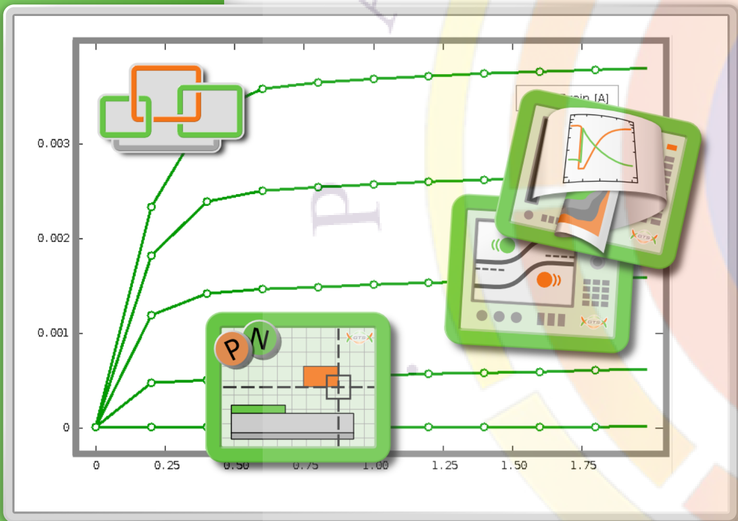
GLOBAL TCAD SOLUTIONS — GTS FRAMEWORK — TUTORIAL

GTS FRAMEWORK



Projects, ToolFolders, Device Simulation

Getting Started I Tutorial

global TCAD solutions®

Projects, ToolFolders, Device Simulation – Getting Started I Tutorial, GTS Framework Release 2017.09
Revision of November 6, 2017

Global TCAD Solutions GmbH
Bösendorferstraße 1/12
1010 Wien
Austria / Europe

Phone: +43 1 9255049
Email: info@globaltcad.com

Additional information is provided at: http://www.globaltcad.com/

Cover & layout: prausedesign

Contents

Introduction

We hope you find this tutorial useful to quickly get familiar with ***GTS Framework***. Besides describing the framework, it explains working with the provided *tutorials* and *examples*, which are in fact a special kind of *projects* that you can open.

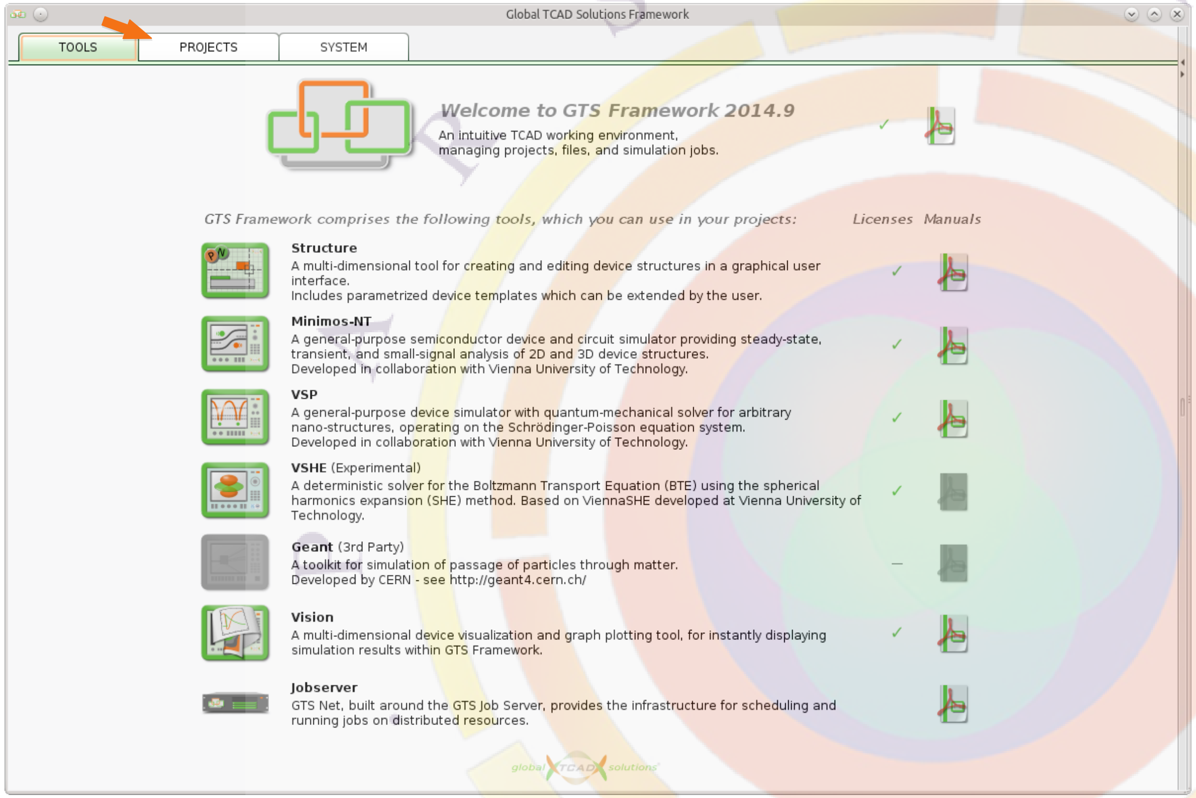
Using GTS Tutorials and Application Examples

GTS ***tutorials*** generally consist of (a) a written description (printed or PDF) and (b) a GTS *project* containing all files necessary to reproduce the described simulations as well as the respective results. (In fact, the project also contains the PDF for reference.)

The written description is intended as a guide while working through the project hands-on in ***GTS Framework***. Having created a working copy of a tutorial project, you can create folders in the project step by step, next to the GTS-supplied tutorial folders – and you can always compare, or even use these folders for subsequent steps.

The next few pages briefly explain this principle, as a prerequisite for starting with the first example in the next chapter.

Besides the tutorials, GTS also provides ***application examples***, which are available as projects in ***GTS Framework***, too – however, being brief descriptions, they mainly focus on the results rather than the step-by-step approach of the tutorials.



GTS Projects

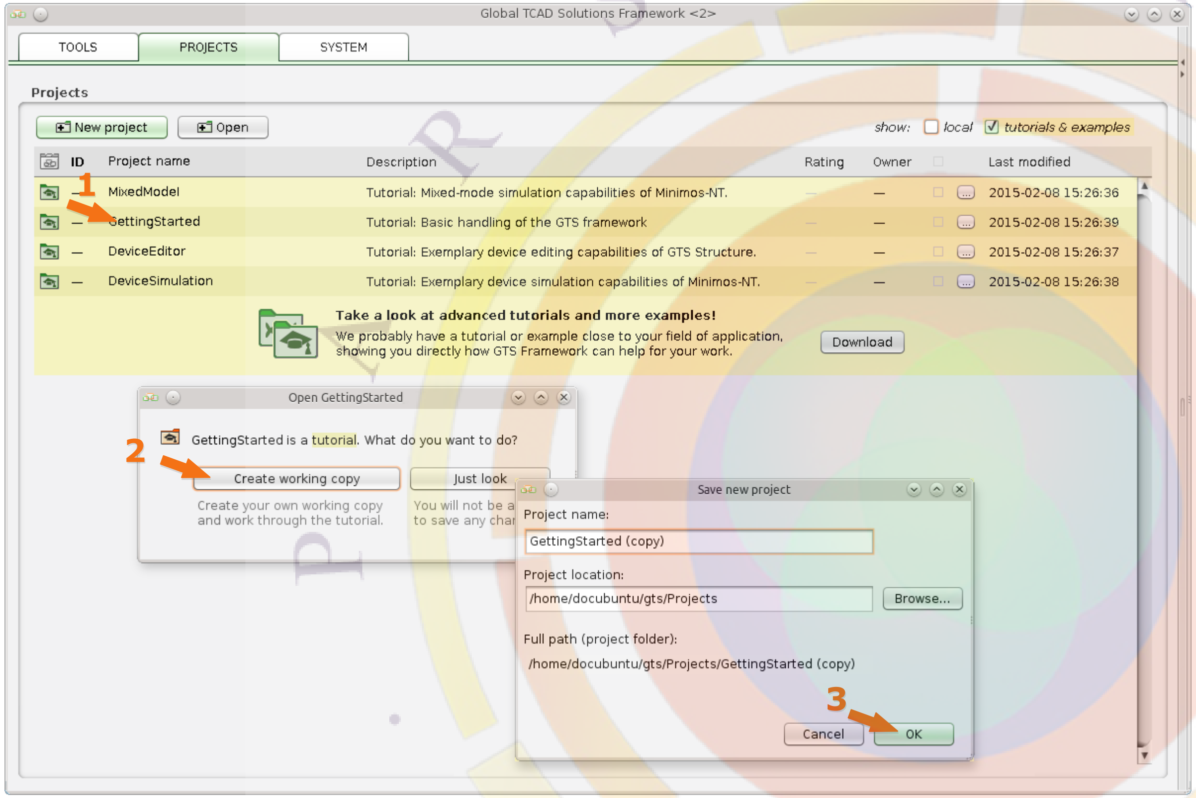
0.0.1. The Framework Home

After starting ***GTS Framework***, you should see the *Framework Home* with the *TOOLS* page. It provides an overview of the available tools with short descriptions, installed licenses and the respective manuals in the rightmost column. You can open the manuals by clicking on the PDF icons. The PDF icon at the top opens the ***GTS Framework*** manual, which provides more in-depth information than this tutorial.

You can start all included tools right here, but for a more efficient workflow we highly recommend using projects, as described in the following steps.

To start working with projects:

- Click the "PROJECTS" tab (or the ***GTS Framework*** icon) to switch to the "PROJECTS" *page*.



0.0.2. Opening (Tutorial) Projects

The *PROJECTS* page shows your own projects as well as GTS-supplied tutorial projects (highlighted in yellow). You can show or hide them using the filter checkboxes at the top-right corner.

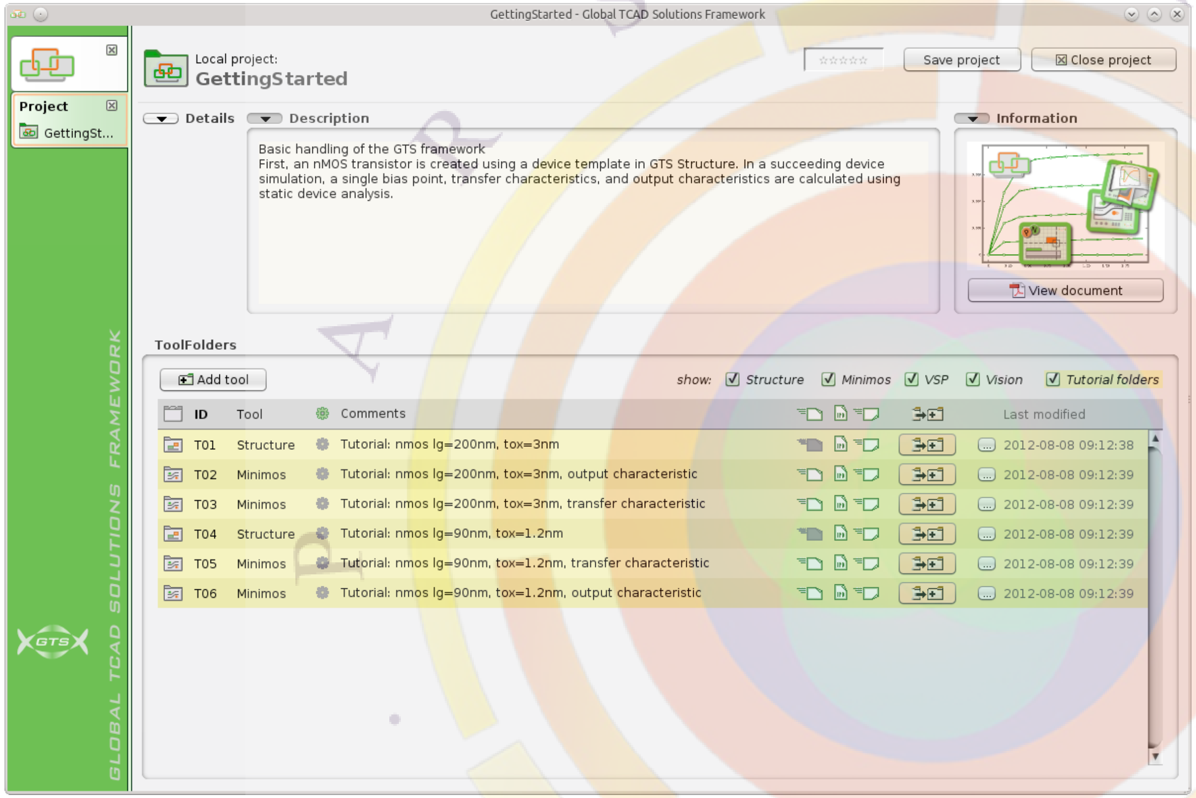
You can click on any project to open it. In case of tutorials, there are two options:

- “Just look” is useful to quickly explore the tutorial (read-only); one can create a working copy later on.
- “Create working copy” creates a new project. So one can experiment with the contained data while leaving the original intact.

To follow this tutorial, create a working copy of the "Getting Started" project:

1. Click on the tutorial “GettingStarted_I”
2. Choose “Create working copy” and
3. Confirm the name by pressing “OK”

The project is created and opened, so that you see the *Project Home*.



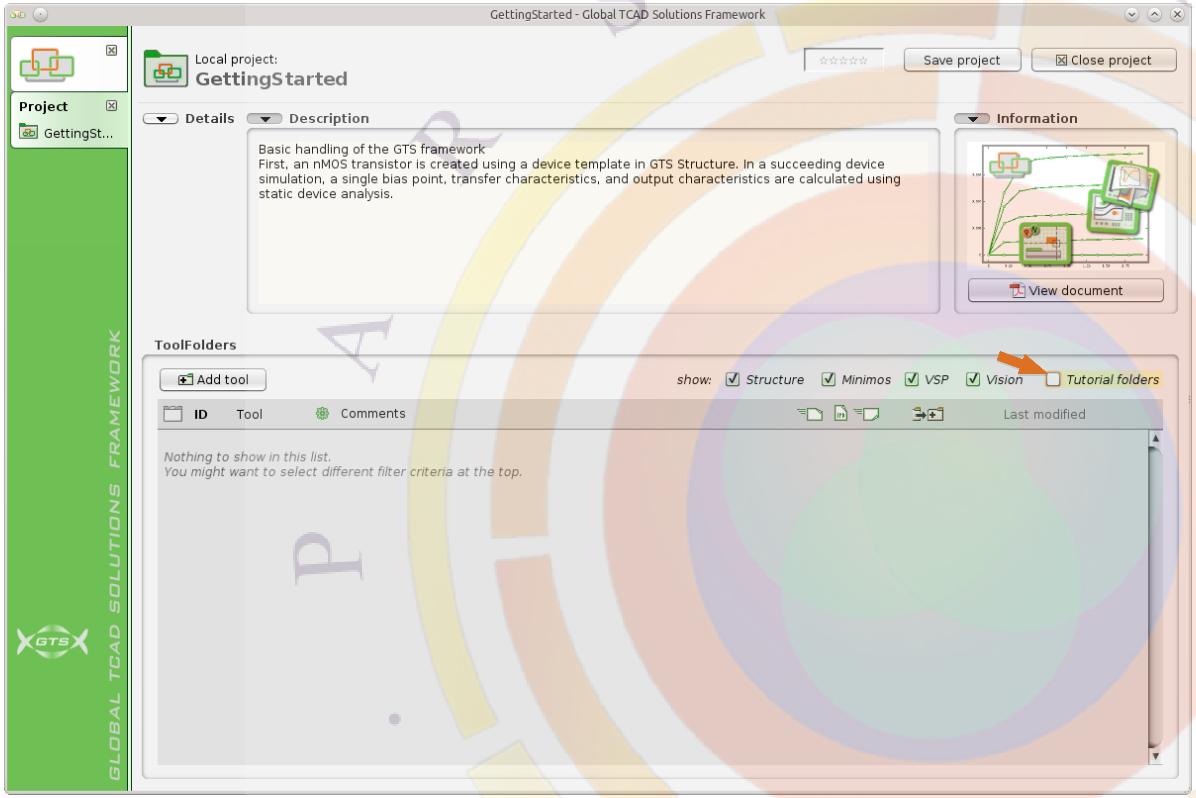
0.0.3. *The Project Home, ToolFolders*

The *Project Home* shows some general data in its upper part, including the *Description* field which you can use for taking notes about the project. At the right, you can open the PDF (this document).

