DIPARTIMENTO
DI INGEGNERIA
DELL'INFORMAZIONE

UNIVERSITÀ
DEGLI STUDI
DI PADOVA

Sentaurus Tutorial

MOS capacitor simulation

Andrea Neviani

Main Tools

- Sentaurus Structure Editor
  - line command: **`sde`**
  - aim: graphical tool to create device geometry and doping profiles (either directly or through process emulation) and set mesh specifications
- Sentaurus Device
  - line command: **`sdevice [command_file_name]`**
  - aim: numerical simulation of semiconductor devices
    - numerically solves Poisson and continuity equations on the mesh defined by SDE
    - yields value of electric potential, electron/hole concentration and other physical quantities (electric field, electron/hole current, G/R rate, ...) at any node in the device mesh

Mail Tools

- Sentaurus Visual
  - line command: **`svisual`**
  - aim: visualize results derived by Sentaurus Device (and othe TCAD tools) in one, two and three dimensions
- Sentaurus Workbench
  - line command: **`swb`**
  - aim: graphical interface that allows to prepare, launch and analyze the results of TCAD Sentaurus simulation tools

TCAD Sentaurus Online Tutorial

- Detailed online tutorial on the full TCAD Sentaurus tool suite
- To open the tutorial
  - launch Sentaurus Workbench

    ```
    swb &
    ```

  - menu Help → Training

Tutorial #3: MOS capacitor

- Open a command line terminal, and:

cd /nfsd/cadusers/*your-username*

mkdir TCAD

cd TCAD

- Launch firefox (or another web browser), go to the course webpage on eLearning and download the file **tcad-user.sh** into directory TCAD
- Then, from the terminal, execute the command:

source tcad-user.sh

- and launch Sentaurus Workbench:

sbw &

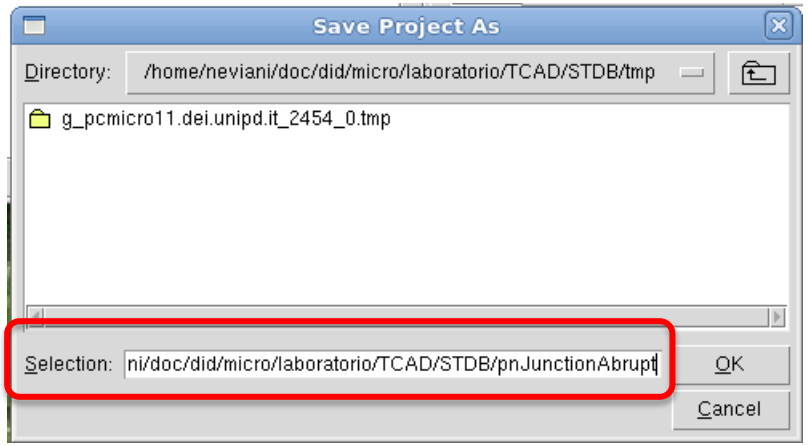
- In the left panel of the SWB window you should see a list of folders, the first of which is your working folder:

/nfsd/cadusers/*your-username*/TCAD/STDB

Tutorial #3: MOS capacitor

- Create your first project
  - in the upper menu bar, click on Project → New → New Project
  - then, Project → Save As → Project ...
  - in the Selection field of the dialog block, write the project save path, then click OK

`/nfsd/cadusers/`*`your-username`*`/TCAD/STDB/MOScap`

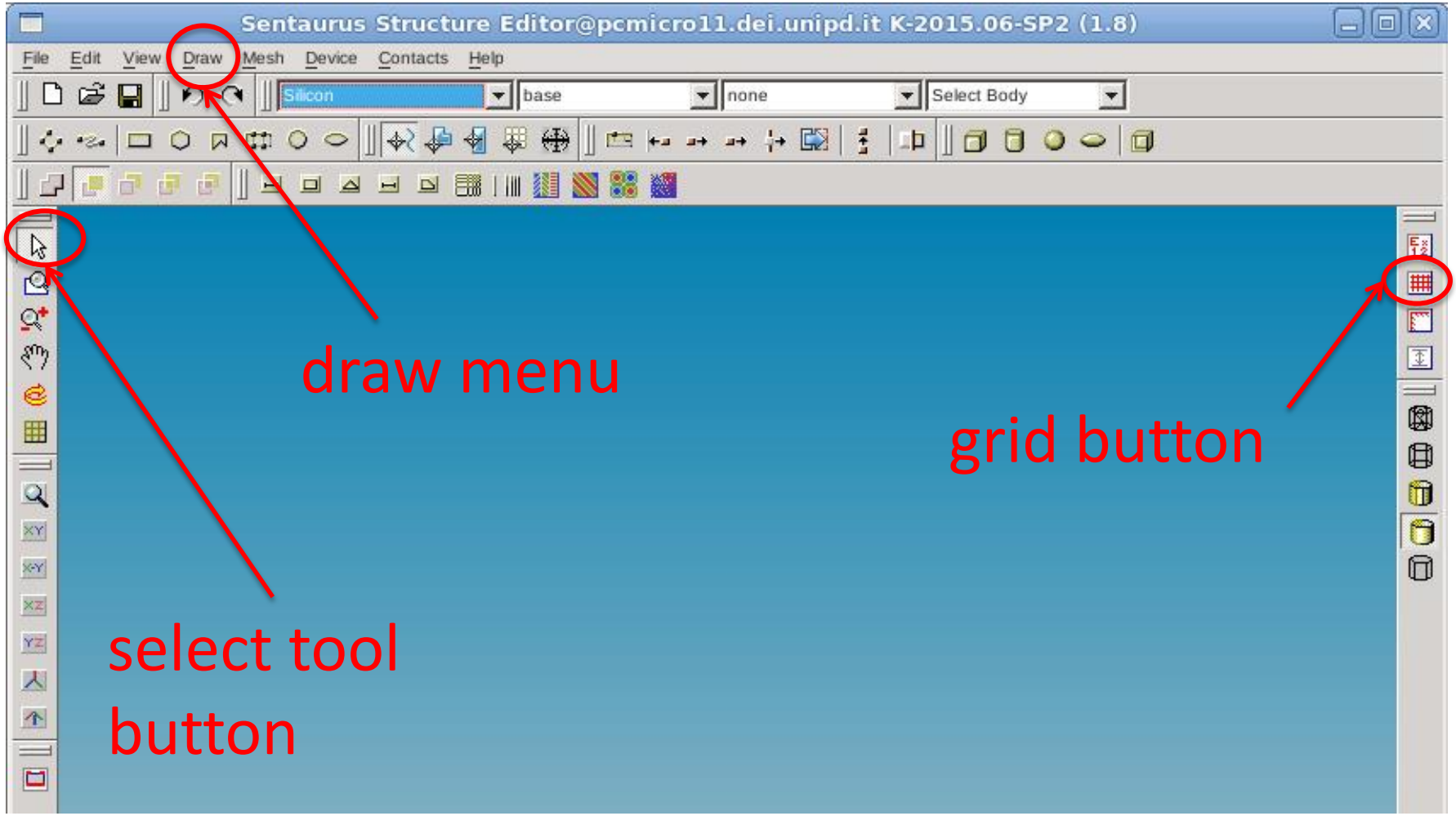


Tutorial #3: define device geometry

- We will now define the device geometry, doping and mesh using SDE
- Launch SDE from the upper menu bar
  - Extensions → Run Sentaurus Structure Editor
  - SDE window should appear
- Prepare SDE for editing
  - choose Select tool on left icon bar (white arrow)
  - click on the grid symbol in the right icon bar, then click Show and Close in the dialog box; a 1μm x 1μm grid appears in the drawing area
  - in the upper menu bar, click on Draw, then **de-select** Auto Region Naming and **select** Exact Coordinates

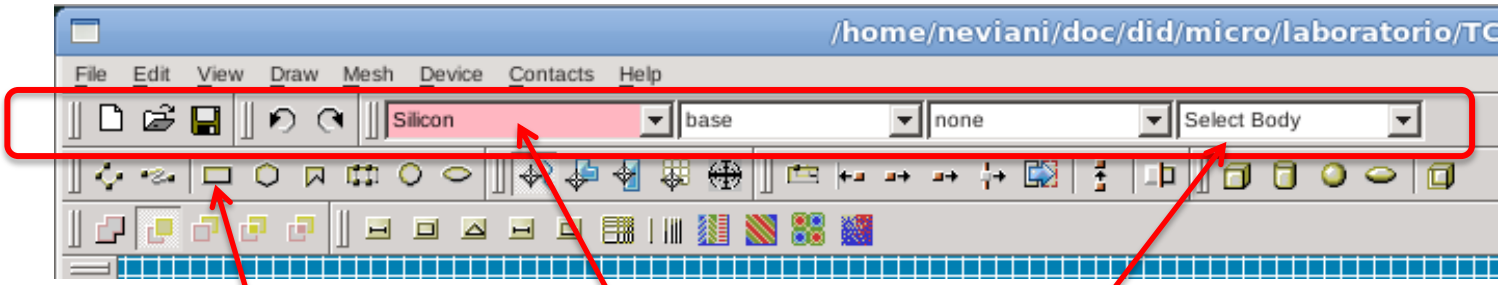
DIPARTIMENTO
DI INGEGNERIA
DELL'INFORMAZIONE

Tutorial #3: define device geometry



Tutorial #3: define device geometry

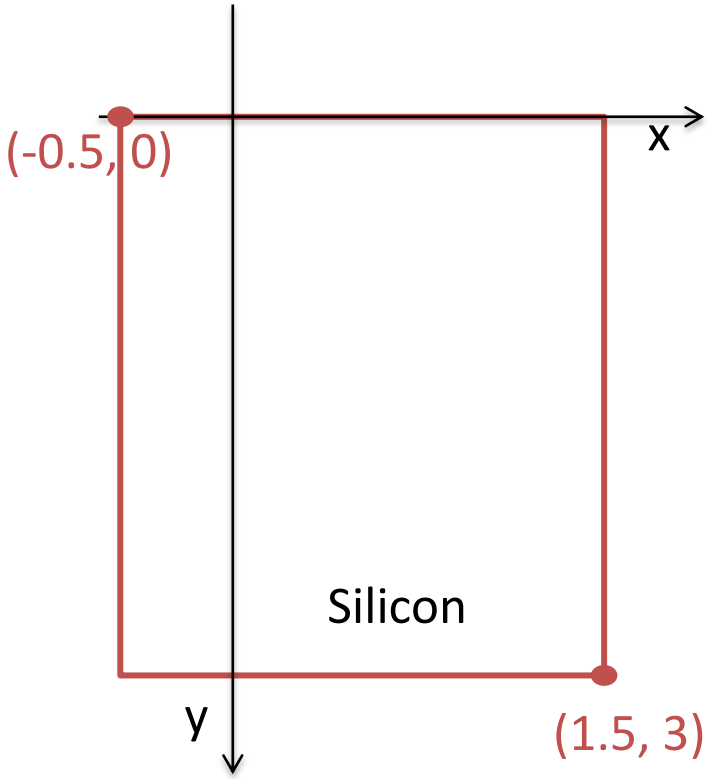
- Define the device geometry:
  - In the upper icon bar



