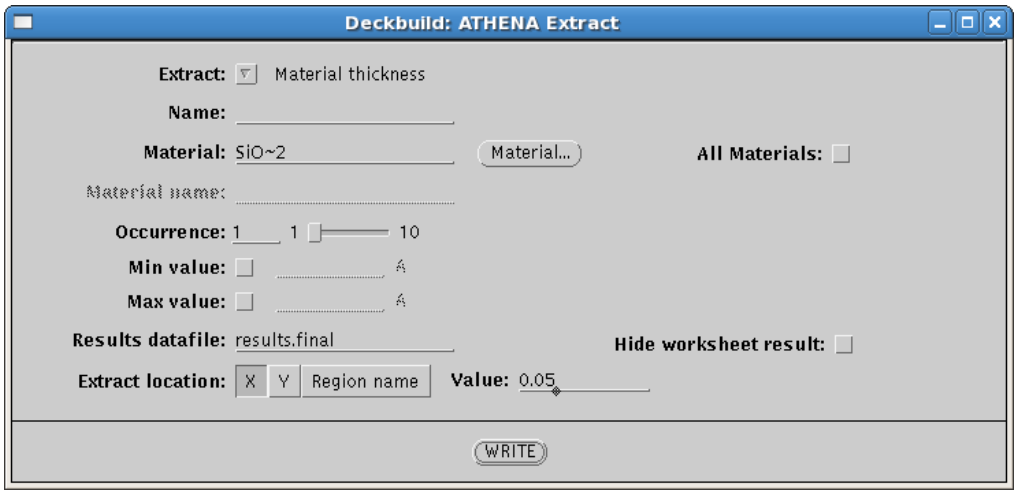
1. Open the Etch window
2. Select 'Etch Method' as 'Geometrical' and 'Geometrical type' as 'Right'
3. Select 'Material' as 'Aluminum'
4. Enter location as 0.18 um
5. Click on 'WRITE'

Note: You can now run Tonyplot and check how the device looks so far. You will notice that only the half of the device has been designed. This is because the device is symmetrical and in the latter part of this example, we will complete the other half as well.

Step 20: Extracting design parameters

Material Thickness

1. Right click on 'Commands'
2. Click on 'Extract...'
3. A window similar to this should appear:



4. Select 'Extract' as 'Junction Depth' by right clicking on the drop down menu
5. Give a name. Here, the name is given as “nxj”
6. Select 'Material' as 'Silicon'

7. Enter 'Occurrence' as 1 and 'Junction Occurrence' as 1
8. Select 'X' in 'Extract Location' and enter value as 0.1
9. Click on 'WRITE'

N++ region sheet resistance

1. Open the Extract window
2. Select 'Sheet resistance' from the 'Extract' drop down menu
3. Give a name. Here, it is “n++ sheet rho”
4. Select material as 'Silicon'
5. Enter 'Occurrence' and 'Region occurrence' as 1
6. Select 'X' in 'Extract location' and enter 0.05
7. Click on 'WRITE'

Sheet Resistance under the spacer, of the LDD region

1. Open the Extract window
2. Select 'Sheet resistance' from the 'Extract' drop down menu
3. Give a name. Here, it is “ldd sheet rho”
4. Select material as 'Silicon'
5. Enter 'Occurrence' and 'Region occurrence' as 1
6. Select 'X' in 'Extract location' and enter 0.3
7. Click on 'WRITE'

Surface Concentration under the channel

1. Open the Extract window
2. Select 'Surface concentration' from the 'Extract' drop down menu
3. Give a name. Here, it is “chan surf conc”
4. Select 'Material' as 'Silicon'
5. Enter 'occurrence' as 1
6. Select 'Impurity' as 'Net doping'
8. Select 'X' in 'Extract location' and enter 0.45
9. Click on 'WRITE'