Each project contains *ToolFolders*, storing the data for each step in your workflow. Each ToolFolder is associated with one tool.

Tutorial projects have *Tutorial ToolFolders* (***T..***, yellow), which can be used for reference as well as for subsequent steps.

This project contains six ToolFolders:
The GTS Structure ToolFolders “001” and “004” contain device structures of 200nm and 90nm NMOS FETs.
The Minimos ToolFolders “002” and “003” are simulation folders belonging to the 200nm NMOS, whereas “005” and “006” belong to the 90nm NMOS, respectively.



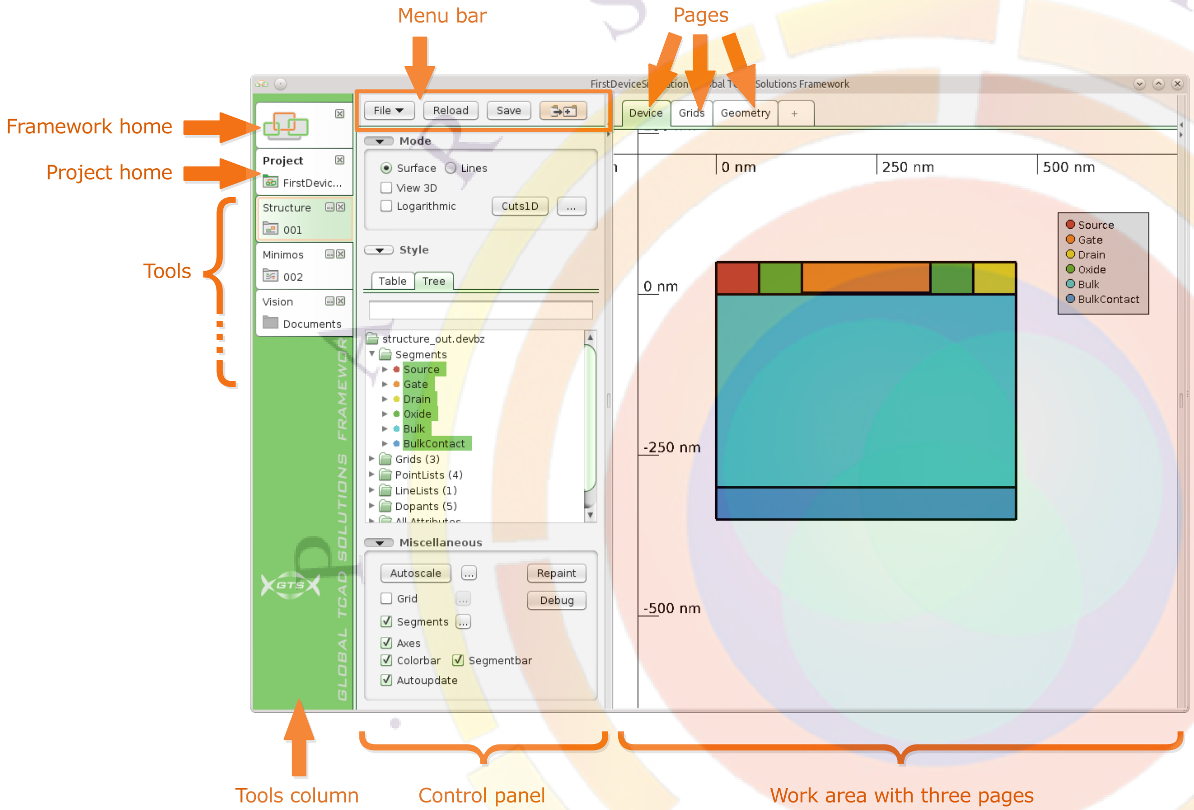
0.0.4. The Project Home II

Later on, the orange *NextTool* buttons in each line will take an important part in your workflow.

Some last tips regarding the *Project Home*:

- You can sort the list by clicking on the headers (click again to reverse order).
- On each item, use the [...] button or right-click for further actions.
- The white file icons indicate whether a ToolFolder contains *input*, *setup*, and *result* files.
- To tidy up, you can show/hide the tutorial folders using the filter check box indicated by the orange arrow.

Remember, while working with the project, you can always come back to the *Project Home* and open any of the yellow tutorial ToolFolders, to compare results or use the contained data for subsequent steps.



0.0.5. Tool User Interface

Finally, a few words on the user interface shared by all GTS tools:

The green *Tools* bar at the left provides access to:

- The *Framework Home* at the top
- The *Project Home* directly below
- Opened ToolFolders further down

All other parts of the window belong to the current tool:

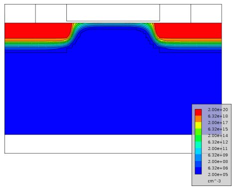
- The *menu bar* providing framework-related functions
- The *Control panel* at the left
- Several *Pages* at the right

For more, refer to the **GTS Framework** Manual (PDF: *Framework Home / TOOLS*).

Now being familiar with the GTS UI concept, we hope you will enjoy the first simulation in the following example!

Part 1

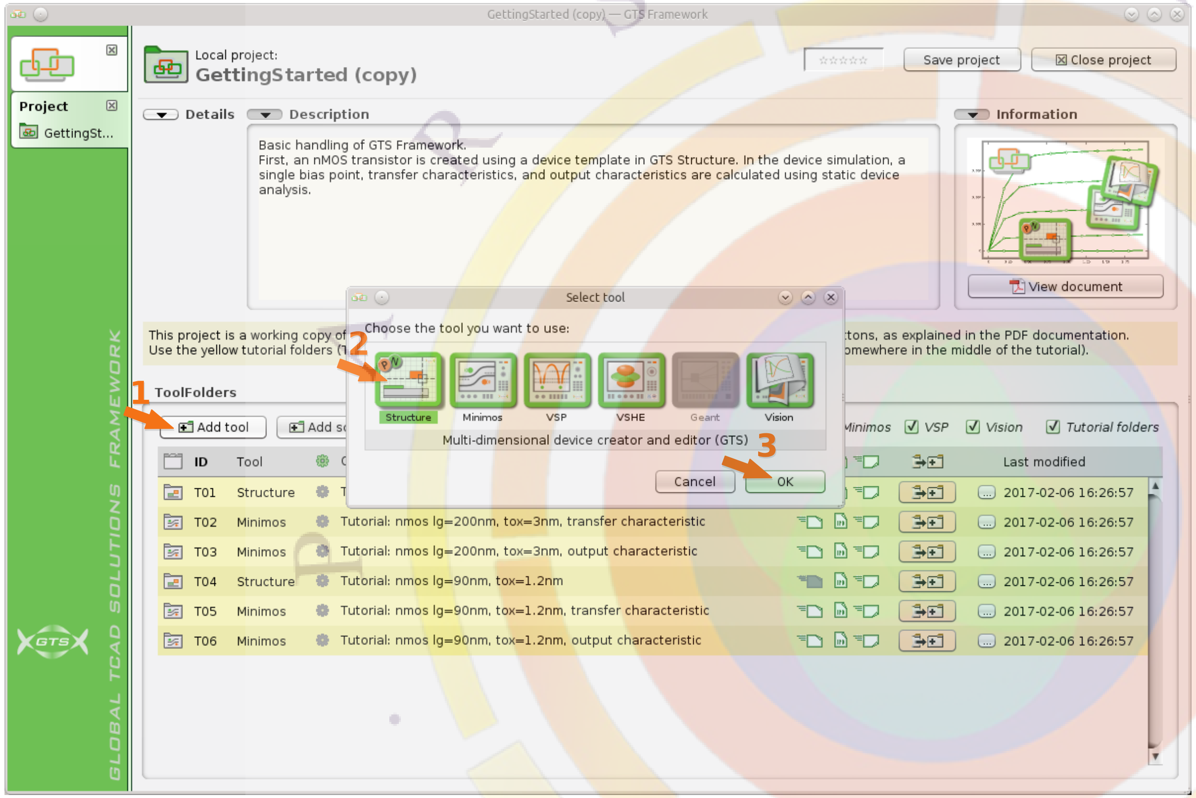
NMOS Transistor



By example of an NMOS FET with 200 nm gate length, the basic handling of ***GTS Framework*** will be shown. The transistor is created using a device template in ***GTS Structure***. In a succeeding device simulation with ***Minimos-NT***, a single bias point as well as transfer and output characteristics are calculated using static device analysis.

Prerequisites

It is assumed that you have worked through the previous introductory chapter, and that you already have created a working copy of the *GettingStarted_I* project, as described therein. If you have created the working copy at an earlier time, just re-open it (or create a new one).



1.1. New Structure

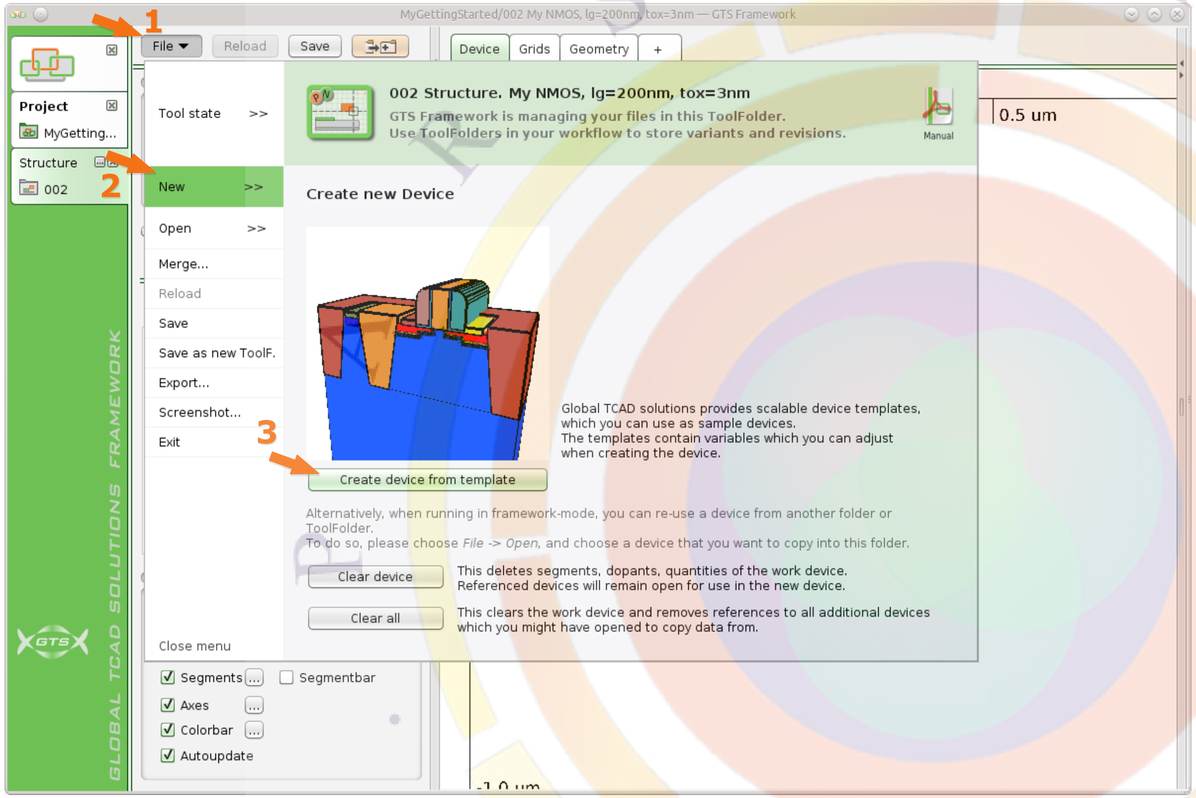
Assuming you have your *Getting Started* working copy open already, you should be seeing the *Project Home*. (If not, please refer to the introduction above.)

First, we add a ***GTS Structure*** ToolFolder as a place for creating and saving the device:

1. Press “Add tool”,
2. Select “Structure” to add a GTS Structure ToolFolder
3. Confirm with “OK”

You will now see the new ***GTS Structure*** ToolFolder, and the File menu will open automatically, as shown on the next figure.

According to the default settings, the ToolFolders can be found in the folder “`gts/project name/ToolFolder`” in the home directory.