  - make sure that Silicon is selected in the Material List pull-down menu and that Selection Type is set to Select Body
  - then, select the Create Rectangular Sheet Region icon

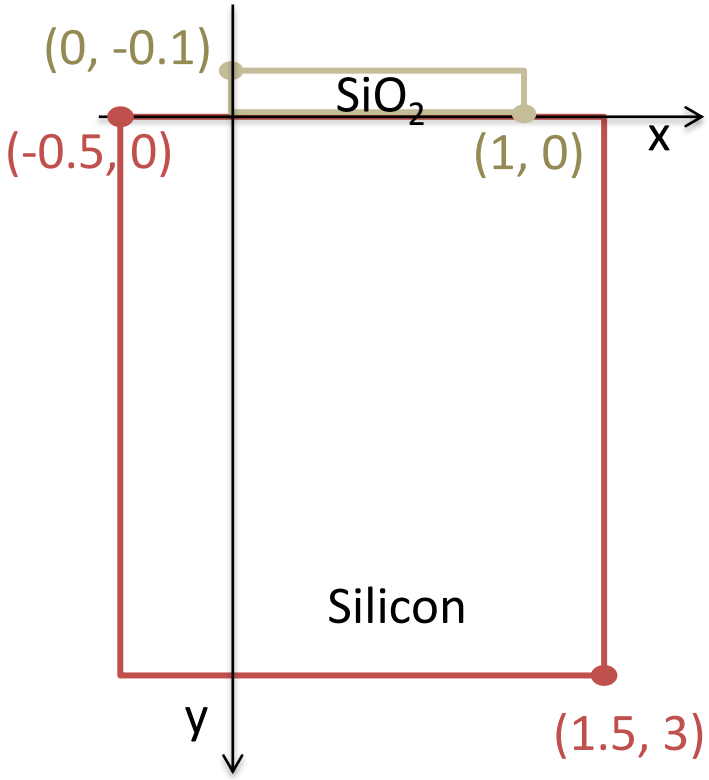
Tutorial #3: define device geometry

- Let us first define the Silicon substrate geometry:
  - left-click and drag the mouse to draw a rectangle
  - insert the substrate upper left corner and lower right corner coordinates in the Exact Coordinates dialog window, then click OK
    - First Vertex (-0.5, 0); Second Vertex (1.5, 3)
  - Enter Region Name (pSub) and click OK



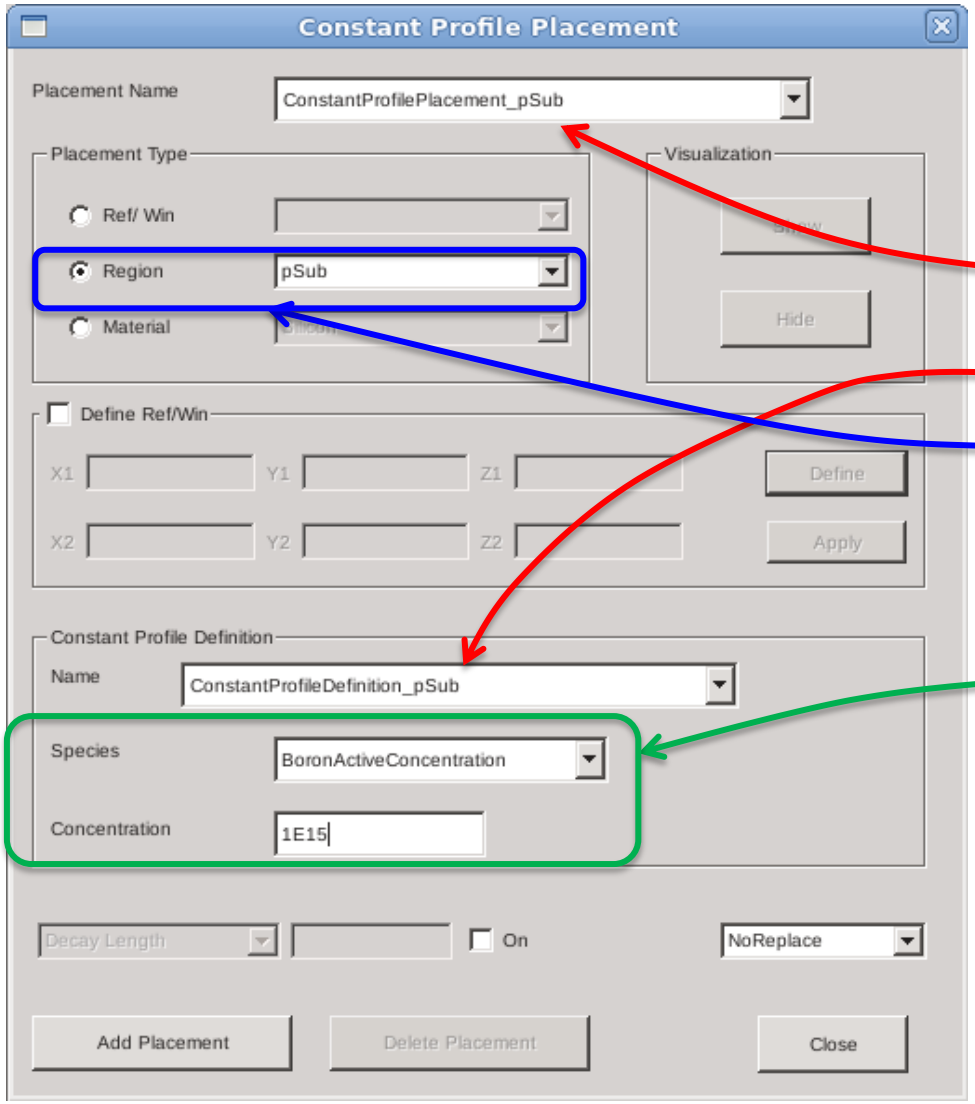
Tutorial #3: define device geometry

- Then, define the gate oxide geometry:
  - make sure that SiO_2 is selected in the Material List pull-down menu
  - as before, left-click and drag the mouse to draw a rectangle and insert the gate oxide region coordinates
    - First Vertex (0, -0.1); Second Vertex (1, 0)
  - Enter Region Name (gateOxide) and click OK
  - Save the model (File → Save Model) with name MOScap



DIPARTIMENTO
DI INGEGNERIA
DELL'INFORMAZIONE

Tutorial #3: add uniform doping



- In the upper menu bar, click Device → Constant Profile Placement...
- Add the suffix pSub to the Placement Name and to the Profile Definition
- Select Region as profile type and pSub in the pull-down menu
- Choose Boron Active Concentration as dopant Species and 1E15 as Concentration
- Finally, click Add Placement and Close

Tutorial #3: create contacts

- To complete the device definition, we will now add the gate contact (G) and the substrate ohmic contact (B) on top of the gate oxide and at the bottom of the substrate pSi region, respectively.

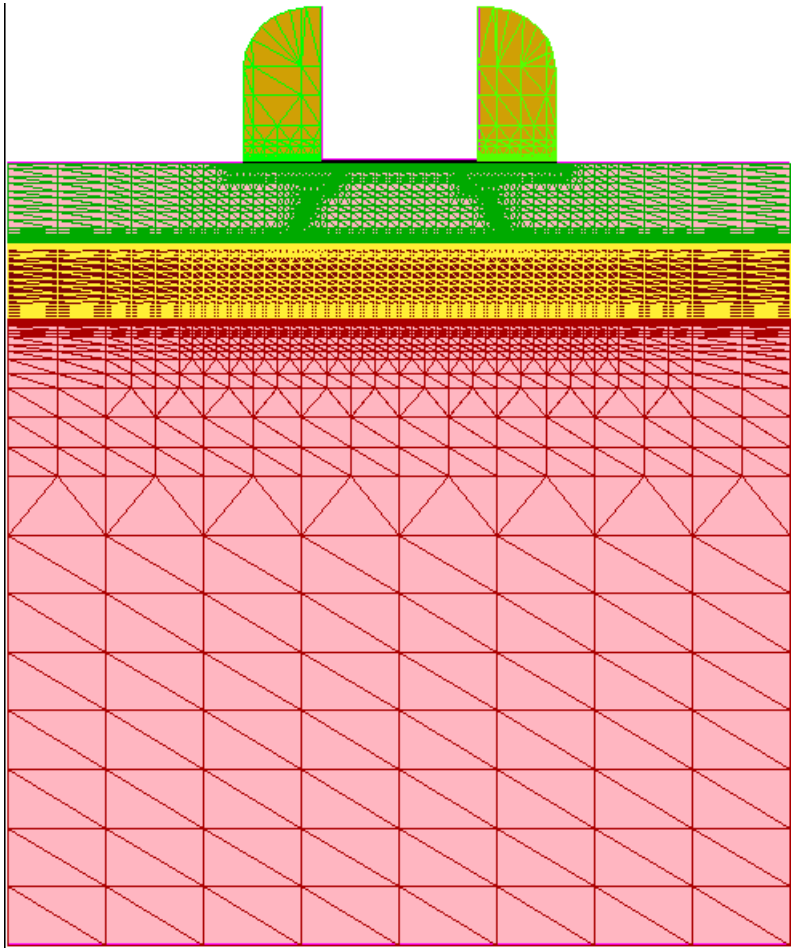
Tutorial #3: create contacts

- Contact creation requires two steps
  1. Define the list of contacts of the device
  2. Associate (set) each contact to an edge of the device
- Define contacts
  - You will now define contacts **G** and **B** of the MOS capacitor
  - Select menu **Contacts → Contact Sets**
  - In the dialog box, enter the name of the contact in the **Contact Name** field
  - Click **Set** to add the contact to the **Defined Contact Set**
  - Repeat for each contact, then click Close