Curve of conductance vs bias

1. Open the Extract window
2. Select 'Sheet conductance versus bias' from the 'Extract' drop down menu
3. Give a name. Here, it is “sheet cond v bias”
4. Select 'Material' as 'Silicon'
5. Enter 'Occurrence' and 'Region occurrence' as 1
6. In 'X vs Y axis', select 'Bias vs Conductance'
7. In 'Store X/Y datafile', select 'Yes' and enter a filename. Here, it is “extract.dat”
8. In 'Extract location', select 'X' and enter value as 0.45
9. In 'Define layers', select 'Bias'

10. In the Bias menu, select 'Material' as 'Polysilicon'
11. Select 'Occurrence' and 'Region occurrence' as 1
12. Enter 'Bias' as 0.0
13. Click on 'Insert'
14. Select the 'Ramped Bias' checkbox
15. Enter the 'Bias Step' as 0.2
16. Enter the 'Bias Stop' as 2
17. Click on 'WRITE'

Long channel Vt

1. Open the Extract window
2. Select 'QUICKMOS 1D Vt' from the 'Extract' drop down menu
3. Give a name. Here, it is “n1dvt”
4. In 'Device type', select 'NMOS'
5. Select the 'Qss' checkbox and enter value 1e10
6. In 'Extract location' select 'X' and enter value 0.49
7. Click on 'WRITE'
8. After the statement has been written, make an edit by adding vb=0.0. Like this:

```
extract name="n1dvt" 1dvt ntype vb=0.0 qss=1e10 x.val=0.49
```

Step 21: Completing the other half of the device

Taking the mirror image

1. Right click on 'Commands'
2. Right click on 'Structure'
3. Click on 'Mirror...'
4. Mirror window should open
5. Since we have developed the left part of the device, we need a mirror image on the right.
6. Therefore, select the 'Right' tab and click on 'WRITE'

Step 22: Placing Electrodes

Gate

1. Right click on 'Commands'
2. Right click on 'Structure'
3. Click on 'Electrode...'
4. Electrode window should open
5. In 'Electrode Type', select 'Specified Position'
6. Enter name as “gate”
7. x = 0.5 and y = 0.1

Source

1. Open the 'Electrode' window
2. In 'Electrode type' select 'Specified Position'
3. Enter name as “source”
4. x = 0.1

Drain

1. Open the 'Electrode' window
2. In 'Electrode type' select 'Specified Position'
3. Enter name as “drain”
4. x = 1.1

Substrate

1. Open the 'Electrode' window
2. In 'Electrode type' select 'Backside'
3. Enter name as “substrate”

Step 23: Creating an output file

In the top window of DeckBuild, enter the following statement:

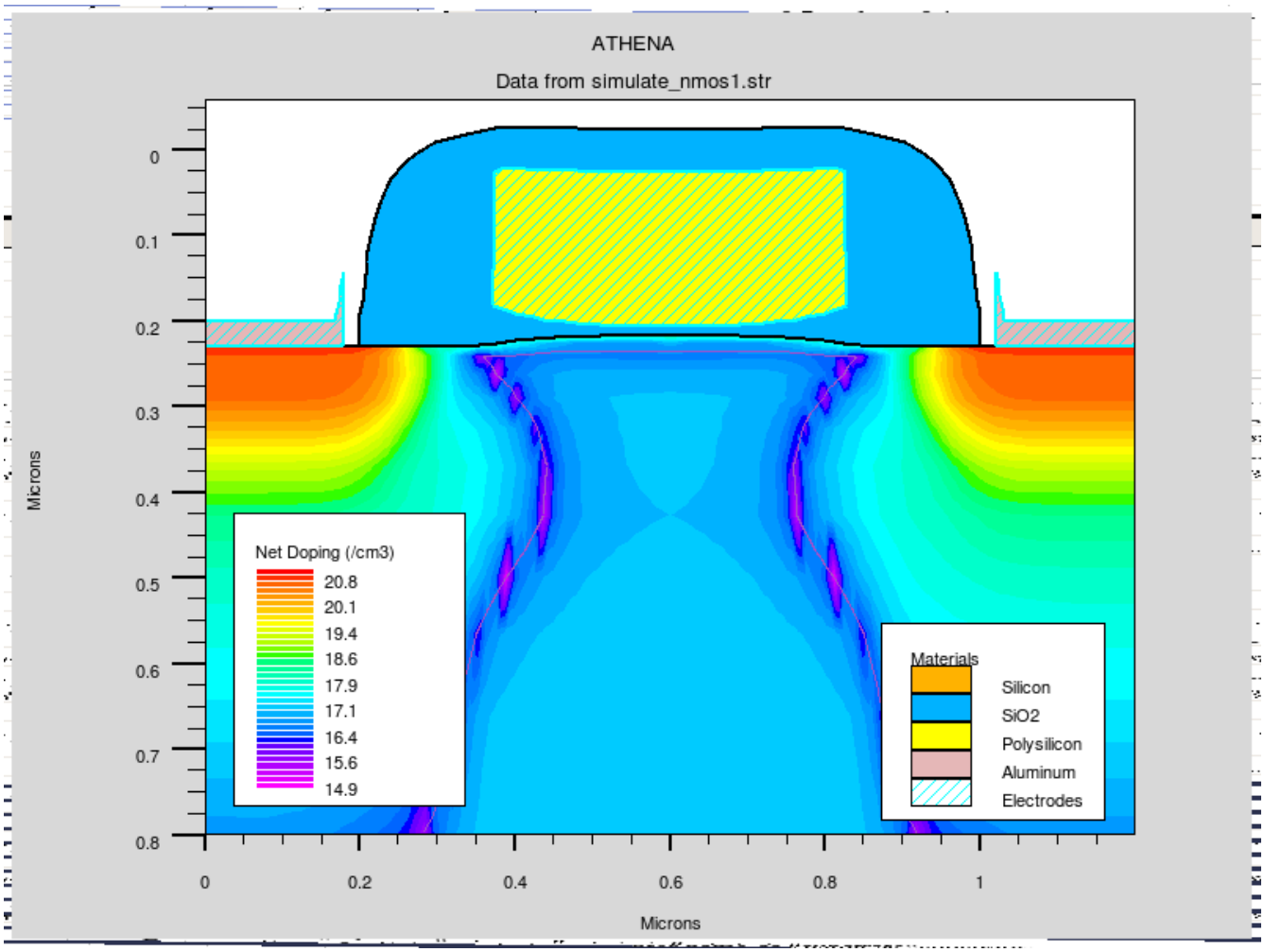
```
structure outfile=simulate_nmos1.str
```