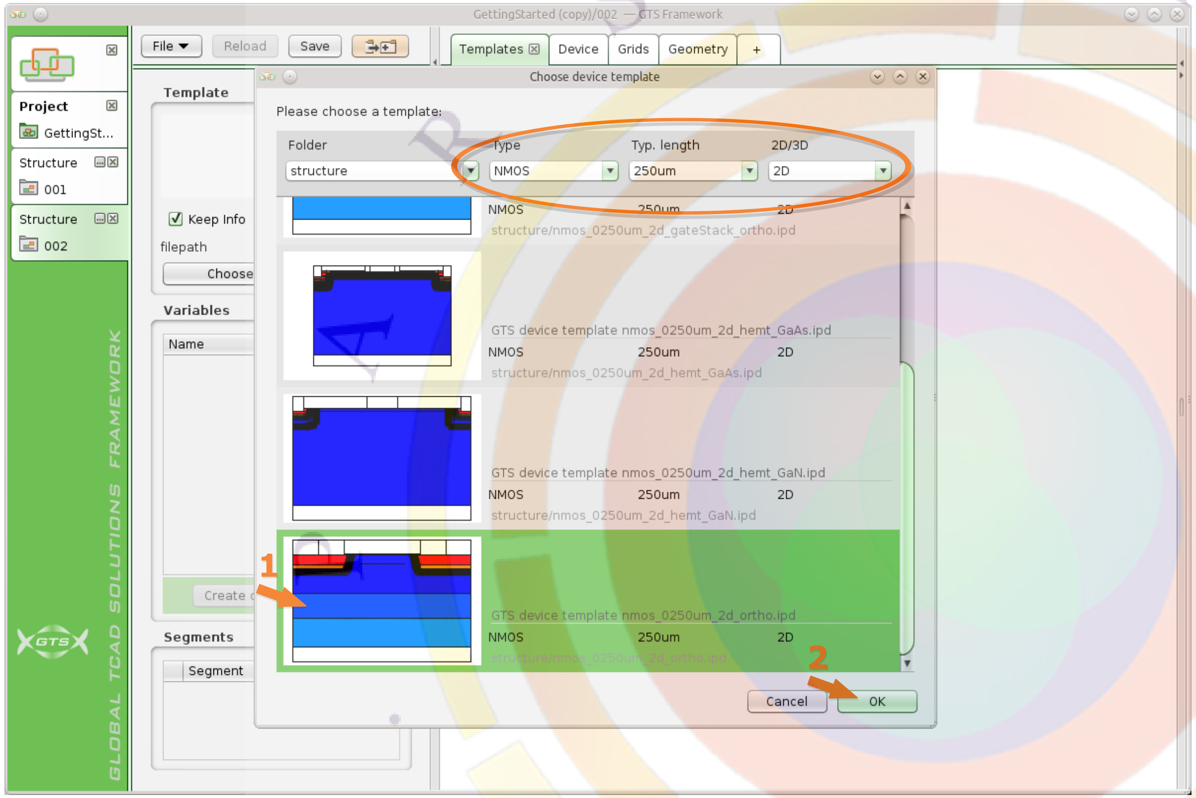
1.1.1. Creating a New Device

- If you prefer, you can add some notes on the device in the light-green header of the file menu that just opened. You can also do this any time later.

GTS Structure includes parameterized templates for common devices.

We will now create a device based on such a template:

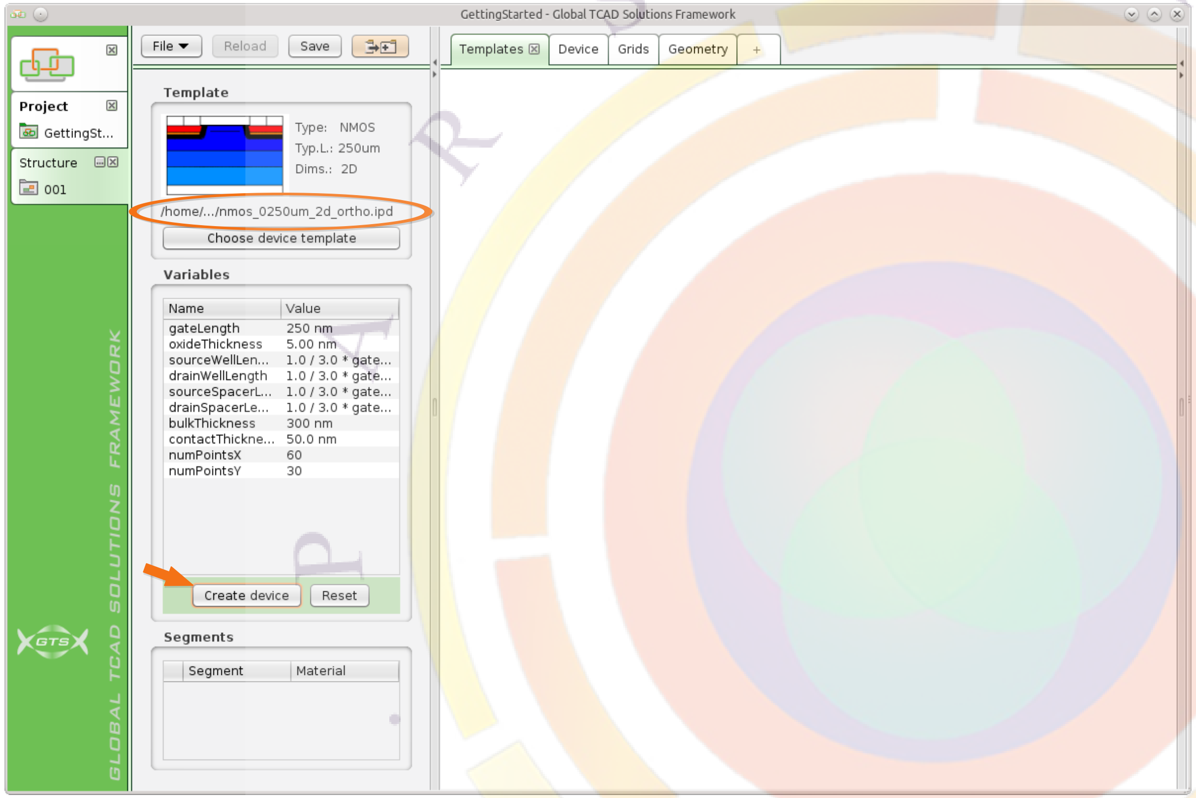
1. If the *File* menu is not already open, press “File” in the menu bar
2. Choose “New”
3. Press “Create device from template” to open the list of available templates.



1.1.2. Choosing a Template

There are templates for various device types and technologies; the filters in the list header help to quickly find the desired template.

1. Set the filters to: *NMOS, 250um, 2D.*
2. Select the template with the name `nmos_0250um_2d_ortho.ipd`
3. Confirm with "OK"



1.1.3. Creating Device I

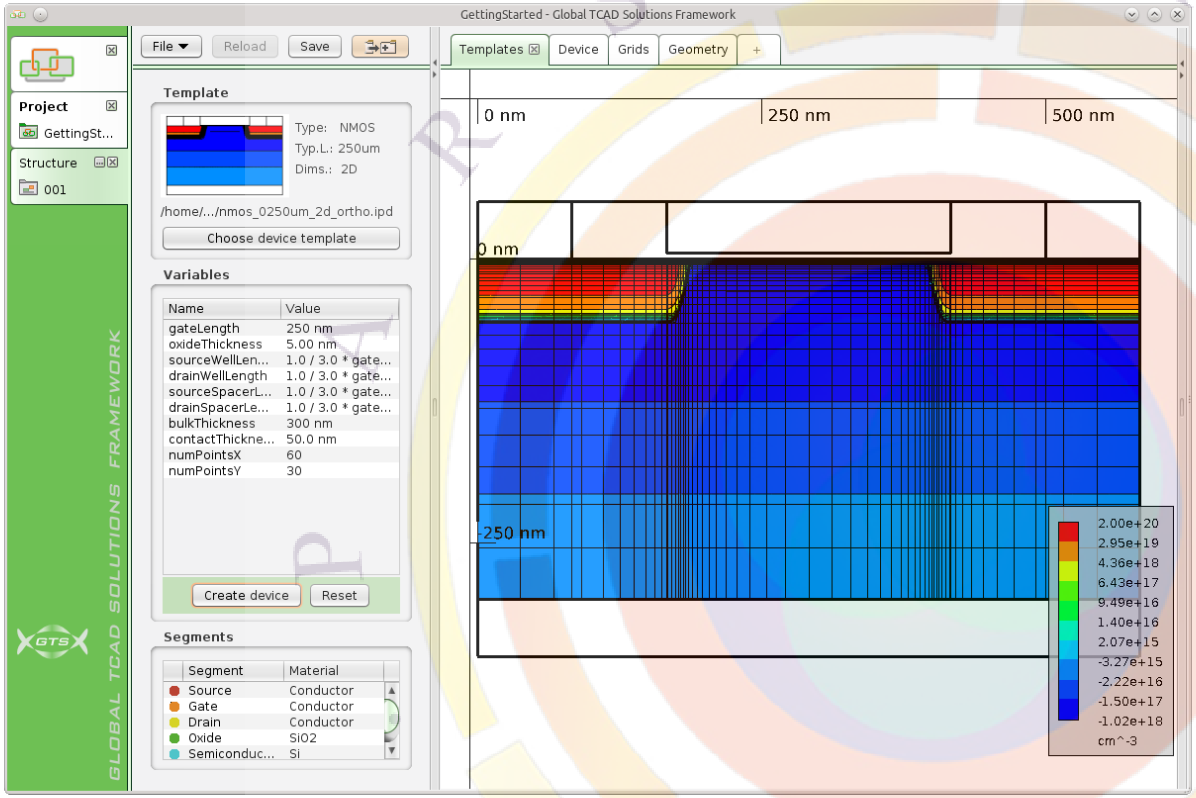
Now, the control panel at the left shows a preview and the name of the chosen template in the top group.

The *Variables* table below lists all customizable parameters of the template (we will use it later on).

To create the device structure with the current parameter values:

1. Click “Create device”

The chosen template file `nmos_0250um_2d_ortho.ipd` can be verified in the highlighted area.



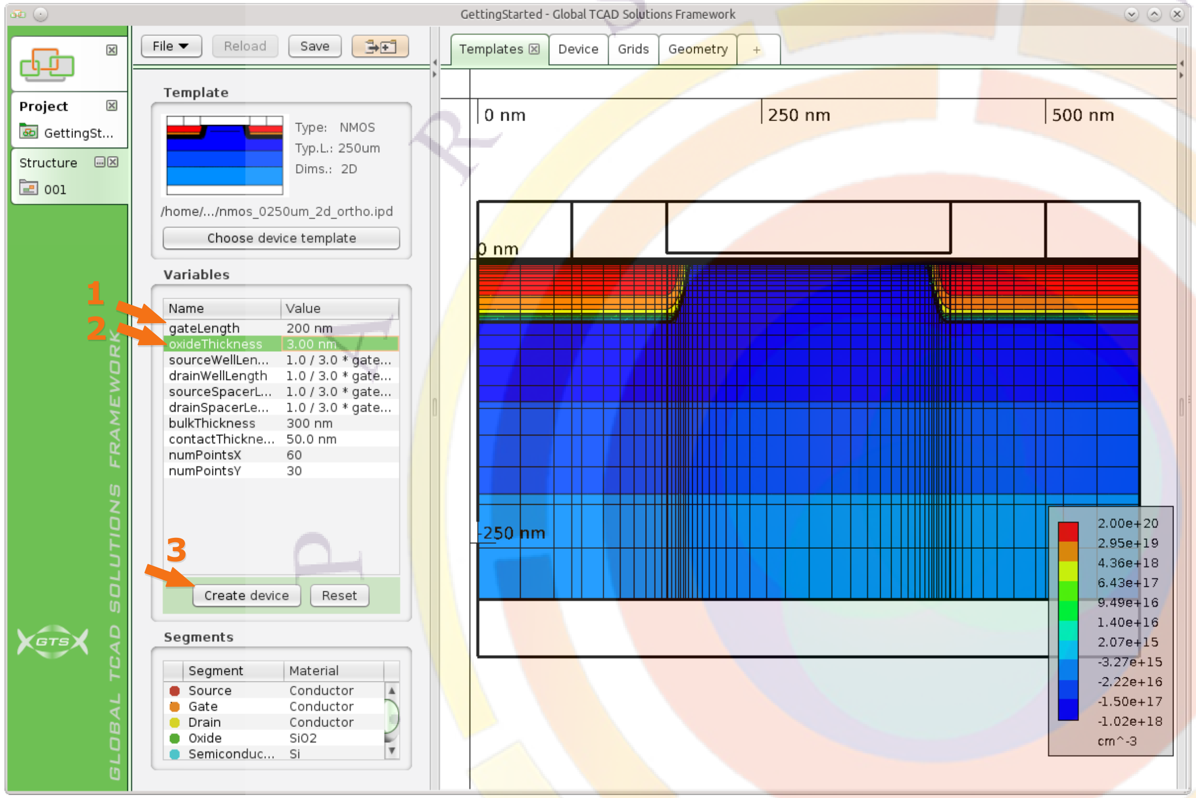
1.1.4. Creating Device II

Now, the device is being created and displayed in the work area on the "Templates" page.

The *Segments* table at the bottom shows the names and types of the segments in the device.

In device views in ***GTS Framework***, you can generally use the mouse as described below:

- Use the mouse wheel to zoom in / out
- Drag with right mouse button to pan
- 3D: Drag with left button to rotate
- Right-clicking opens the context menu (more options).



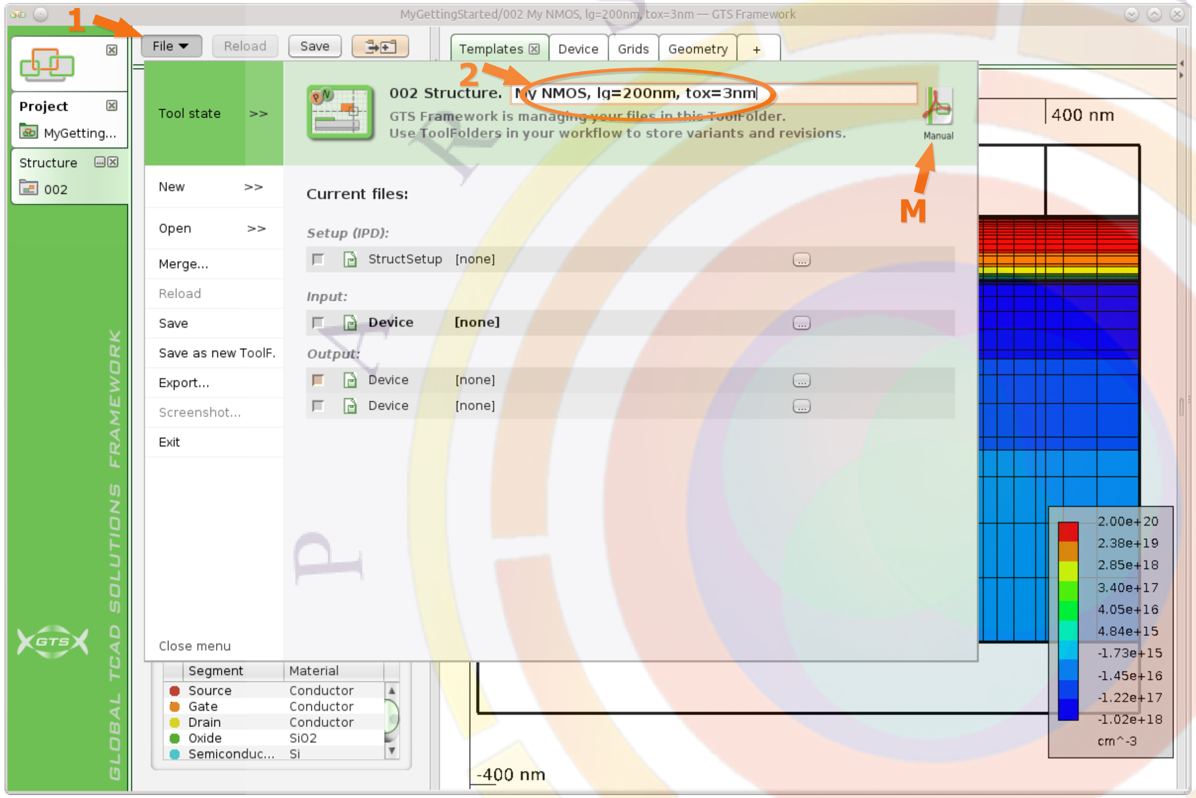
1.1.5. Template Parameters

We want to modify some of the device parameters of our NMOS FET. In the *Variables* table, you can edit all values by double clicking on the respective fields (press enter when done).