Tutorial #3: create contacts

- Set contacts on existing edges
  1. You will now associate contacts G and B to the oxide region top edge and to the substrate region bottom edge
  2. Select menu **Contacts → Contact Sets**
  3. Select contact G from the Defined Contact Sets
  4. Click Activate and then Close
  5. In the Selection Level list, choose Select Edge
  6. Click the select button (white arrow) on the left panel
  7. Click the top edge of the oxide region; that's where we want to place the gate contact
  8. Select menu **Contacts → Set Edges**.
  9. Repeat steps 2 to 8 to place contact B on the bottom edge of the substrate region
- Save the model

Tutorial #3: mesh generation



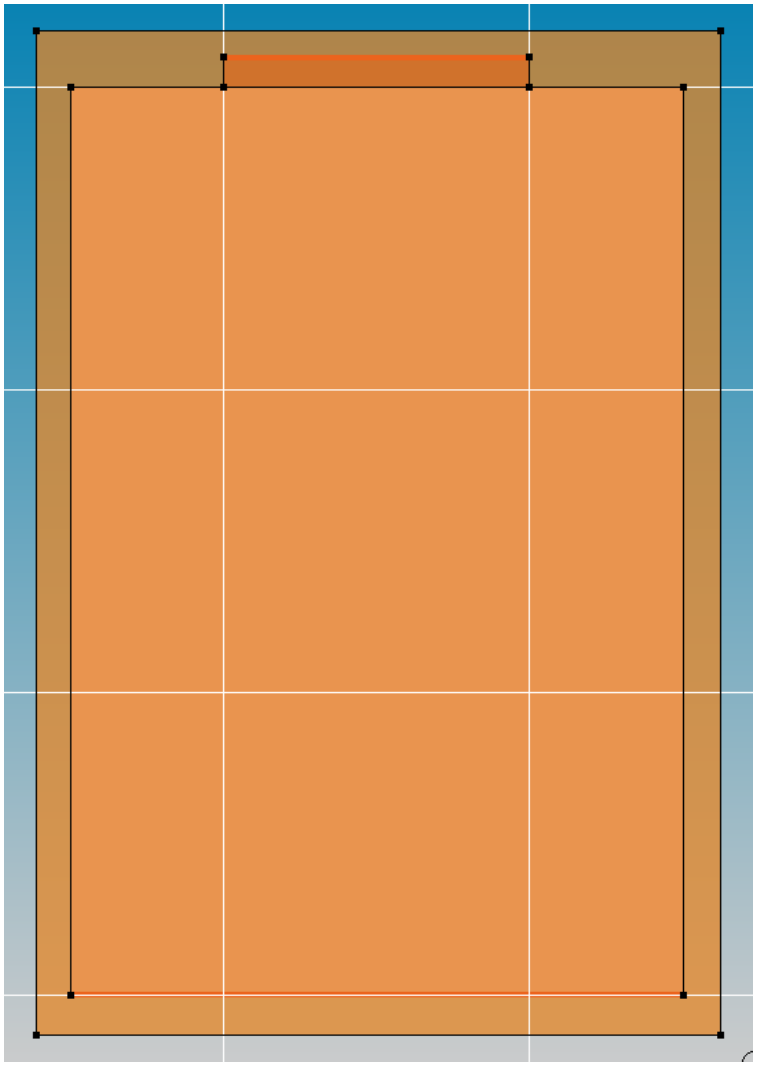
example of device geometry with mesh

- Numerical simulation tools require the generation of a mesh in order to solve numerically the semiconductor equations
  - a mesh is a graph with nodes and edges that covers the entire device area
  - differential equations are discretized on the mesh nodes becoming algebraic (nonlinear) equations

Tutorial #3: mesh generation

- Mesh generation is usually performed in two steps:
  - definition of global mesh settings
  - definition of regional (local) refinements
- Global mesh setting
  - defines mesh controls that are applied to the entire device structure
  - Select Mesh → Define Ref/Eval Window → Rectangle and draw a rectangle covering the entire device with some margins
  - In the dialog box, do not change the vertex coordinates and insert RefWin.all as Ref/Eval Window Name
  - The global mesh window should appear as shown in the following slide

Tutorial #3: mesh generation

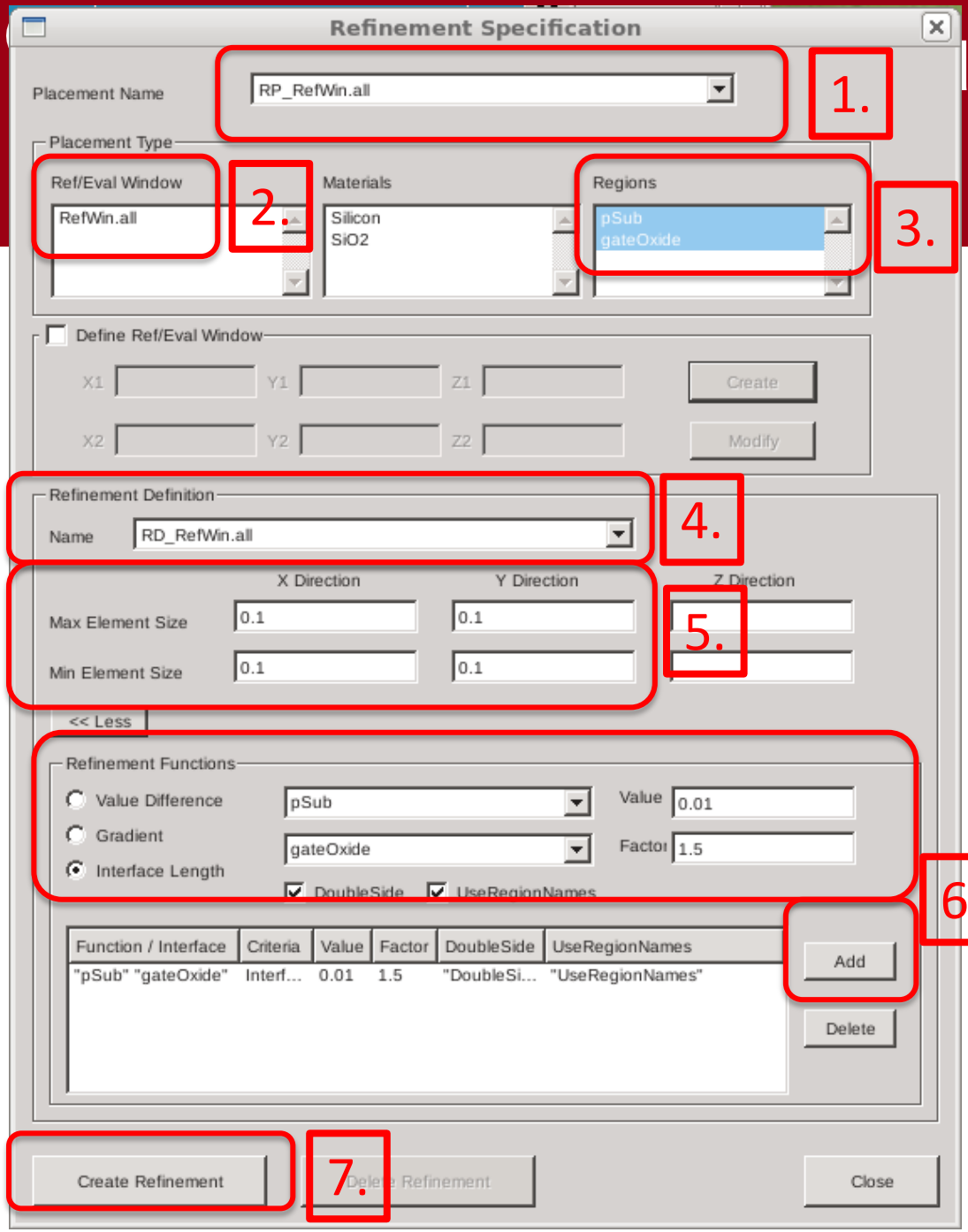


appearance of MOScap device model after global mesh definition

Tutorial #3: mesh generation

- We will now set the global mesh specifications
- Select Mesh → Refinement Placement
- In the Refinement Specification dialog box (see next slide):
  1. change Placement Name to RP_RefWin.all
  2. select RefWin.all in the Ref/Eval Window panel (this identifies the area where refinement specifications will be applied)
  3. select pSub and gateOxide in the Region panel
  4. change the Refinement Definition Name to RD_RefWin.all
  5. insert 0.1 in the Max/Min Element Size fields (X and Y directions only)
  6. click on More, then, as Refinement Function, choose Interface Length, select pSub and gateOxide from the pull-down menus, insert 0.01 and 1.5 as (sensitivity) Value and Factor, respectively, then click Add
  7. finally, click on Create Refinement and Close
  8. Save the model

torial #3: mesh generation



refinement specifications for the global mesh

Tutorial #3: mesh generation

- We will now define the regional refinements
  - the area immediately below the Si/SiO_2 interface requires a finer mesh due to the expected strong gradients of main physical quantities in the space charge region (carrier concentration, electric field, ...)
  - to this aim, we shall first define a refinement window below the Si/SiO_2 interface and then set new refinement specifications to that region
  - Select Mesh → Define Ref/Eval Window → Rectangle and draw a rectangle approximately below the interface
  - In the dialog box, insert the following coordinates
    - First Vertex X: -0.1 Y: 0
    - Second Vertex X: 1.1 Y: 1
    - Ref/Eval Window Name RefWin.Channel