Step 24: Plot the structure

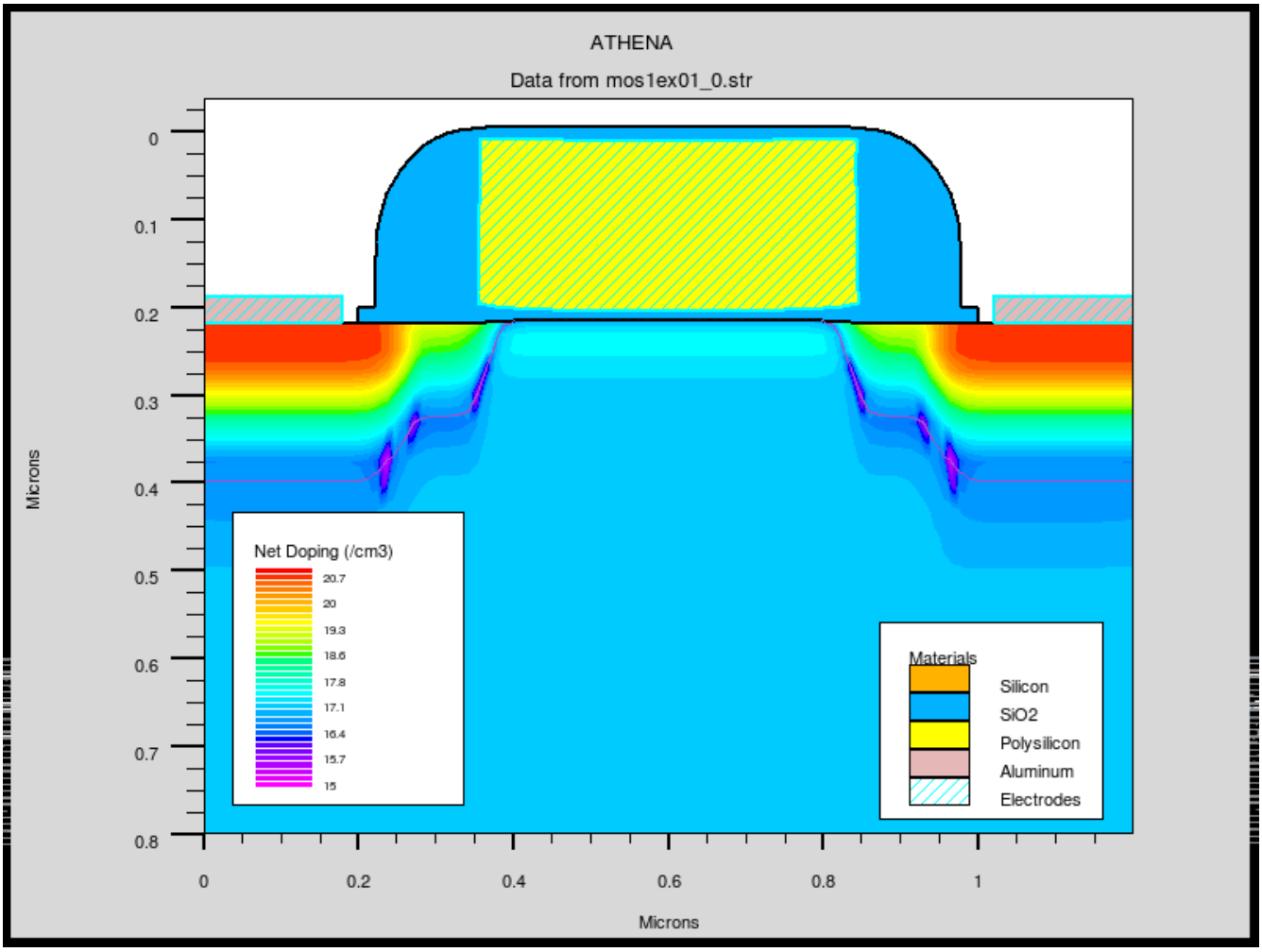
In the top window of DeckBuild, enter the following statement:

```
tonyplot  simulate_nmos1.str -set simulate_nmos1.set
```

The following device structure is obtained:



This can be compared with the structrure obtained by running the mos1ex_01_0 example:

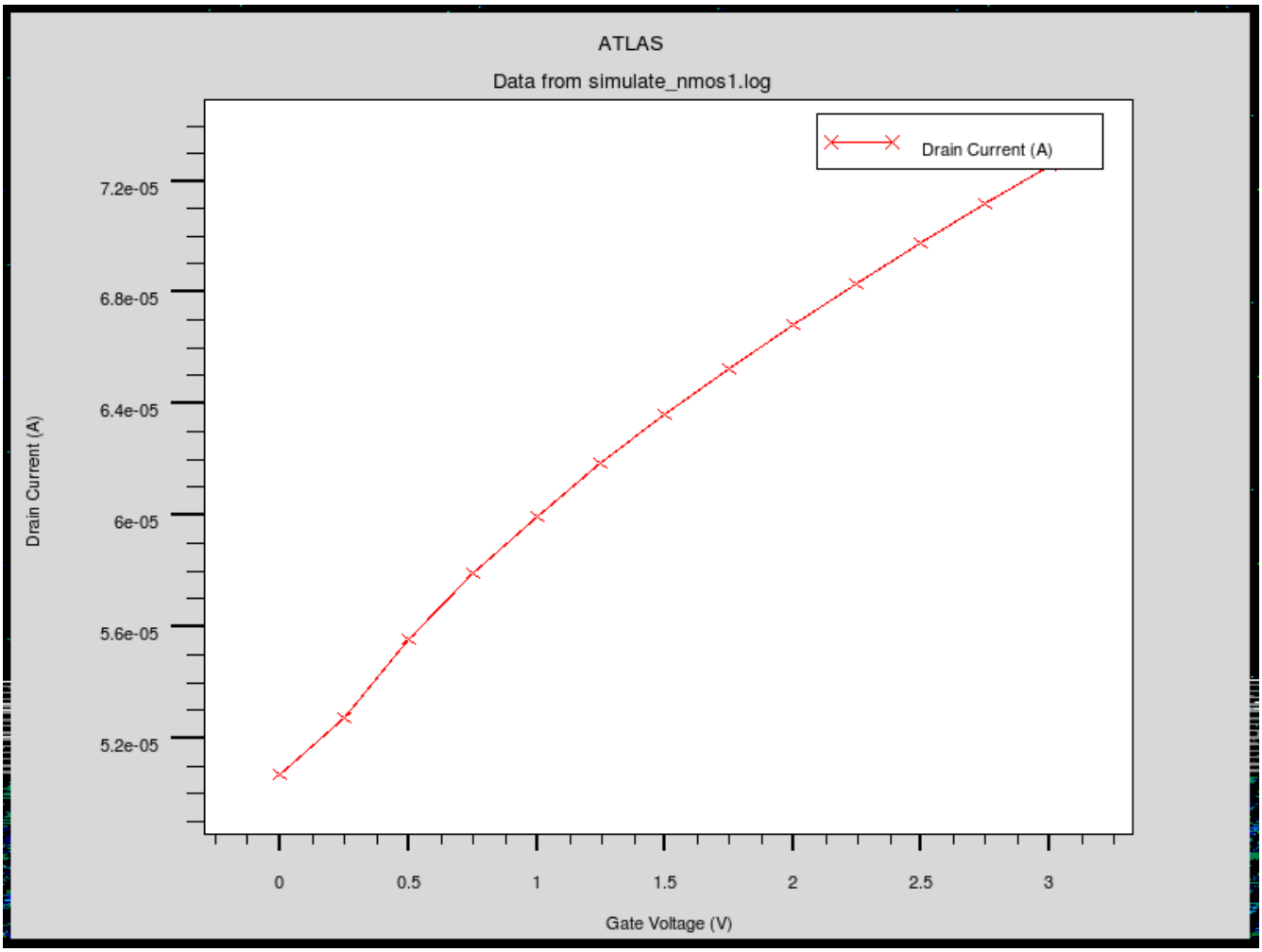

ATHENA
Data from mos1ex01_0.str
Microns
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0
0.2
0.4
0.6
0.8
1
Microns
Net Doping (/cm3)
20.7
20
19.3
18.6
17.8
17.1
16.4
15.7
15
Materials
Silicon
SiO2
Polysilicon
Aluminum
Electrodes

3. Simulation of the the device characteristics

The operating characteristics can be tested and obtained by using ATLAS. The following statements can be used for performing a Vt Test on the NMOS device we fabricated.

```
go atlas

# set material models
models cvt srh print

contact name=gate n.poly
interface qf=3e10

method newton
solve init

# Bias the drain
solve vdrain=0.1

# Ramp the gate
log outf=simulate_nmos1.log master
solve vgate=0 vstep=0.25 vfinal=3.0 name=gate
save outf=simulate_nmos1.str

# plot results
tonyplot  simulate_nmos1.log -set simulate_nmos1_log.set

# extract device parameters
extract name="nvt"
(xintercept(maxslope(curve(abs(v"gate"),abs(i."drain")))) \
    - abs(ave(v"drain"))/2.0)
extract name="nbeta"
slope(maxslope(curve(abs(v"gate"),abs(i."drain")))) \
    * (1.0/abs(ave(v"drain")))
extract name="ntheta" ((max(abs(v"drain")) *
$"nbeta")/max(abs(i."drain"))) \
    - (1.0 / (max(abs(v"gate")) - ($"nvt")))
    -
```

On execution, this generates an Id vs Vg plot:



Hence, various parameters can be studied with ATLAS

Thank you.

I hope this was helpful. For further queries and clarifications, you can reach me at sanket@ieee.org