For our device:

1. Set gateLength to 200 nm
2. Set oxideThickness to 3 nm
3. Press “Create device” again (this replaces the current device)

> You can press “Reset” to reload the default values from the template.

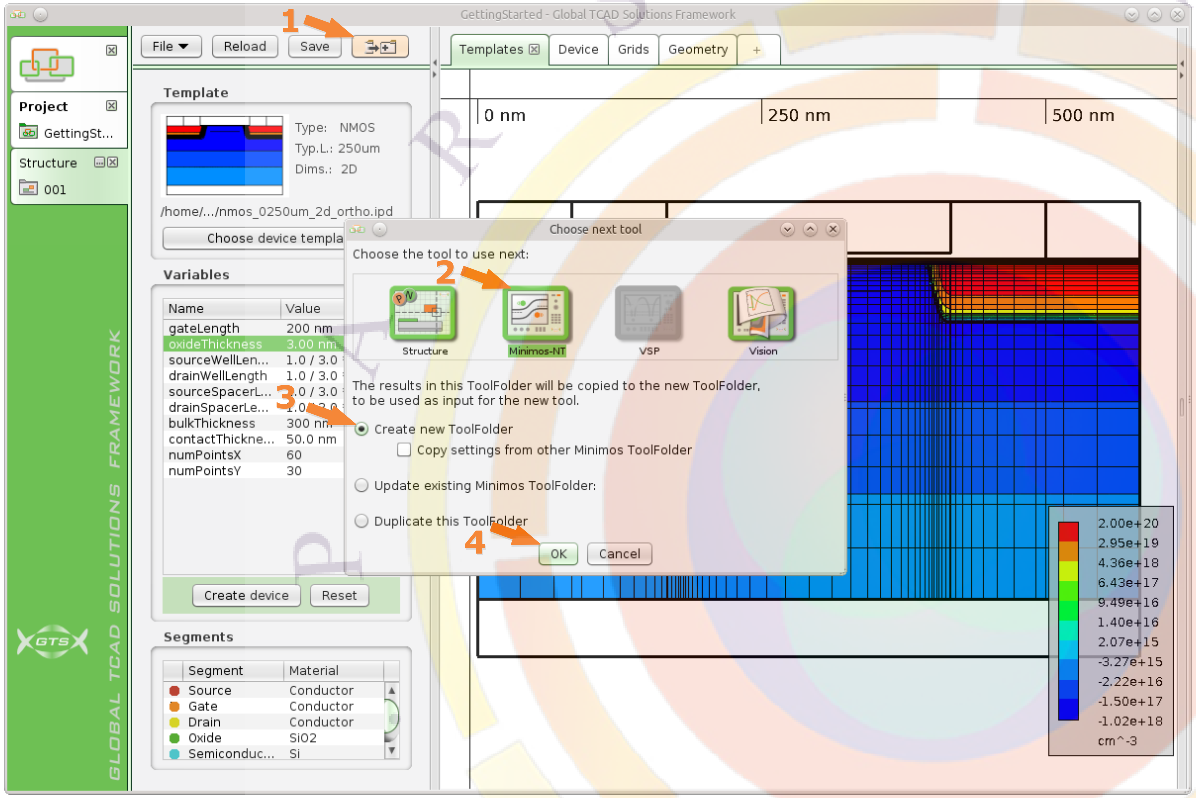


1.1.6. Comments, User Manuals

For your convenience, it is recommended to assign comments to your ToolFolders, if you have not already done so. The comments are shown in the ToolFolders list in the *Project Home* and can be used to sort the list.

1. Press “File” in the menu bar. Note the empty space in the light-green header of the menu
2. Click and write your comment in the designated area in the figure, press Enter when done

For quick reference, clicking the PDF icon at the top right corner of the *File* menu always opens the user manual of the current tool (M in the figure).



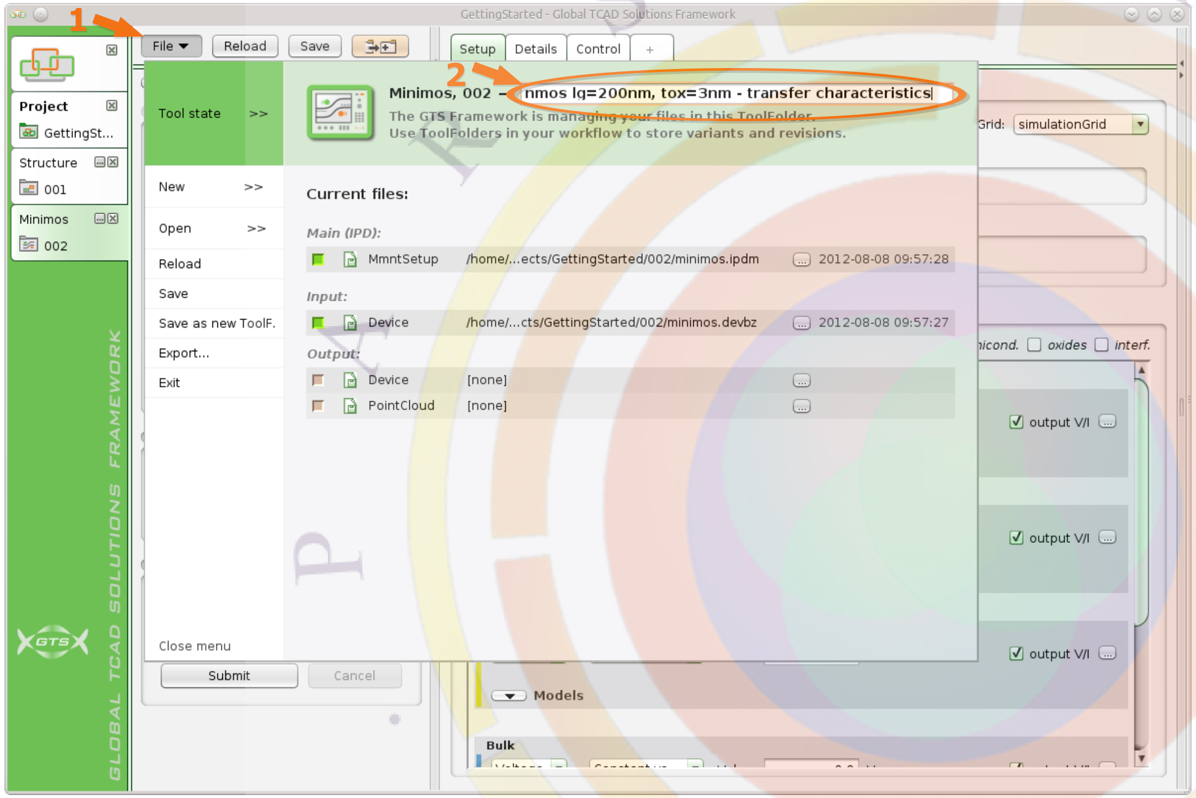
1.2. Device Simulation

1.2.1. New Device Simulation

We want to get the transfer- and the output characteristics of this device. Thus, we will use ***Minimos-NT*** for the next step, adding a ***Minimos-NT*** ToolFolder for storing the simulation data. The "Next tool" function provides a one-button approach for this:

1. Press the "Next tool" button in the menu bar
2. Click "Yes" to confirm saving
3. In the following Next Tool dialog, choose "Minimos"
4. Select "Create new ToolFolder"
5. Confirm with "OK"

Use the "Next tool" button whenever you take a new step in your workflow, either because you use another tool or because you want to preserve your state of work before making changes. If ***GTS Framework*** requests to save the current data, confirm with "Yes".

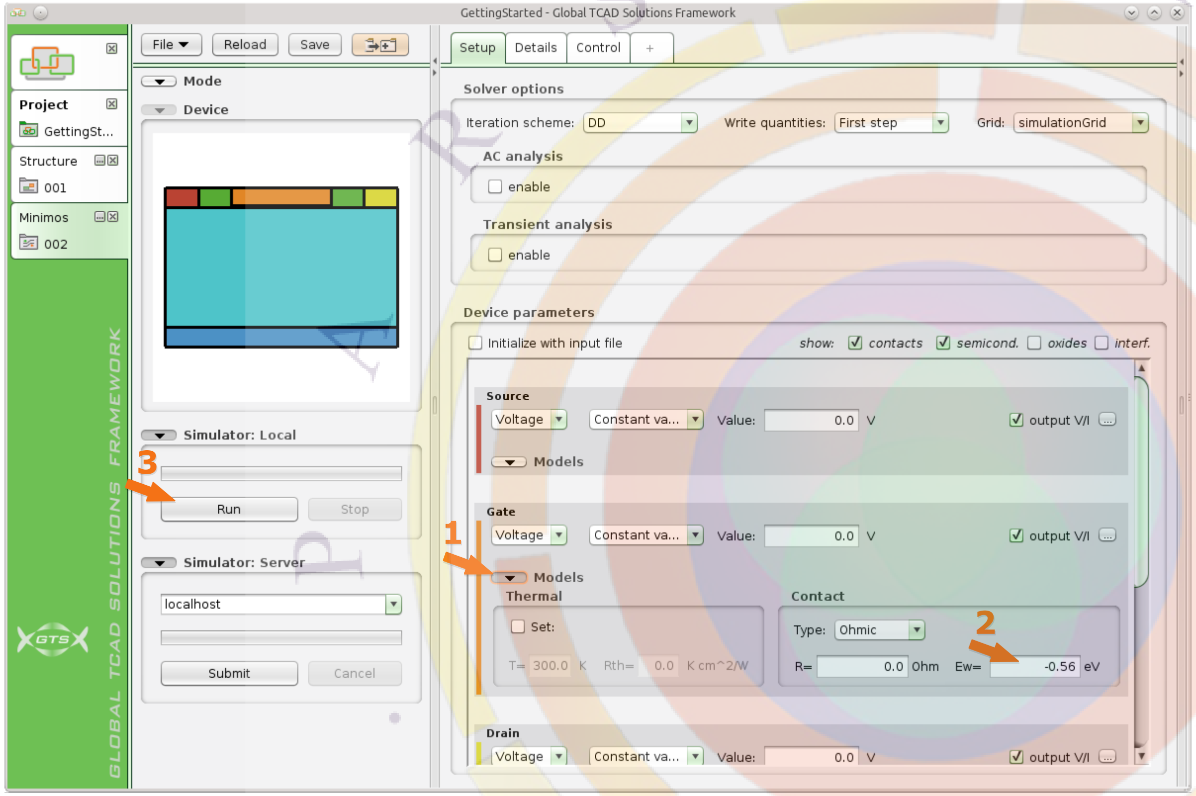


1.2.2. Editing Comments

The newly created ***Minimos-NT*** ToolFolder opens automatically. It automatically got a comment based on our previous ToolFolder, but we like to edit the comment according to our project:

1. Press “File” in the menu bar
2. Click and edit the comment where indicated in the figure to describe the simulation
3. Press enter when done

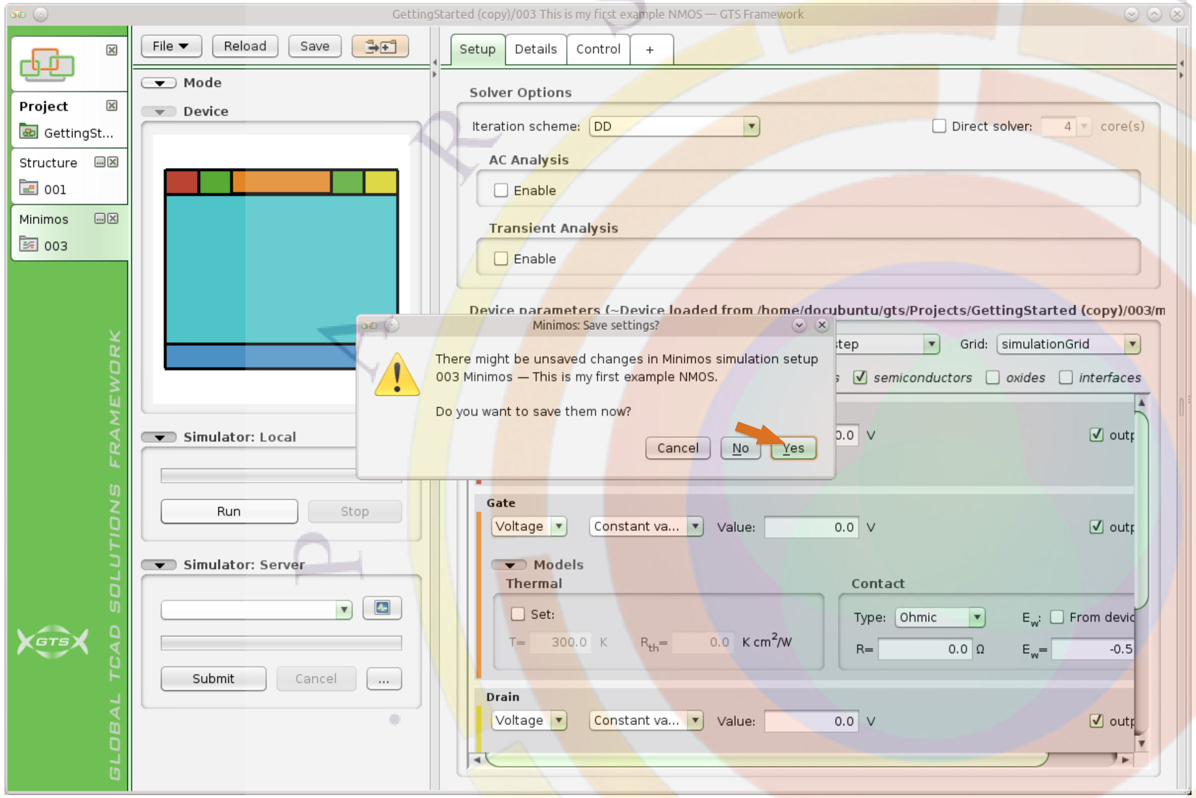
The comments of each ToolFolder can also be edited in the Toolfolders list in the Project Home.