Tutorial #3: mesh generation

- Now let's set the mesh specification in the RefWin.Channel window
- Select Mesh → Refinement Placement
- In the Refinement Specification dialog box:
  - change Placement Name to RP_RefWin.Channel
  - select RefWin.junction in the Ref/Eval Window panel
  - change the Refinement Definition Name to RD_RefWin.Channel
  - insert 0.1 and 0.05 in the Max/Min Element Size fields for the X direction, 0.05 and 0.01 for the Y direction
  - finally, click on Create Refinement and Close
  - Save the model

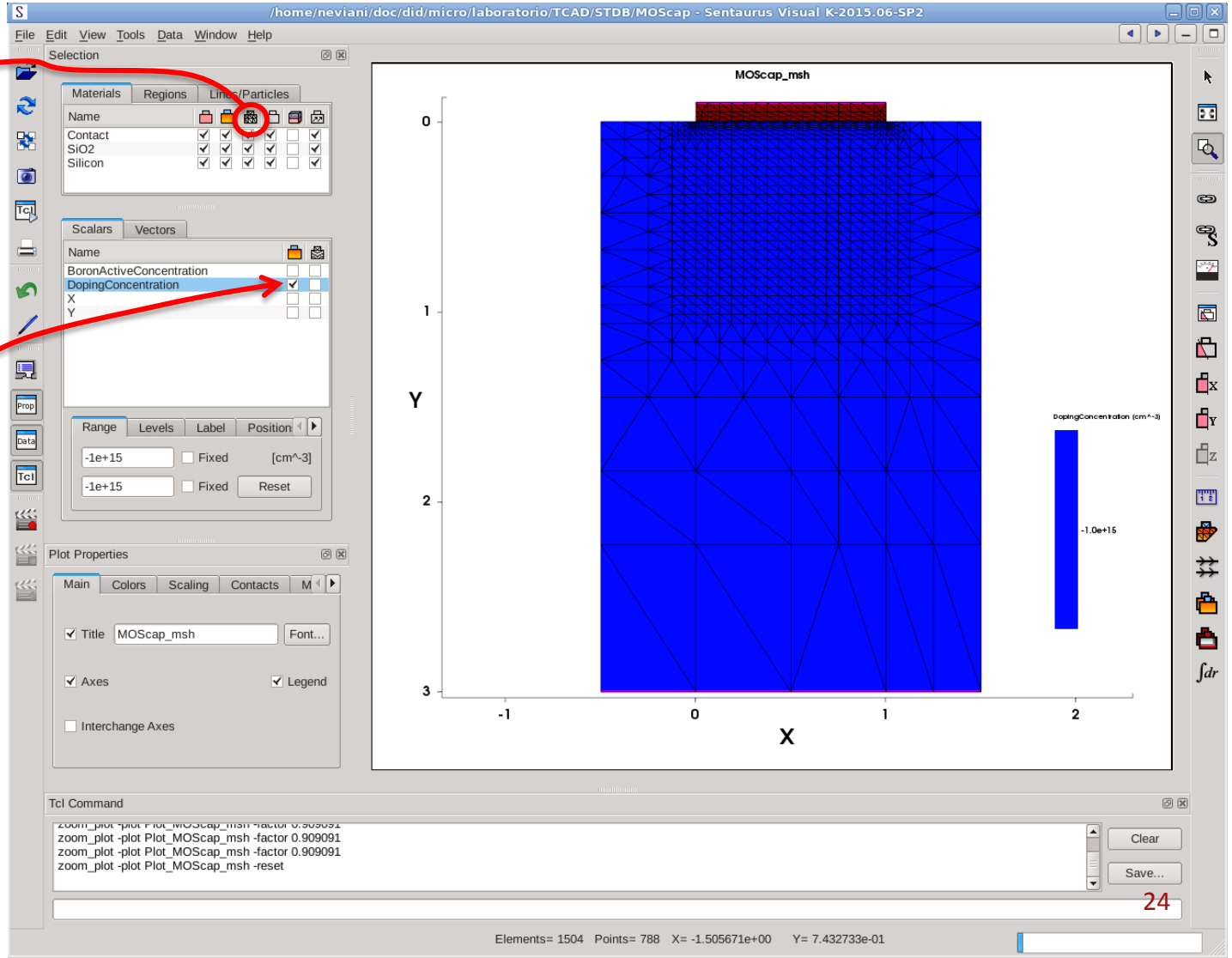
Tutorial #3: mesh generation

- We finally generate the mesh
  - Select Mesh → Build Mesh
  - In the dialog box, make sure that the file full path in the Save Grid to File field is correct (/nfsd/cadusers/your-username/TCAD/STDB/MOScap/MOScap)
  - Select SVisual as Mesh Viewer, then click Build Mesh
  - After mesh generation is completed, SVisual (the plotting tool for visualizing TCAD data) opens, showing the device structure and mesh (see next page)

Tutorial #3: mesh generation

click the Mesh icon to show the device mesh

check the Doping Concentration box to show active doping density



Tutorial #3: device simulation

- We'll now use Sentaurus Device to run a series of drift-diffusion simulations of the MOS capacitor
- We'll use the SWB interface to setup and execute the simulation
- Sentaurus Device actions and behavior are driven by statements contained in the command file
- The command file contains six sections:
  - File → input and output files
  - Electrode → contact types and boundary conditions (V, I)
  - Physics → physical models used in the simulation
  - Plot → variables to be saved in the plot file
  - Math → numerical simulation parameter
  - Solve → simulation commands

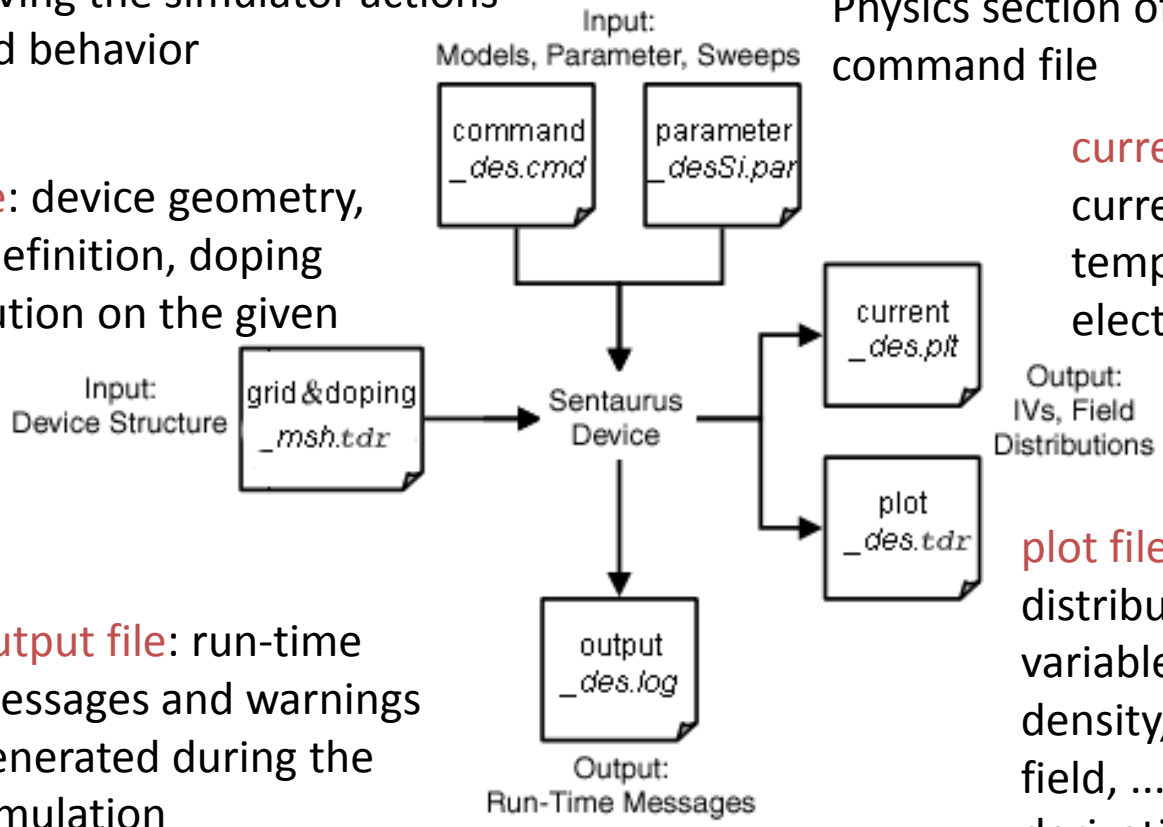
Tutorial #3: device simulation

command file: statements driving the simulator actions and behavior

parameter file: parameters of physical models defined in the Physics section of the command file

grid file: device geometry, mesh definition, doping distribution on the given mesh