1.2.3. Setting Up the Simulation

The "Setup" Page is preselected, and we can specify our first simple simulation:

1. Locate the *Gate* segment in the *Device Parameters* list, and press the button "Models" to expand the model panel
2. In the contact group, set the contact work function difference E_{w} to a typical value of $-0.56\,\mathrm{eV}$
3. Run the simulation by pressing "Run" in the control panel

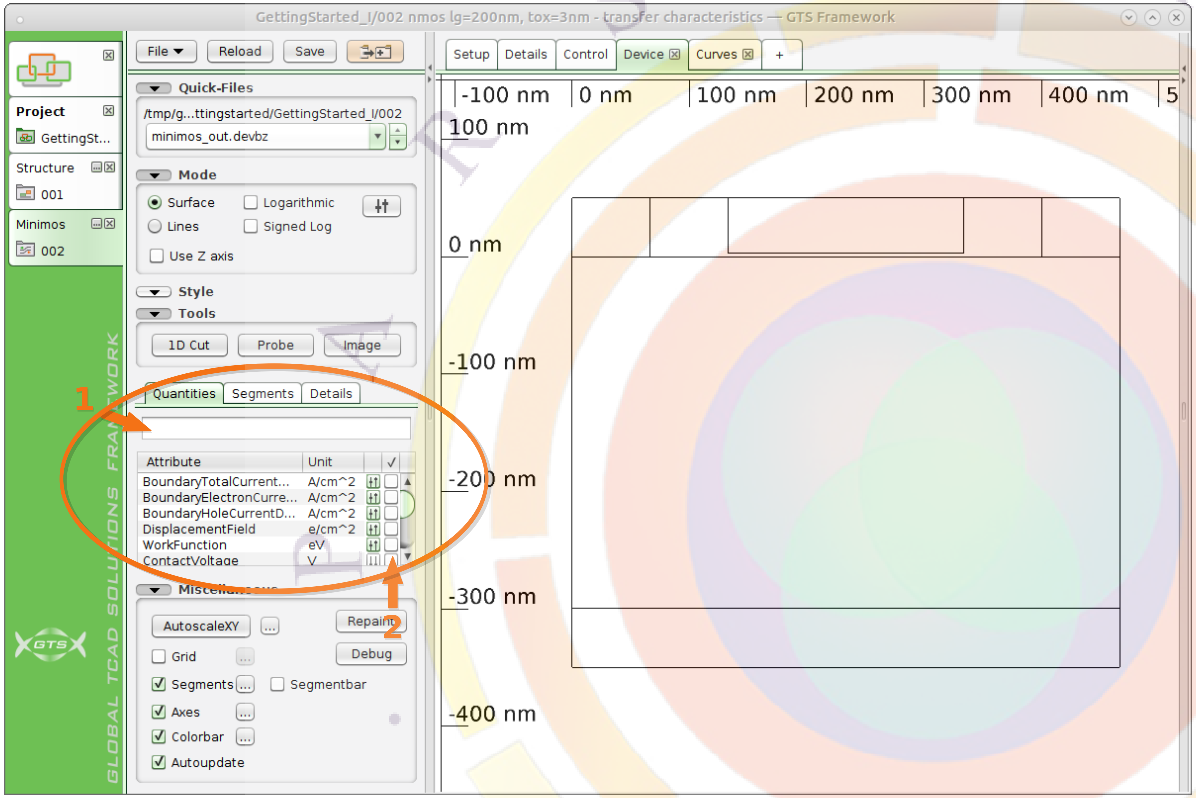


1.2.4. Running the Simulation

GTS Framework requests to save the ToolFolder if it is important for further steps:

1. Confirm the "Save settings?" dialog by pressing "Yes"

Now the simulations starts, and the page "Control" is automatically activated for viewing the simulator log / progress.



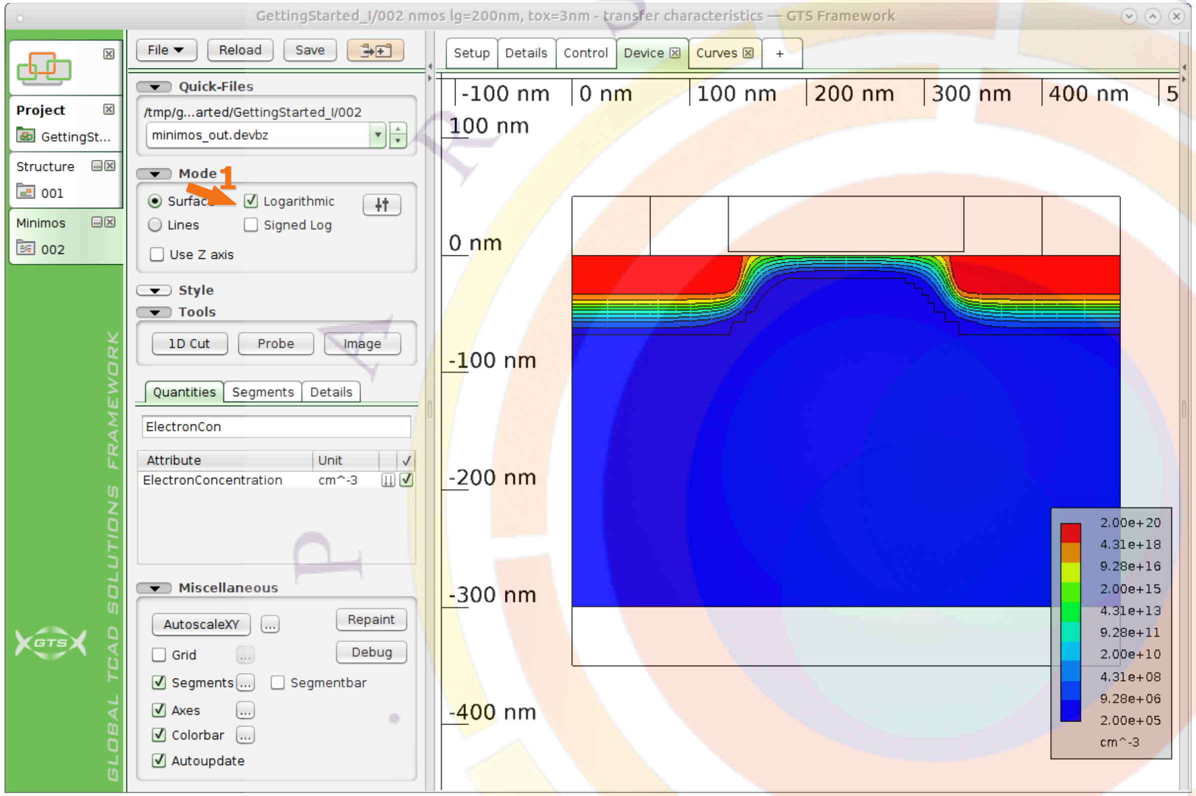
1.2.5. Electron Concentration I

When the simulator has finished, the "Device" page appears. Switch to the "Device" page.

We can now select the quantities to view using the table highlighted in the figure:

1. Filter the table by starting to type the desired quantity name in the text field. We want "ElectronConcentration". Note that the table refreshes while you type to show the matching quantities.
2. One or more quantities can be displayed by selecting the respective checkboxes. Check "ElectronConcentration"

To display one quantity exclusively, you can de-select all other quantities (even those invisible due to the filter) by double-clicking the respective check box.

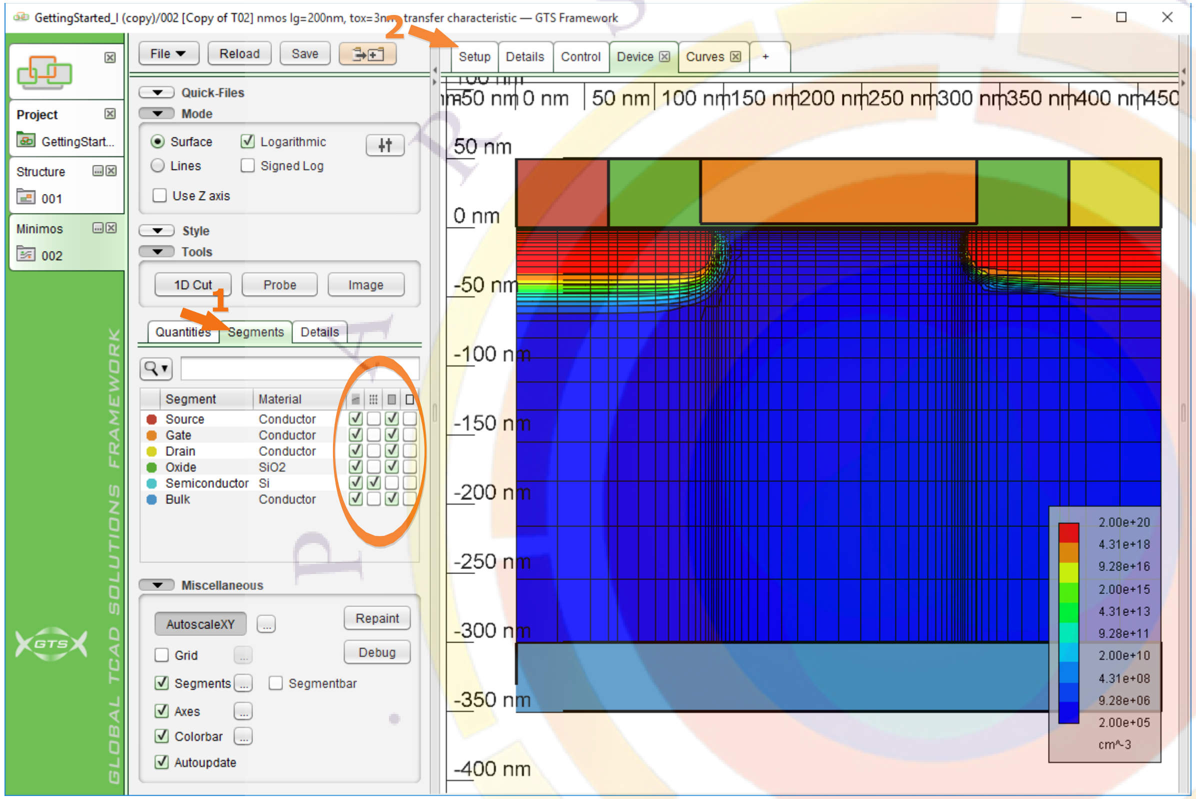


1.2.6. Electron Concentration II

To nicely display the electron concentration:

1. In the *Mode* group, choose the logarithmic view

In the Device page, use the mouse wheel to zoom, or right-drag to pan as described in Section 1.1.4. Right-click for further options.



1.2.7. Electron Concentration III

For more specific display settings:

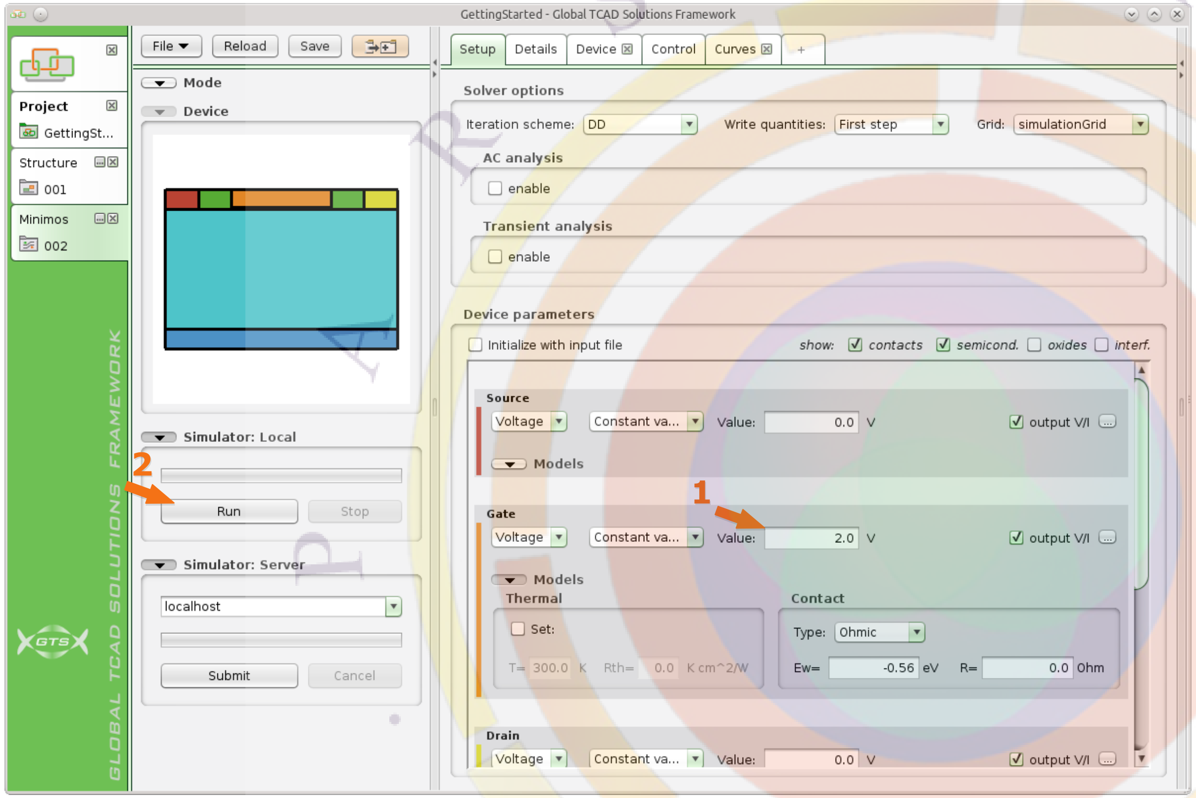
1. Switch to *Segments*.

The four checkboxes control the visible components for each segment:

- Quantity
- Simulation grid
- Filled boundary (volume)
- Wireframe (lines)

You can always re-run a simulation with modified settings (the settings as well as the result files are overwritten, as long as you keep working in the same ToolFolder). This is what we will do now:

2. Switch back to the “Setup” page

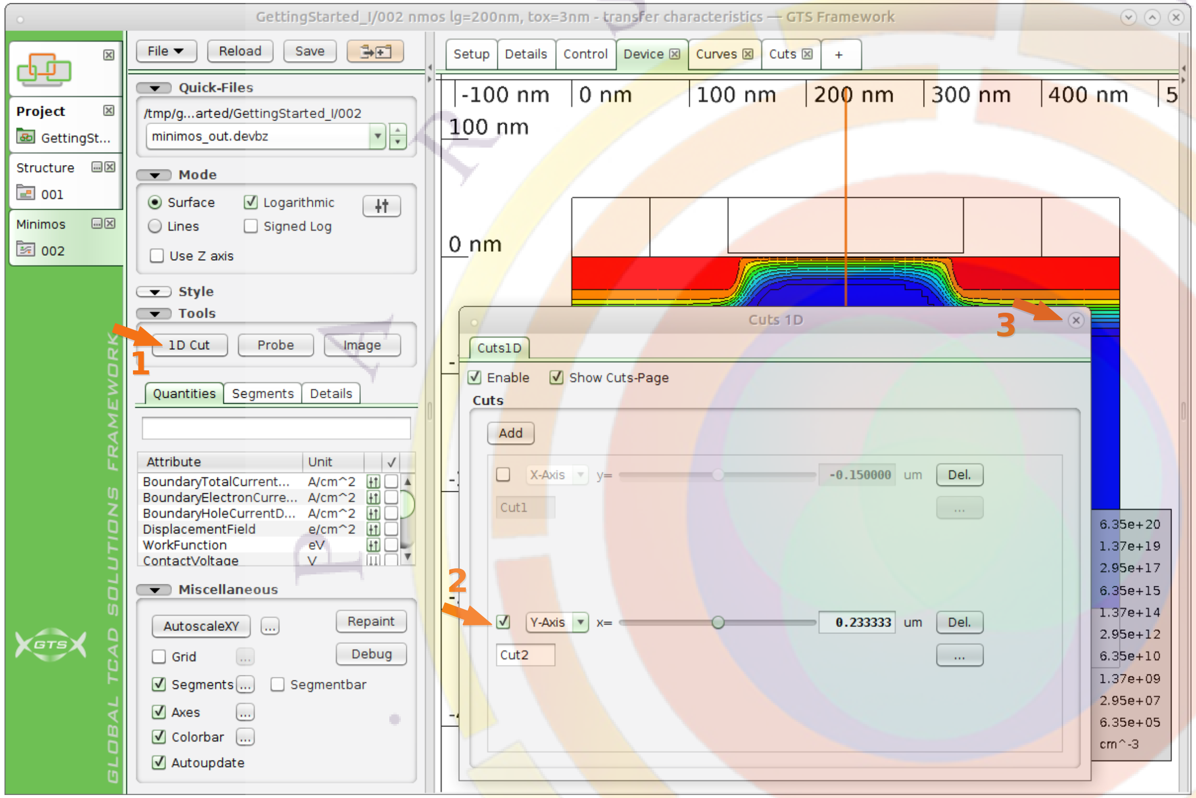


1.2.8. *Setting Gate Bias*

We are now going to set the transistor to on-state and re-run the simulation:

1. Enter a Gate voltage of 2.0 V
2. Press "Run" in the control panel and confirm to save

When the simulation has finished, the "Device" page will be shown automatically.