current file: voltages, currents, charges, temperature at device electrodes



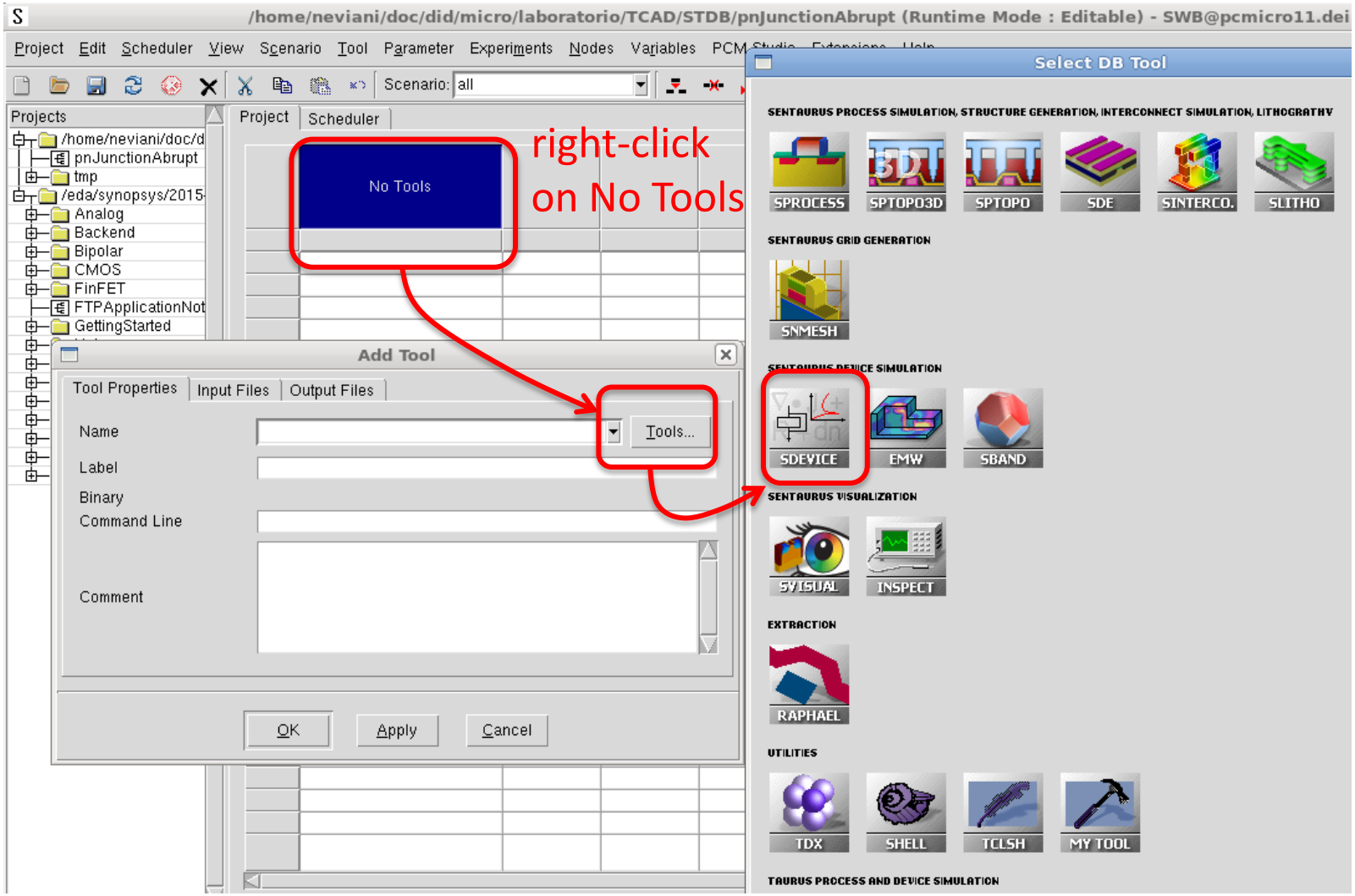
output file: run-time messages and warnings generated during the simulation

plot file: spatially distributed solution variables (electron/hole density, potential, electric field, ...) and their derivatives

Tutorial #3: device simulation

- In the SWB window
  - right-click on the header **No Tools** of the spreadsheet leftmost column and then click on Add... (see next page)
  - then, in the dialog box, click on Tools..., then, in the Select DB Tool window, click SDEVICE, then OK, and again OK
  - the SDEVICE icon now appears on the header of the leftmost column

Tutorial #3: device simulation



Tutorial #3: device simulation

- Command file creation
  1. Right-click on the SDEVICE icon, then Edit Input → Commands...
  2. An empty file opens; the file name is **`sdevice_des.cmd`** and it is located in the MOScap working directory
  3. In the web browser, open the course webpage on eLearning and download the file **`command_file1.txt`** into directory TCAD
  4. Open the file, select all the file content, then copy and paste the content in the **`sdevice_des.cmd`** empty file
  5. Save the file; before exiting, we'll take a closer look to the file content

Tutorial #3: device simulation

```
File {
        Grid="MOScap_msh.tdr"
        Plot="@tdrdat@"
        Current="@plot@"
        Output="@log@"
}
Electrode {
   {name="G" voltage=-1.0
      barrier=-0.51}
   {name="B" voltage=0.0}
}
Physics {
   Mobility( DopingDep
     HighFieldSaturation Enormal )
   EffectiveIntrinsicDensity(
     OldSlotboom )
   Recombination( SRH(DopingDep) )
}
```

File section:

- name of file containing device geometry, doping and grid
- output files names based on simulation node

Electrode section:

- definition of electrode type, features and boundary condition (V or I)

Physics section:

- declaration of physical models to be used in the simulation

Tutorial #3: device simulation

```
Plot { eDensity hDensity
       eCurrent hCurrent
       ...
}
Math {
       ...
}
Solve {
  Coupled ( Iterations= 150){
    Poisson }
  Coupled { Poisson Electron
    Hole }
* Gate voltage sweep
  Quasistationary(InitialStep=0.01
    MinStep= 1e-5 MaxStep= 0.2
    Goal{Name="G" Voltage=@VGB@})
    {Coupled {Poisson Electron
     Hole } } }
```

Plot section:

- internal variables to be saved

Math section:

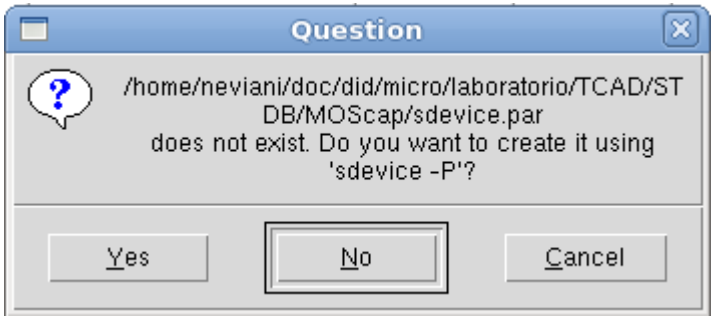
- simulator numerical solver parameters

Solve section:

- sequence of simulation commands activated sequentially

Tutorial #3: device simulation

- Parameter file
  - Optional file used to specify parameters of physical models different from the default values
  - Not needed when the device regions are made with standard materials contained in SDE material list
  - Right-click on the SDEVICE icon, then Edit Input → Parameters...

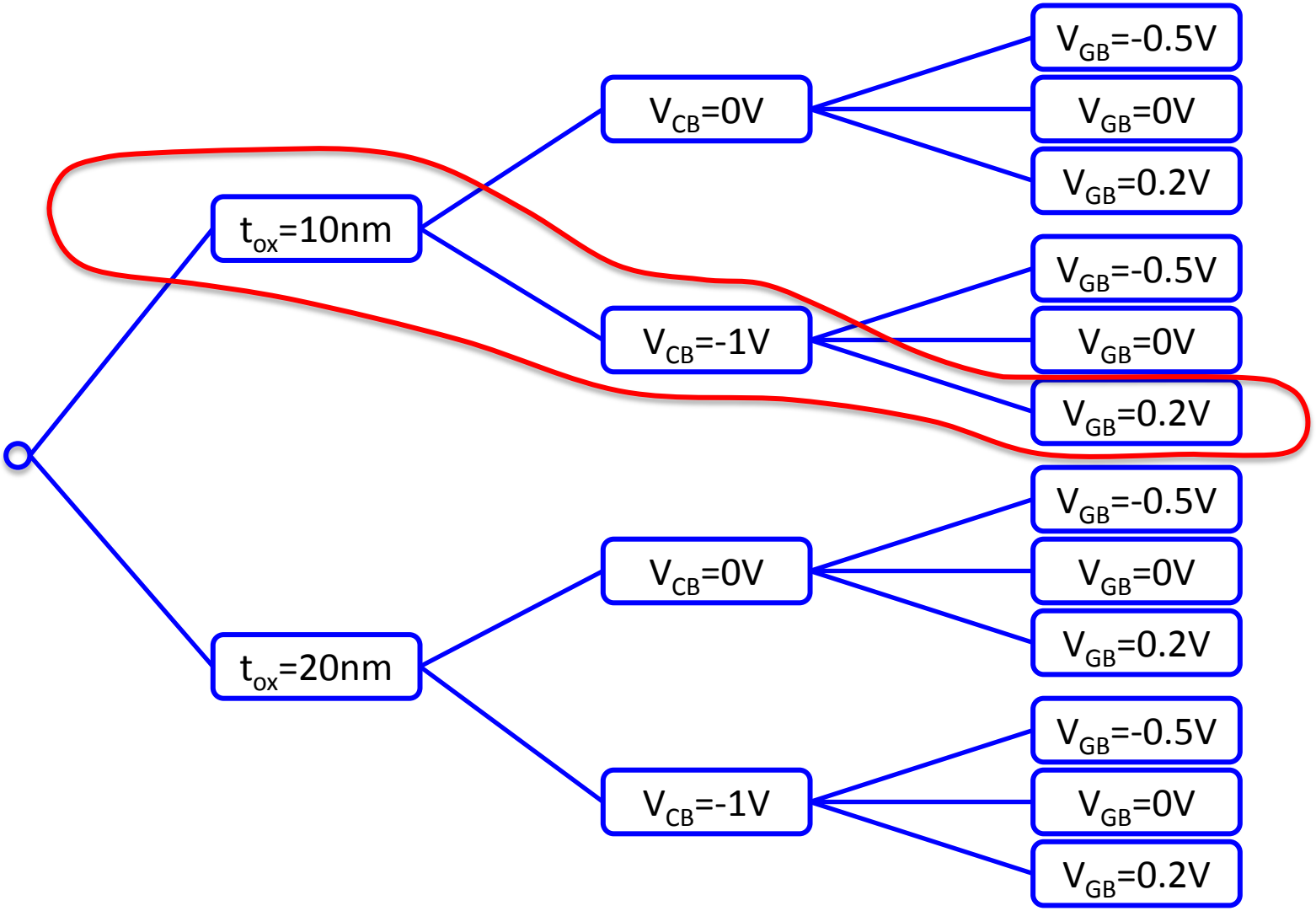


  - left-click on No
  - an empty file opens; save the empty file and exit

Tutorial #3: device simulation

- Device simulators are normally used to execute a set of experiments (simulations) where one or more parameters are changed
- The simplest example is when e series of voltage values are applied to a terminal
  - one dc simulation per voltage value is executed
- In general, multiple parameters are changed, originating a graph tree where each root-leaf path corresponds to a combination of parameters values
  - Example: three-terminal MOS capacitor where t_{ox}, V_{GB} and V_{CB} are changed