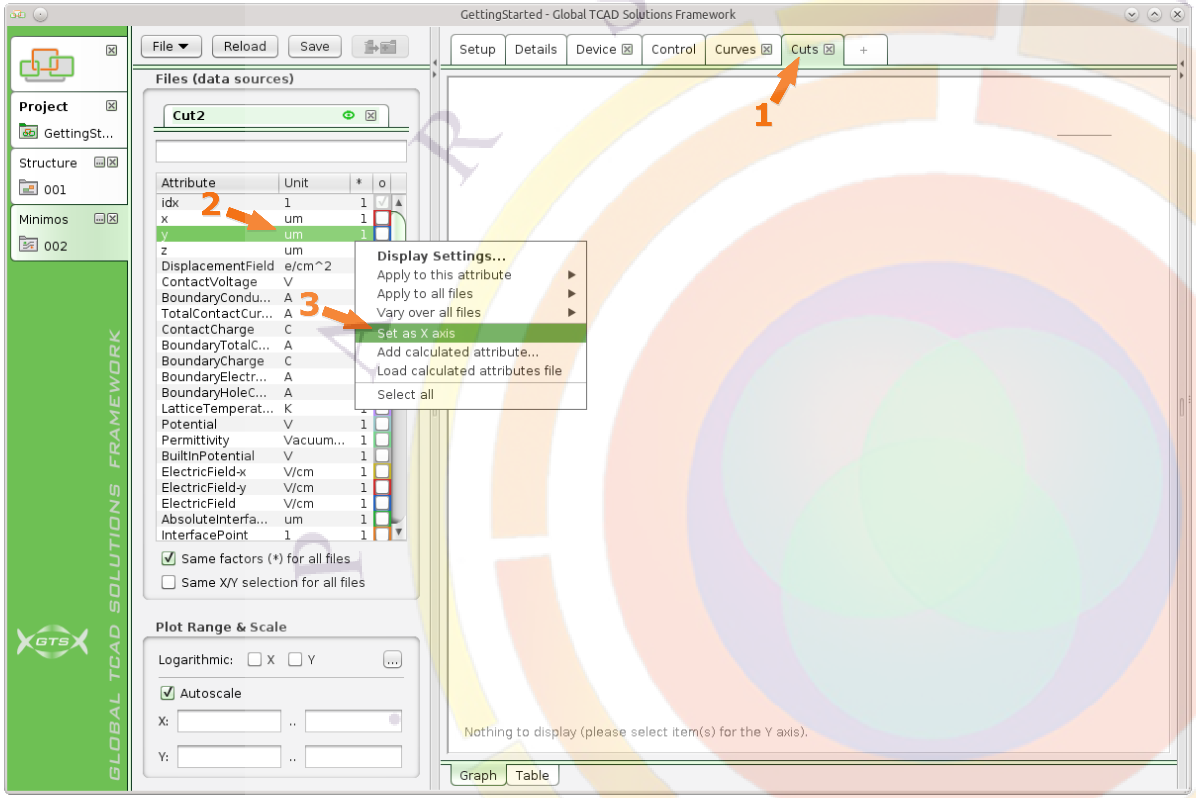


1.2.9. One-Dimensional Cut I

After the simulation is done, the “Device” page shows the result.

To investigate the electron concentration, we want to create a one-dimensional cut.

1. Press “Cuts1D” in the control panel to show the Cuts list.
   Here, you can manage all cuts for the current device
2. The *Cuts* list has two default entries, for X and Y respectively. Enable the one with Y-Axis.
   Note the orange line in the device view, indicating the location of the cut
3. Close the *Cuts* list window.



1.2.10. One-Dimensional Cut II

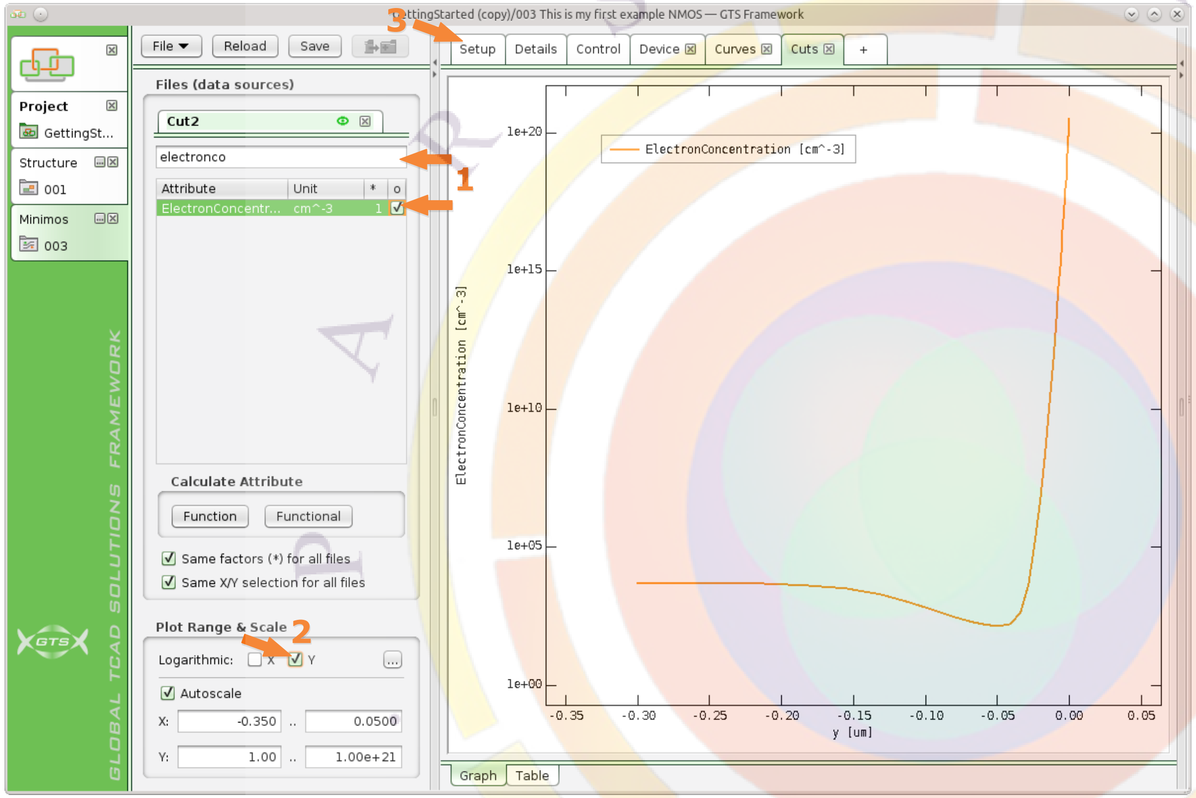
Note the new “Cuts” page in the work area.

1. Switch to the “Cuts” page.

Below the name of the cut, the control panel shows all quantities you can choose to display by selecting the respective check boxes. Similar to the 2D device view, there is a filter text box above the table. Each table row has a context menu, accessible by right-clicking the respective row.

2. Verify that the “y” Attribute has been set as X axis (abscissa) automatically. (In cases where you need to set this manually, you can right-click on the desired quantity and choose “Set as X axis” from the context menu (3).)

To check/uncheck all filtered quantities, click on the table header above the chck boxes.



1.2.11. One-Dimensional Cut III

Now we can select a quantity for the ordinate (Y axis) of our plot:

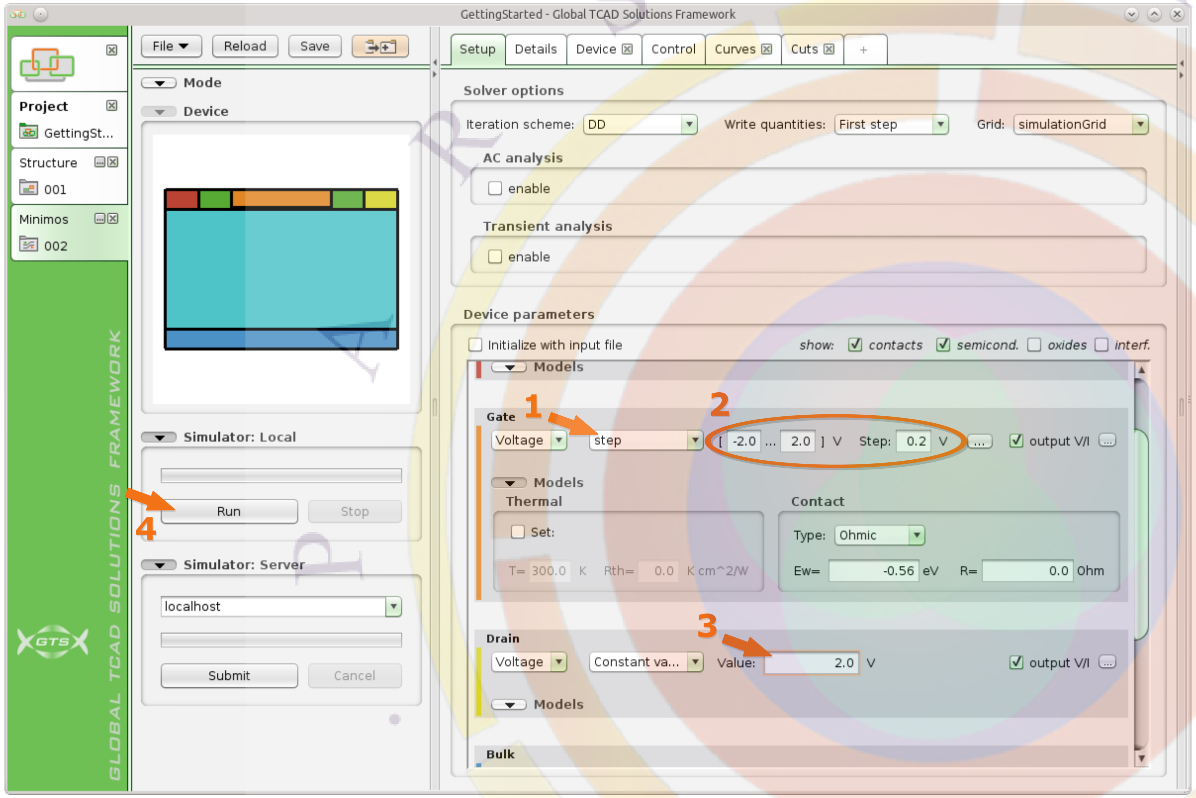
1. Search for “ElectronConcentration” in the attributes list (type it into the filter field) and check the respective box
2. For the Y axis, choose logarithmic scale
3. Right-click on the legend box to choose a nice location (“Left, Top”)

Since no quantum correction model is active, the maximum of the electron concentration is reached directly at the oxide interface.

In 1D plots, you can generally use the mouse as described below:

- Use the scroll wheel to zoom in / out
- Left-click to show coordinates
- Drag with left mouse button to zoom in
- Drag with right mouse button to pan
- Right-click for the context menu

3. To set up further simulations switch back to the “Setup” page

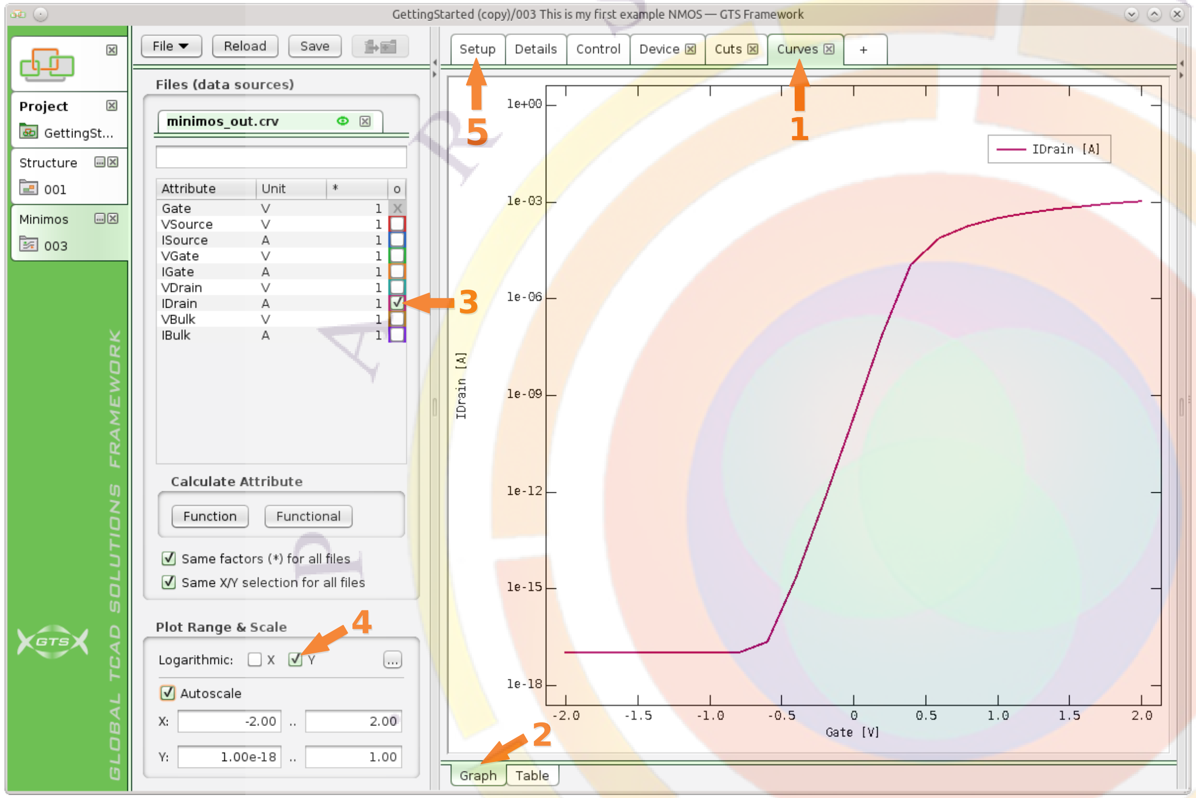


1.3. Transfer Characteristics

1.3.1. Simulation Setup

Finally we want to simulate the transfer characteristics of our 200nm NMOS:

1. In the segments list, locate the *Gate* panel, and select "step" instead of "Constant value" for the voltage (V_g)
2. Define the stepping range from $-2.0\,\mathrm{V}$ to $+2.0\,\mathrm{V}$ with a step size of $0.2\,\mathrm{V}$.
3. Locate the *Drain* contact and set its voltage to $2.0\,\mathrm{V}$
4. Run the simulation



1.3.2. Plot Transfer Characteristics

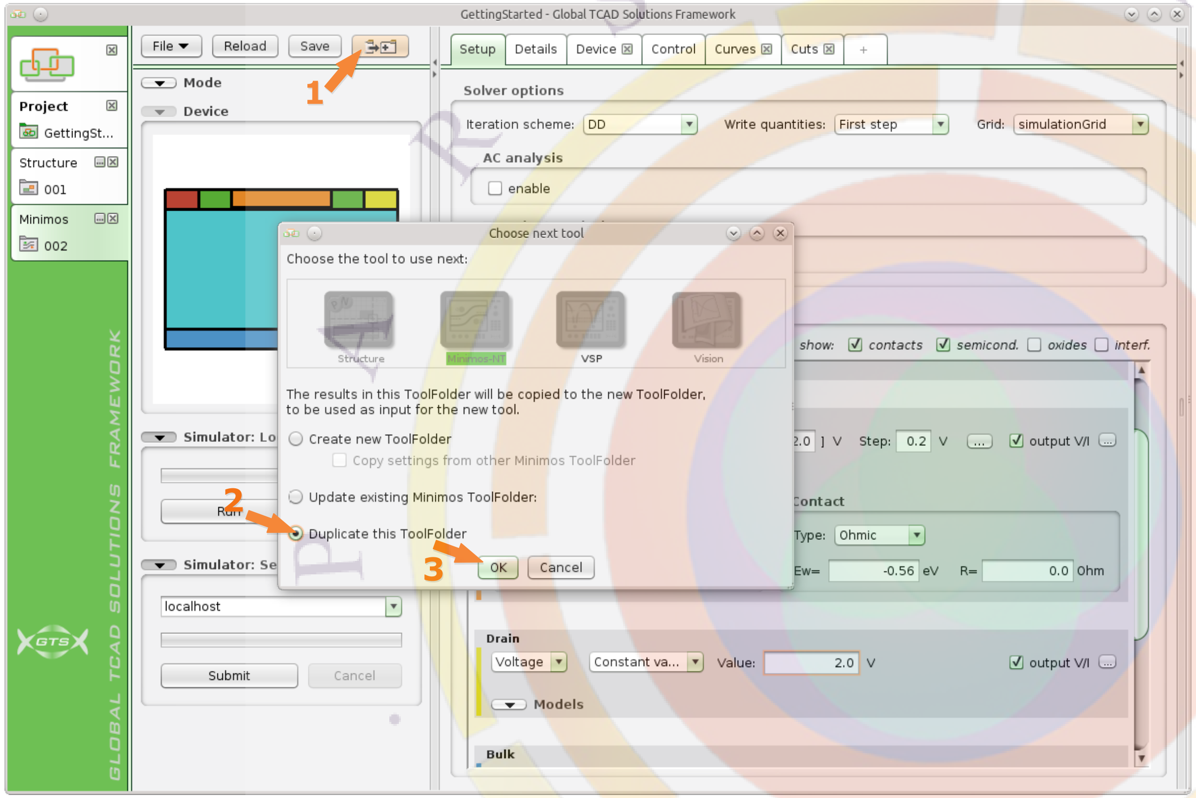
When the simulation is done we want to plot the transfer characteristics:

1. Switch to the "Curves" page
2. If the "Table" view is shown, switch to the "Graph" view using the bottom tabs
3. Select the drain current *IDrain*
4. Choose a logarithmic scale for the Y axis

The plot shows the transfer characteristics. You can right-click the legend to place it at a more convenient location.

In the next step, we will continue with the output characteristics. In order to preserve the transfer characteristics we have just plotted, we will create a new ToolFolder for the next simulation:

5. To enable the "Next tool" button, switch back to the "Setup" page



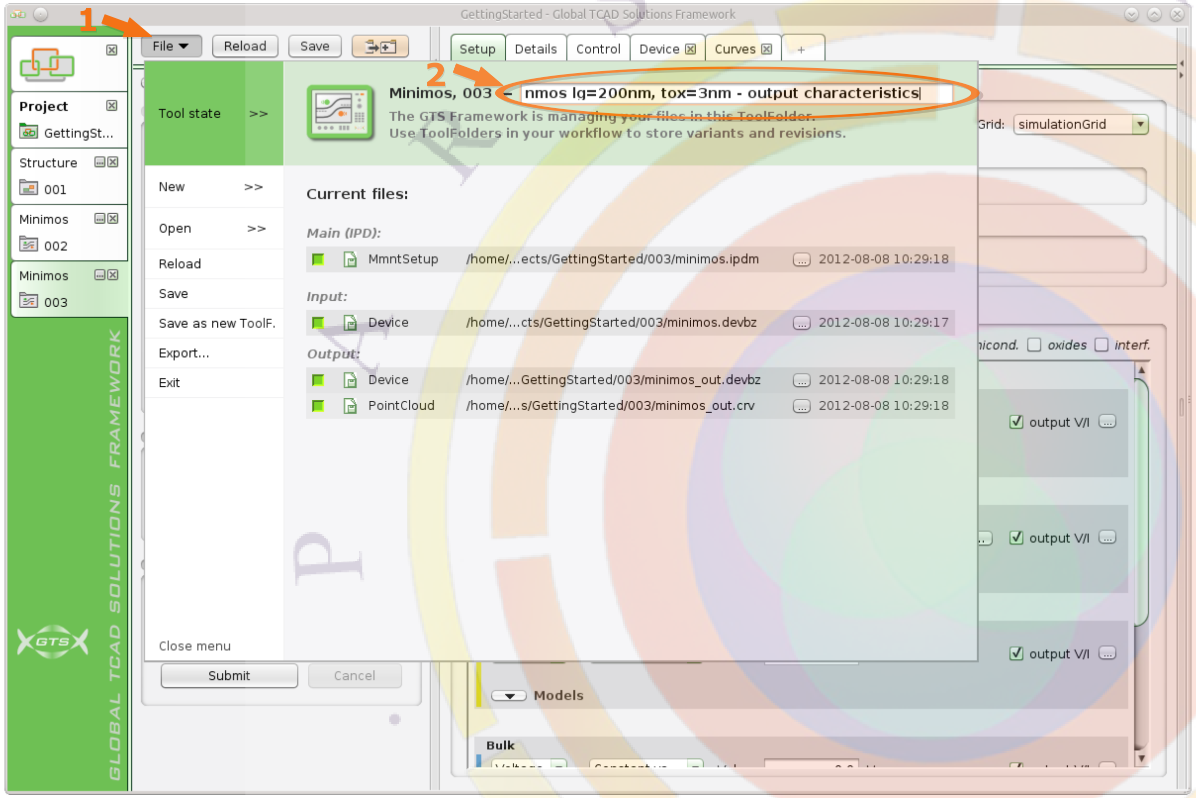
1.4. Output Characteristics

1.4.1. Duplicate Toolfolder

We want to copy the current simulation to use it as a template for the simulation of the output characteristics:

1. Press the “Next tool” button
2. Select “Duplicate this ToolFolder”
3. Confirm with “OK”

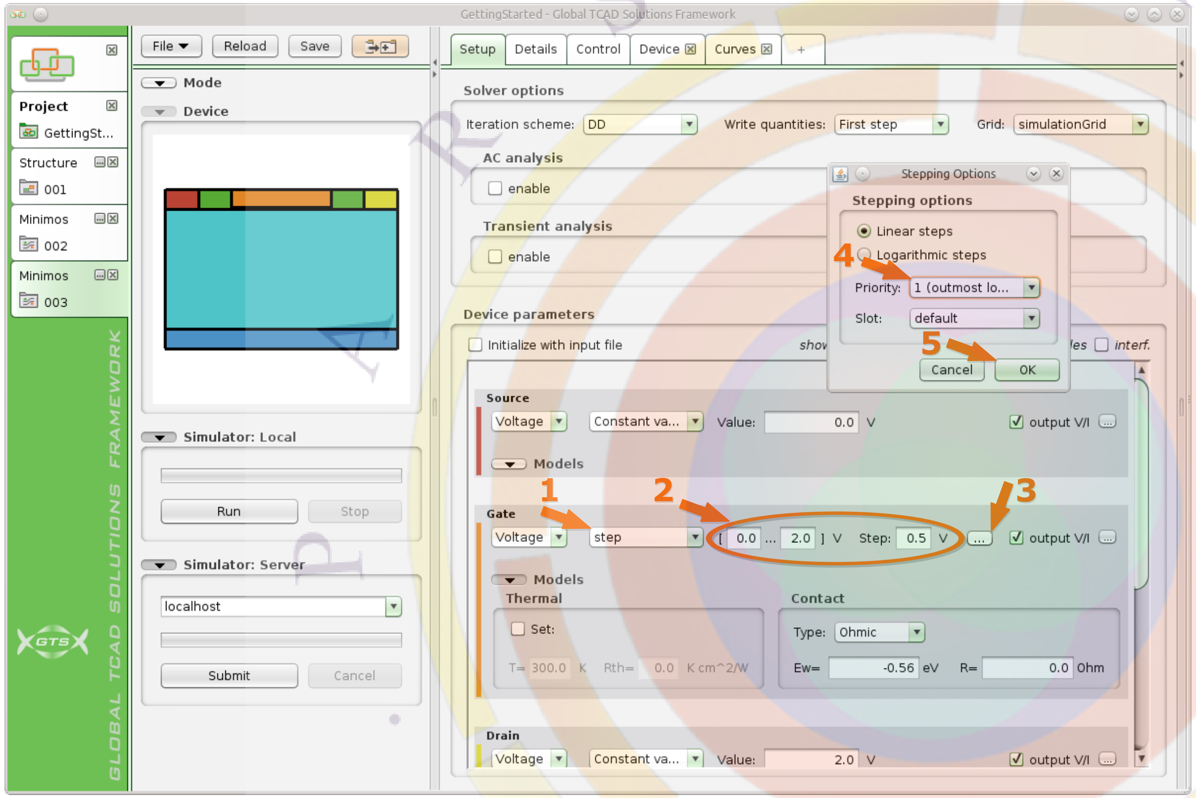
Note that ***GTS Framework*** creates a new ToolFolder and automatically opens it. We can instantly continue with our work in the new Toolfolder.



1.4.2. Edit Comment

Before we continue, lets set an appropriate comment for the new ToolFolder for later reference:

1. Press “File” in the menu bar
2. Click and edit the comment as you feel appropriate, press enter when done

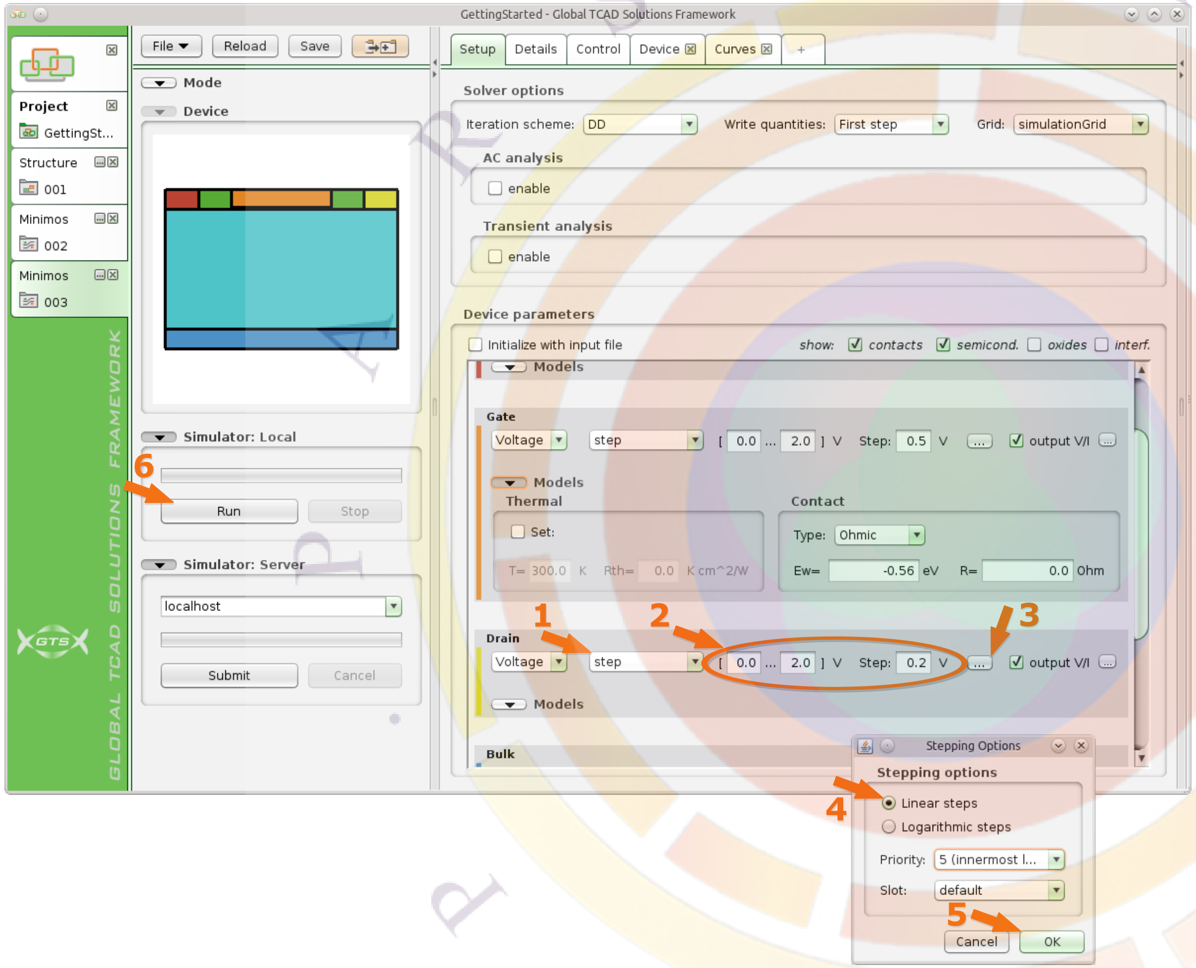


1.4.3. Setup Gate Voltage

For obtaining the output characteristics, we use two nested steppings for V_g and V_d, respectively. The nesting order is defined by the respective *stepping priorities*.