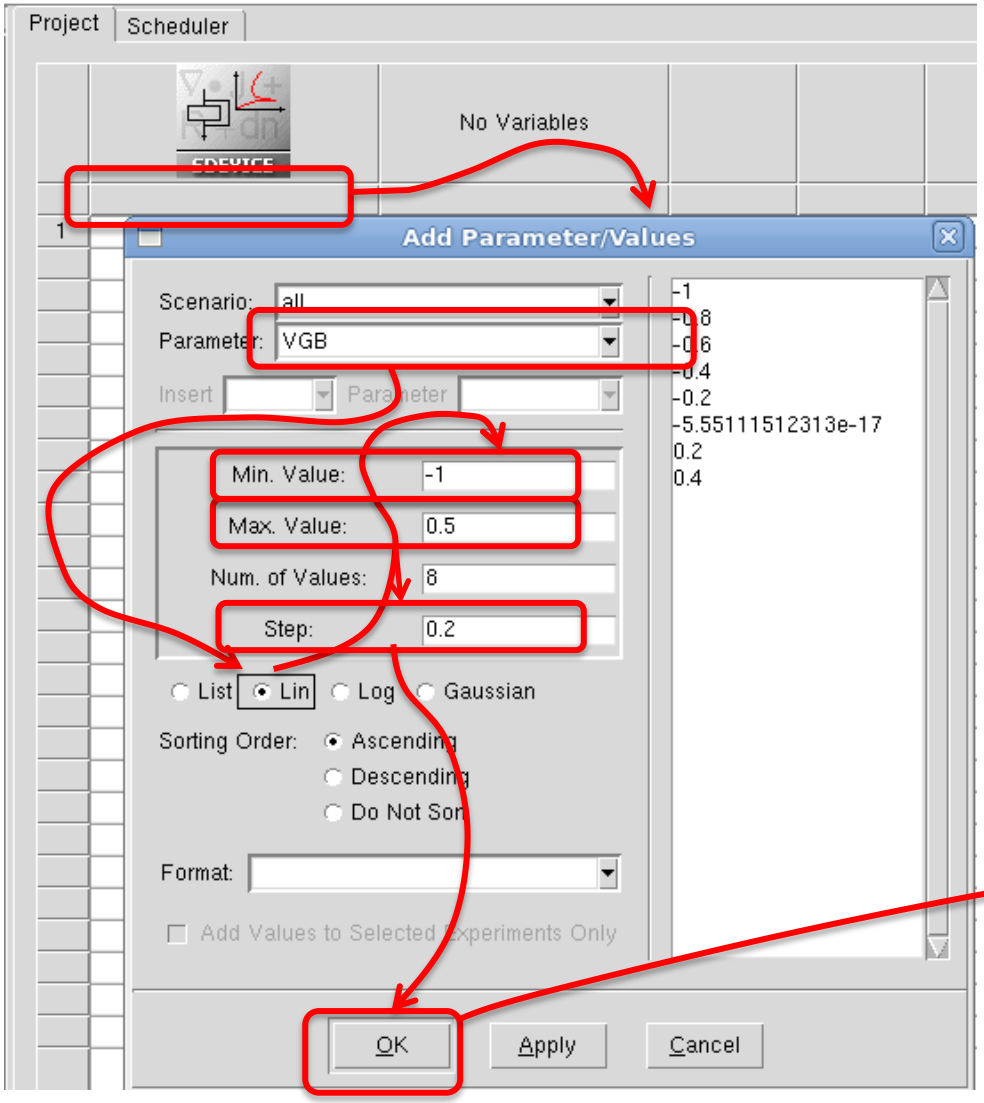
Tutorial #3: device simulation

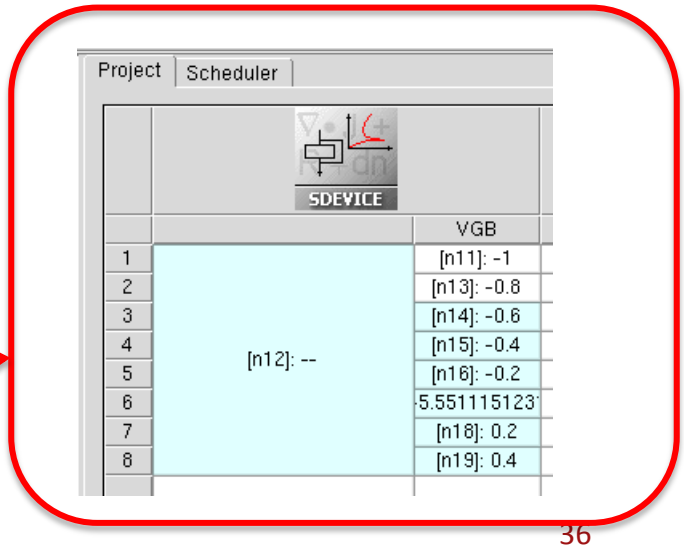


Tutorial #3: device simulation

- We'll define a simple set of experiments where V_{GB} is varied to take on the values listed below:
  - V_{GB} = {-1, -0.8, -0.6, -0.4, -0.2, 0, 0.2, 0.4} V
- In SWB window, right-click on the cell below the SDEVICE icon, then left-click on Add Parameter/Value
- Write VGB in Parameter field
- Select the Lin button, to insert the list of VGB values as a linearly-spaced sequence
- Write -1 as Min Value, 0.5 as Max Value, and 0.2 as Step, then click OK

Tutorial #3: device simulation





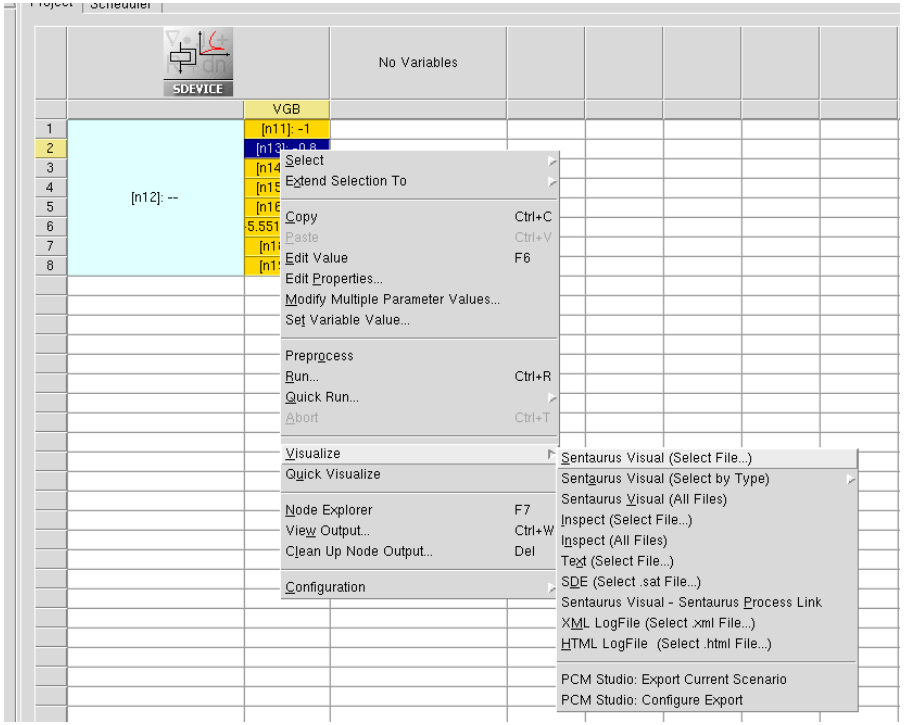
Tutorial #3: device simulation

- To execute a dc drift-diffusion simulation of the MOS capacitor in each of the seven nodes defined before:
  - select the nodes (click on the first node, then Shift+click to add the remaining nodes)
  - right-click on anyone of the selected nodes
  - click Run; if asked to, save the project; then, make sure Preprocess, then run button is selected and click OK
  - after a few seconds, the Project Log window opens showing messages regarding the simulations being run
  - as the simulations progress, the node cell color changes according to the simulation status
  - when all node cells are yellow, the simulations are successfully completed

Tutorial #3: post-processing

Simulation results can be analyzed with the help of SVISUAL

- Right-click the node corresponding to VGB=-0-8V → `Visualize` → `Sentaurus Visual (Select File…)`
- A dialog box with two files opens:
  - `n13_des.plt`
  - `n13_des.tdr`
- Note that the node number might be different
- Select the file with tdr extension and click OK
- Sentaurus Visual window should open



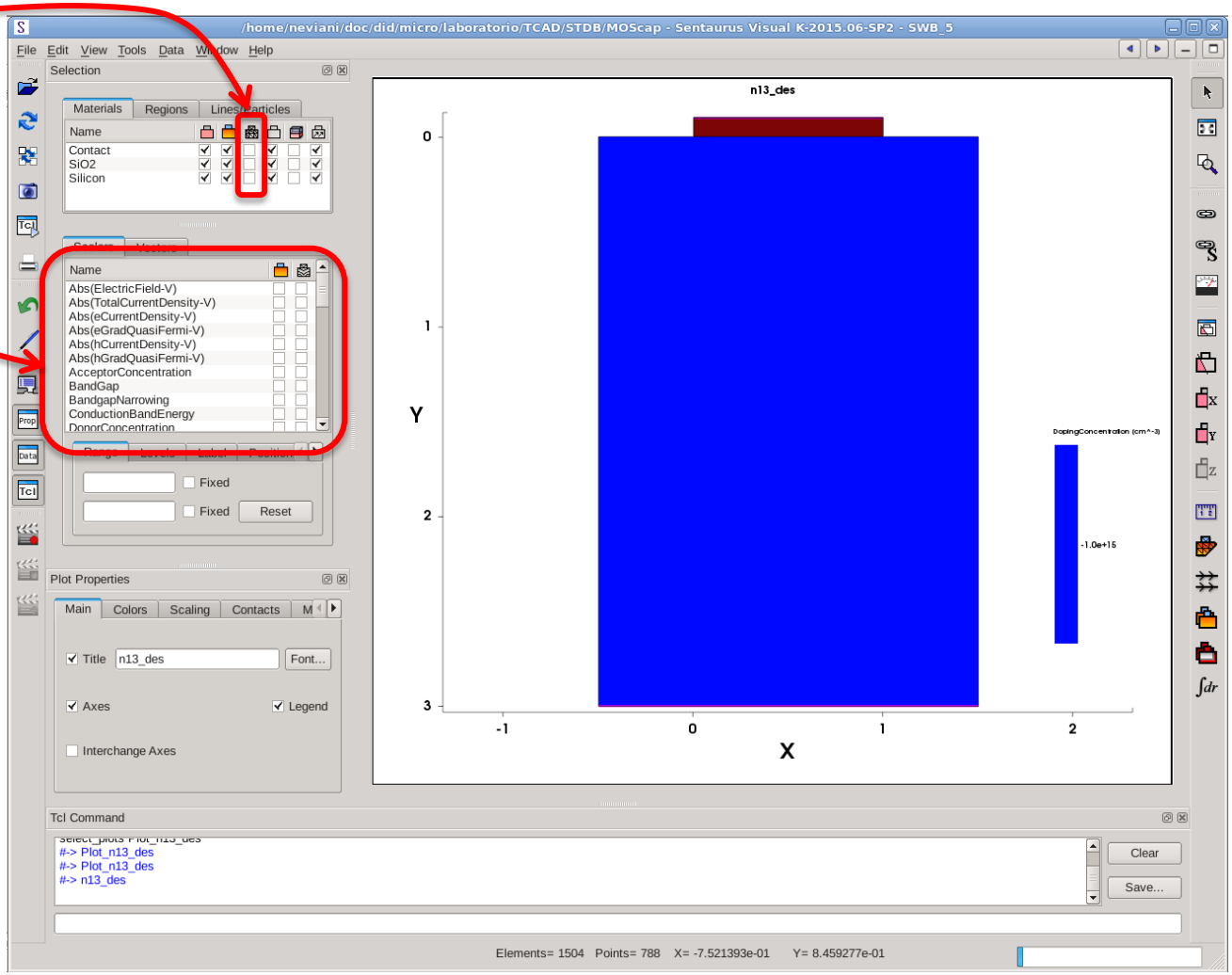
Tutorial #3: post-processing

activate grid visualization

select physical quantity to visualize (default: doping concentration)

try to visualize:
ElectrostaticPotential
eDensity
hDensity

- Sentaurus Visual window



Tutorial #3: ac simulation

- In single-device physical simulation, Sentaurus Device solves the Poisson and Continuity equation using the drift-diffusion transport model on a 2D or 3D portion of the device geometry
- In mixed-mode simulation, the device to be physically simulated is included in a circuit with ideal lumped components (resistors, capacitors, inductors, voltage/current generators) and active components with compact electrical models (transitors, diodes)
- Simulation of the ac response of a physical device is a special case of mixed-mode simulation

Tutorial #3: ac simulation

- In its simplest form, small-signal ac simulation requires voltage (or current) dc and ac signal generators and the device to be physically simulated
- The Solve section of the command file contains statements to:
  - solve the device equation at equilibrium
  - bring the device dc voltages to the initial value
  - ramp the device dc voltage from the initial to the final value
  - inside the ramp statement, an ac solution statement to find the small-signal ac response of the device for every dc voltage defined in the ramp

Tutorial #3: ac simulation

- In order to run a small-signal ac simulation we need to change the command file:
  - adding a **Device** section for each device to be physically simulated
  - adding a **System** section that defines the mixed-mode circuit
  - change the **Solve** section to define a step-by-step variation of a dc voltage/current and a linearized ac analysys to be performed in a subset of dc points

Tutorial #3: ac simulation

- In the SWB window, right-click on the SDEVICE icon → Edit Input → Commands...
- The sdevice_des-cmd command file opens; click on File → Save As... and save the file with a new name (e.g. MOScap_ac.cmd)
- Select and delete the content of the file
- Download command_file2.txt from the eLearning website and copy its content into MOScap_ac.cmd file
- Save the file and before exiting take a closer look to the Device nMOScap section and the System section

Tutorial #3: ac simulation

```
Device nMOScap {
  Electrode {
    {name="G" voltage=0.0
barrier=-0.51}
    {name="B" voltage=0.0}  }
  File {
    Grid="MOScap_msh.tdr"
    Plot="@tdrdat@"
    Current="@plot@"  }
  Physics {
    Mobility( DopingDep
      HighFieldSaturation Enormal)
    EffectiveIntrinsicDensity(
      OldSlotboom )
    Recombination(SRH(DopingDep) )
  } }
```

Device section:

- needed for every device to be physically simulated
- includes an Electrode, a File and a Physics section that define the properties of the device
- a unique device name (nMOScap in this example) identifies the device model

Tutorial #3: ac simulation

```
System {
  nMOScap C1 (G=g B=b)
  Vsource_pset vg ( g 0 ){ dc = 0}
  Vsource_pset vb ( b 0 ){ dc = 0}
}
```

System section:

- defines the netlist of the circuit to be simulated
- in this example, the terminals G and B of nMOScap device defined in the Device section with the same name are connected to the circuit nodes g and b
- a voltage generator vg is connected between node g and ground (node 0)
- a second voltage generator is connected between node b and ground

Tutorial #3: ac simulation

```
Solve {
 NewCurrentPrefix="init"
 Coupled(Iterations=150){Poisson}
 Coupled{Poisson Electron Hole}


 Quasistationary (
   InitialStep=0.1 Increment=1.3
   MaxStep=0.5 Minstep=1.e-5
   Goal { Parameter=vg.dc
     Voltage=-2.5}
   ){ Coupled { Poisson Electron
Hole } }
  ...
```

Solve section:

- initial solution with boundary conditions specified in the Electrode and in the System sections

- second simulation step to bring the vg voltage source dc voltage to -2.5 V

Tutorial #3: ac simulation

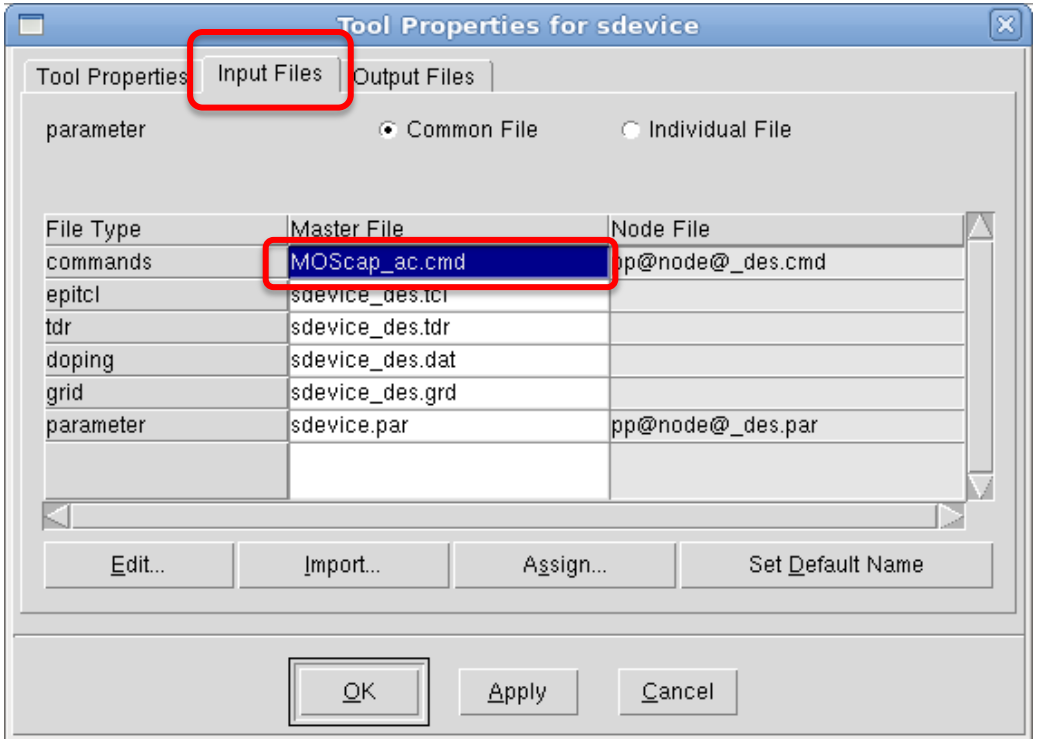
```
Solve {
 ...
 #-ramp gate
 NewCurrentPrefix=""
 Quasistationary (
  InitialStep=0.01 Increment=1.3
  MaxStep=0.05 Minstep=1.e-5
  Goal { Parameter=vg.dc
         Voltage=2.5})
 { ACCoupled (
       StartFrequency=1e3
EndFrequency=1e3 NumberOfPoints=1
Decade
       Node(g b) Exclude(vg vb)
       ACCompute (Time = (Range =
(0 1)  Intervals = 20))
     ){ Poisson Electron Hole }
  } }
```

System section:

- dc voltage of vg generator is ramped from the value reached in the previous step (-2.5V) to 2.5V
- a small-signal ac analysis is coupled with the quasi-stationary dc ramp

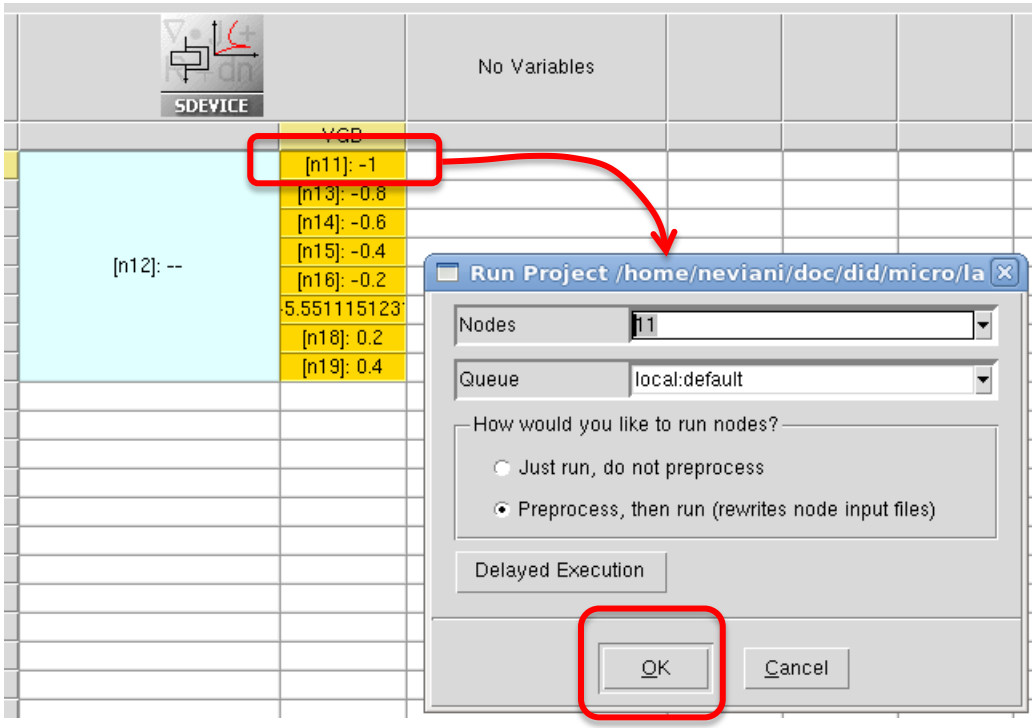
Tutorial #3: ac simulation

- Assign the new command file to SDEVICE
  - right-click on the SDEVICE icon → Properties...
  - in the dialog box, select the Input Files tab
  - in the Master File column change the name of the command file to MOScap_ac.cmd, then click OK



Tutorial #3: ac simulation

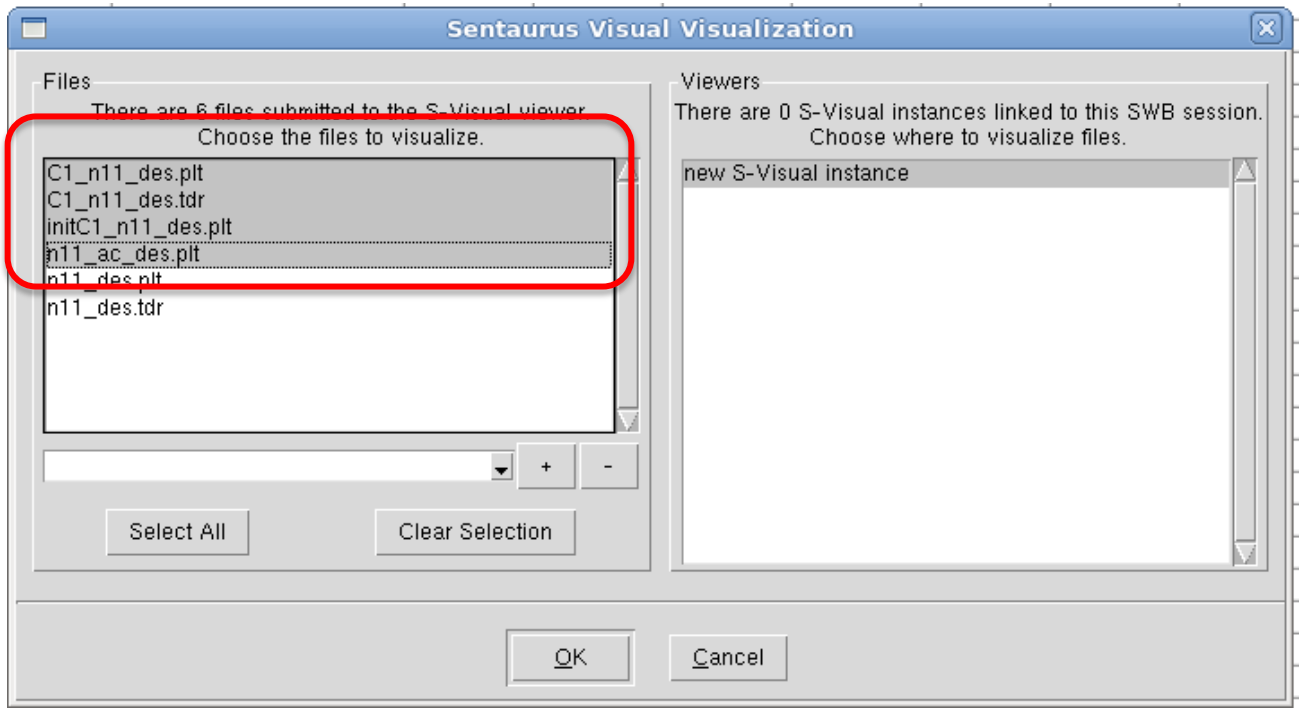
- Run the ac simulation by right-clicking on one of the VGB nodes (e.g. the one corresponding to VGB=-1V) → Run... → OK



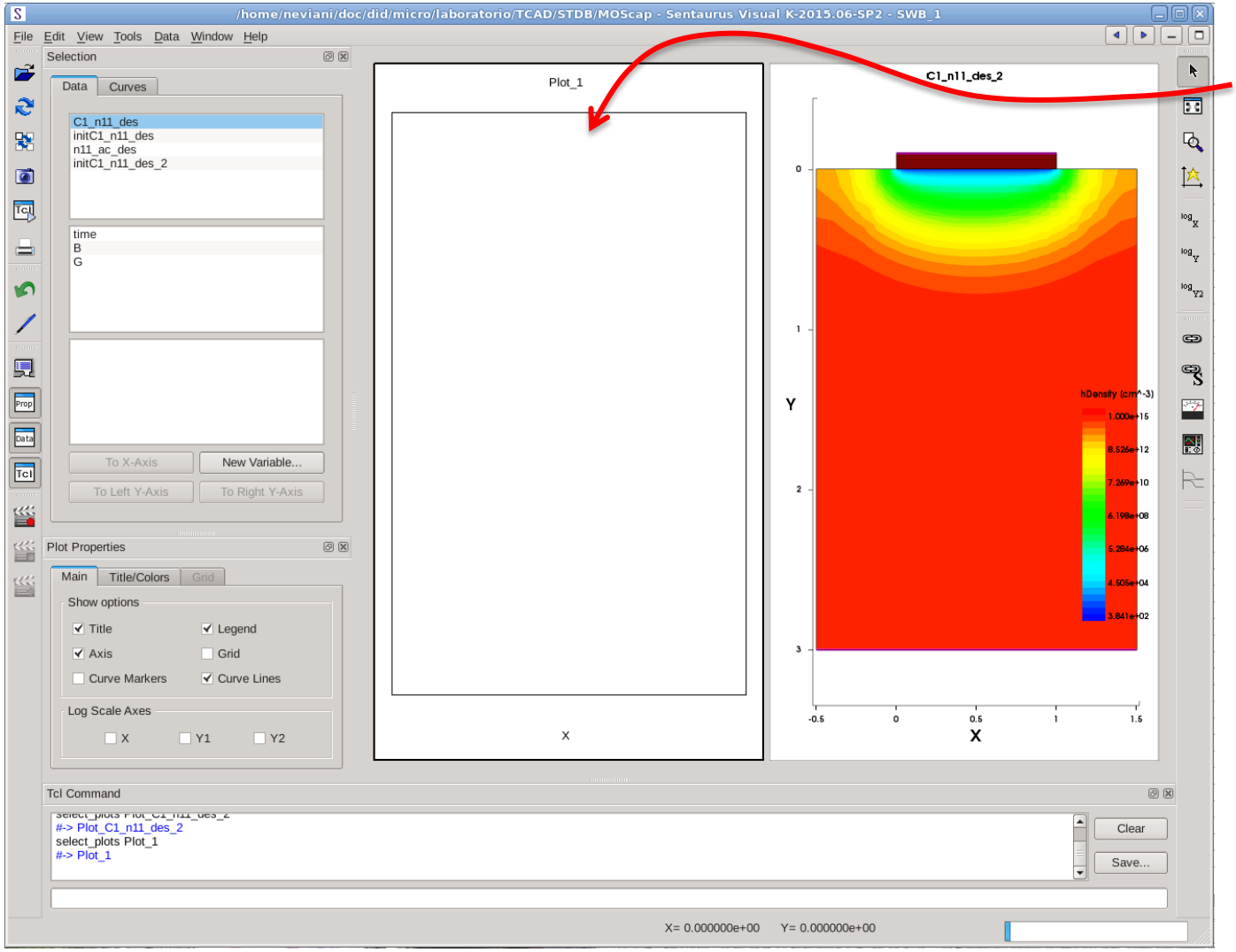
Tutorial #3: ac simulation

- To post-process the simulation results, open SVISUAL by right-clicking on the simulation node → Visualize → Sentaurus Visual (Select File...)

select the output files of the ac simulation and click OK



Tutorial #3: ac simulation



click on the Plot_1 window

the right panel changes to the one shown here

click on n11_ac_des data set (the number may be different from 11)