First we want to set up the stepping for the gate voltage (V_g):

1. Locate the *Gate* panel and make sure "step" is selected for the voltage
2. Define the stepping from 0.0 V to +2.0 V with a step size of 0.5 V.
3. Press the "..." button to open the "Stepping Options" dialog
4. Choose priority "1 (outmost loop)"
5. Confirm with "OK"



1.4.4. Setup Drain Voltage

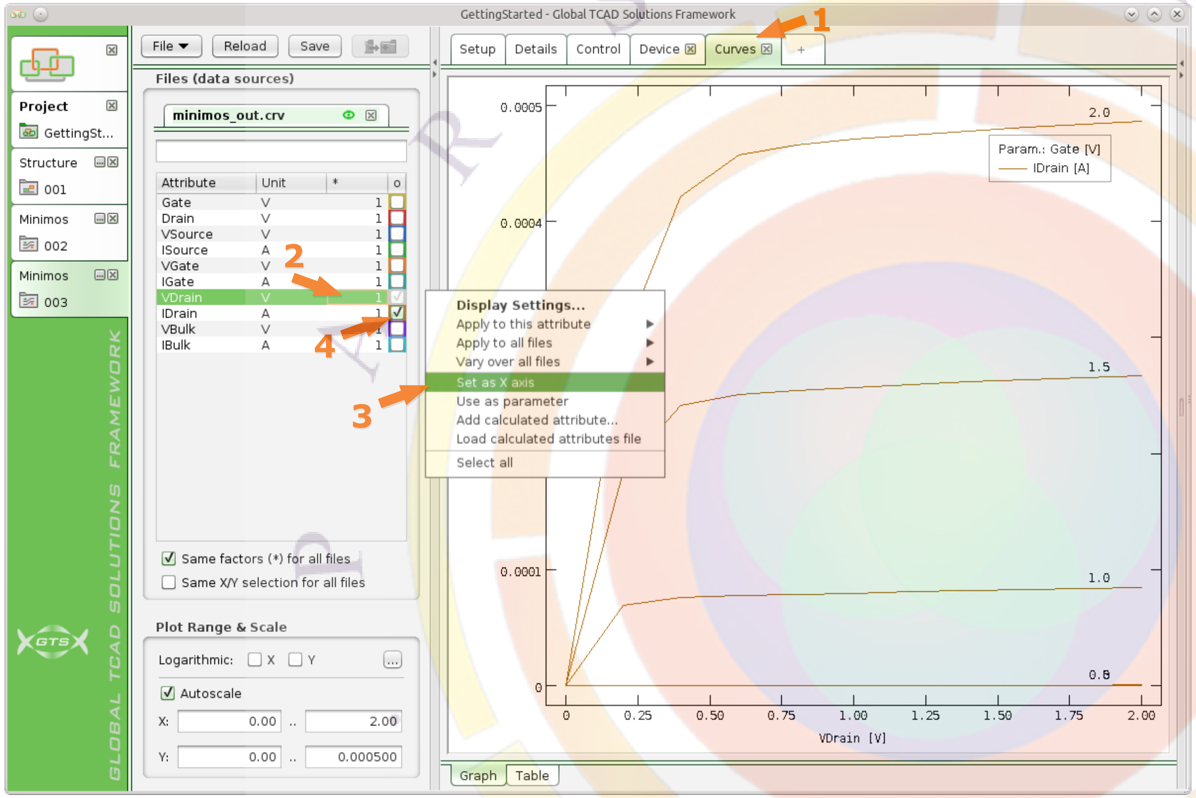
Second, we want to set up the stepping for the drain voltage (V_d). The *Priority* for the drain contact has to be higher than the *Priority* for the gate contact:

1. Locate the *Drain* panel and select "step" for the voltage
2. Define the stepping from $0.0\,\mathrm{V}$ to $+2.0\,\mathrm{V}$ with a step size of $0.2\,\mathrm{V}$.
3. Press the "..." button to open the "Stepping Options" menu
4. Choose priority "5 (innermost loop)"
5. Confirm with "OK"

Be sure to specify different stepping priorities. Two steppings with the same priority (or "Default") produce undefined results.

Start the simulation:

6. Click "Run"



1.4.5. Output Characteristics Plot

Now we can investigate the output characteristics of the 200nm NMOS FET:

1. Switch to the "Curves" page
2. Right-click the Drain voltage to show the context menu
3. Choose "Set as X axis"
4. Select the drain current *IDrain* to be plotted on the Y axis

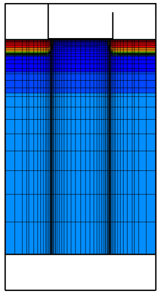
We have plotted the output characteristics. To navigate in the plot, use the mouse as described in Section 1.2.11.

- You can save the plot as a PNG image by selecting "File"–"Screenshot" from the menu.

You can always go to the *Project Home* and open one of the Tutorial ToolFolders to compare with the supplied results, or re-use the supplied data for your simulations by pressing the "Next tool" button.

Part 2

Scaled Transistor (90 nm)



In this example, we will modify parameters such as the gate length of the NMOS FET from example 1. We will simulate the resulting 90nm NMOS FET to demonstrate different ways of setting up simulations in ***GTS Framework***.

Like in the previous example, the transfer and output characteristics are calculated, by taking advantage of ToolFolders in the *Project Home*.

2.1. Device Generation

2.1.1. Creating a New Device

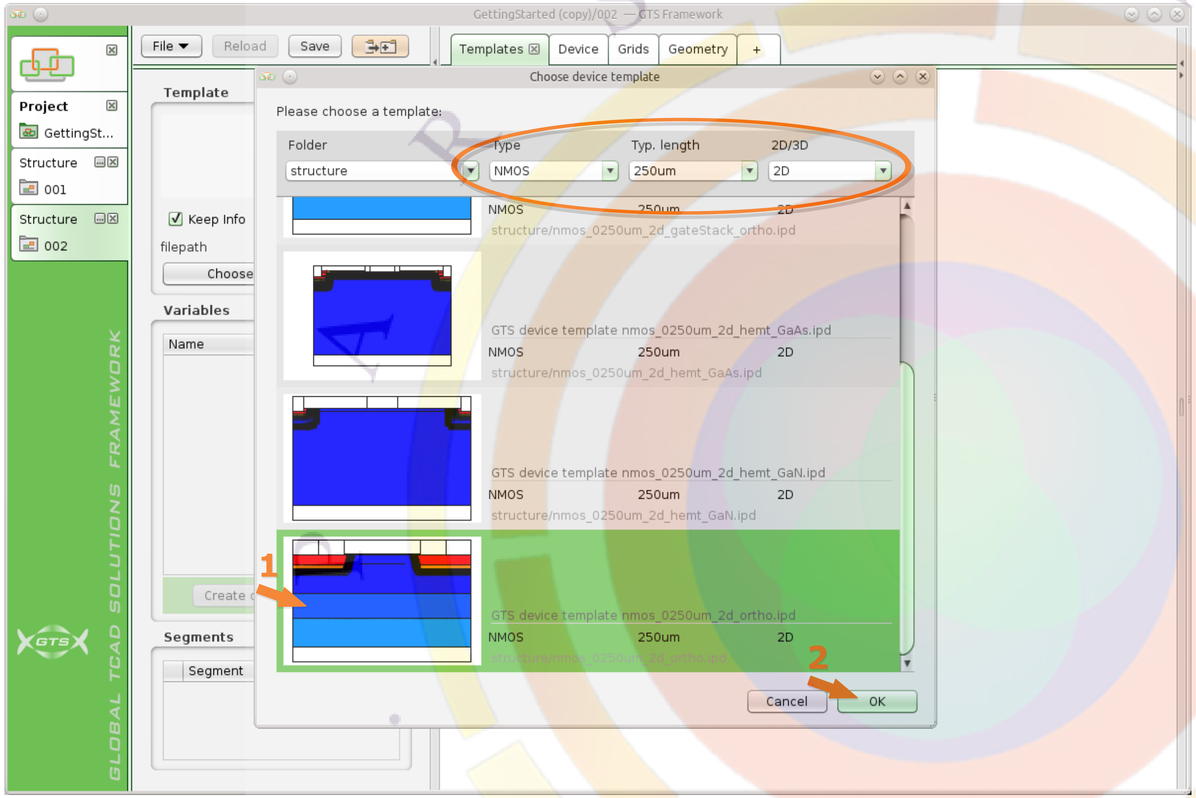
Now we want to simulate the transfer and output characteristics of a 90nm NMOS FET.

First, we create a GTS Structure ToolFolder for the device:

1. Go back to the *Project home*
2. Press “Add tool”
3. Select GTS Structure
4. Confirm with “OK”

(Instead of 3. and 4., you can also just double-click the desired tool.)

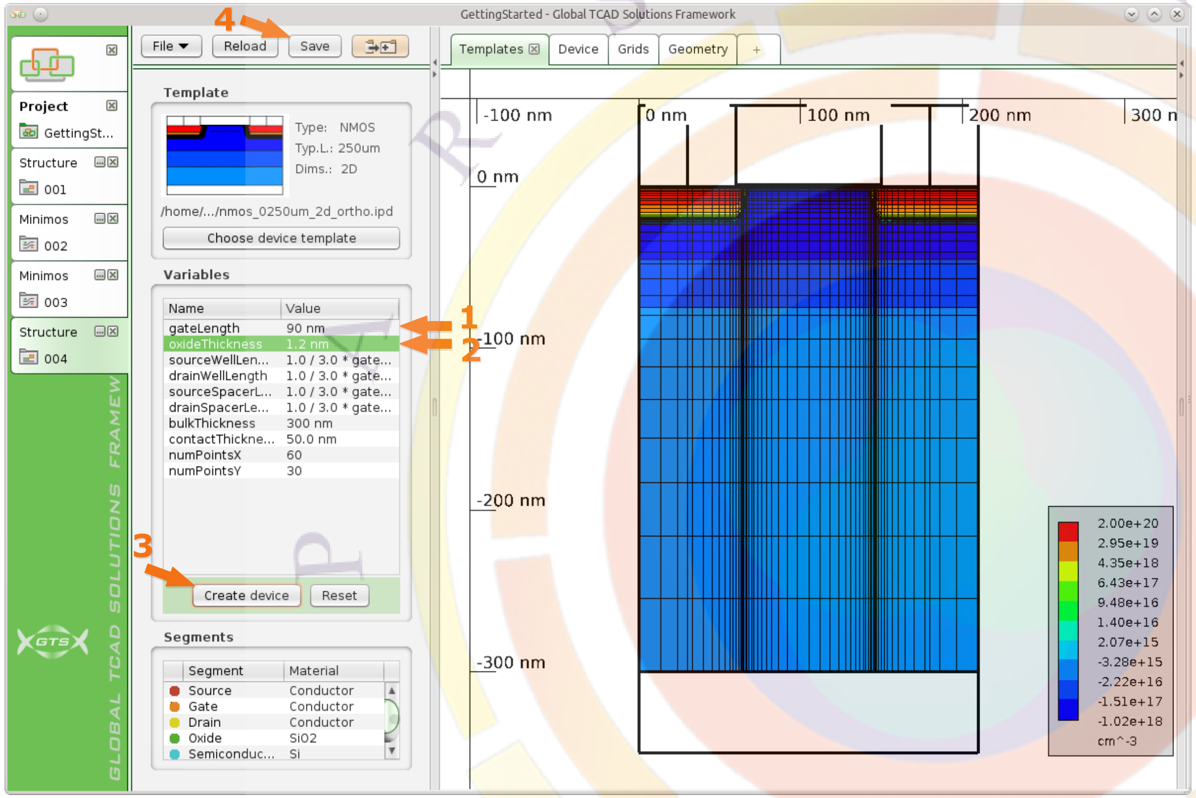
- In the new Structure ToolFolder, the File menu opens automatically so that you can add some notes, if you prefer.



2.1.2. Choosing a Template

We use the same device template as in Section 1.1.1 – therefore, choose “Create device from template” in the File-New menu, and select the template as follows:

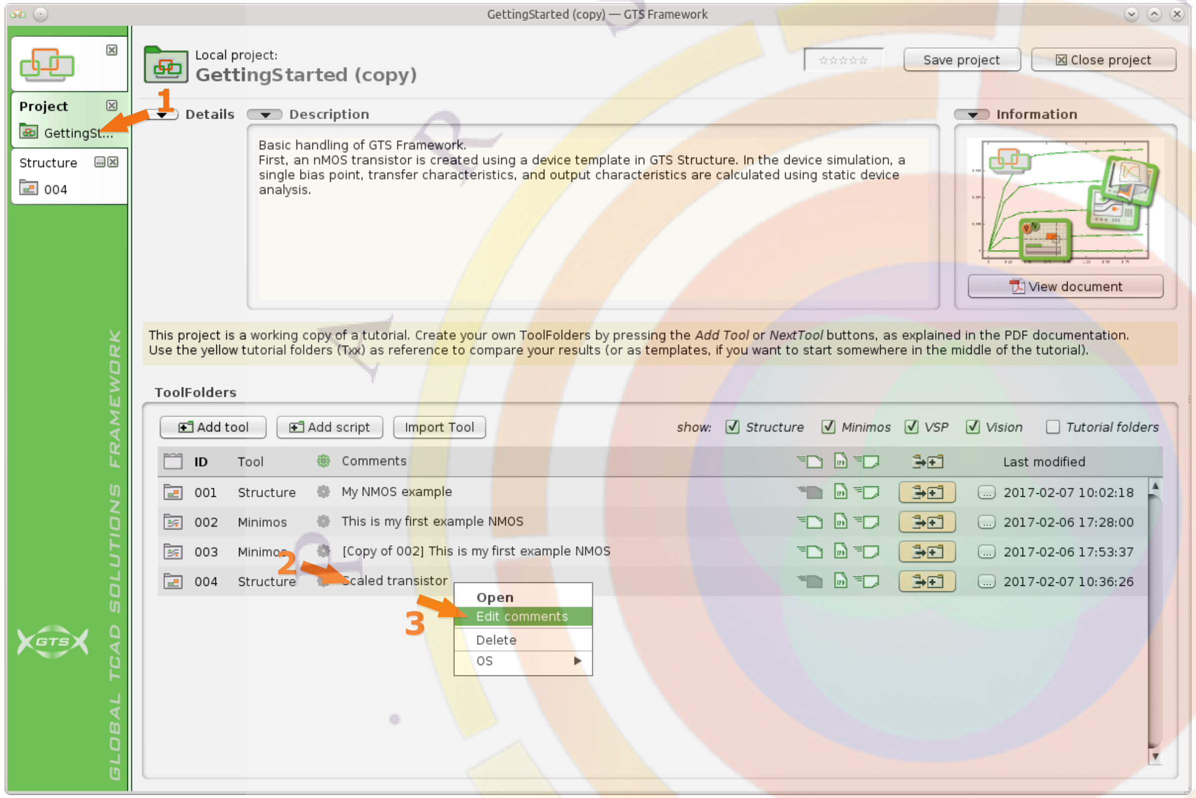
1. Set the filters to: *NMOS, 250um, 2D.*
2. Select the template with the name `nmos_0250um_2d_ortho.ipd`
3. Confirm with “OK”



2.1.3. Editing Template Parameters

We will now set different values for the gate length and the oxide thickness:

1. Enter a gateLength value of 90 nm
2. And an oxideThickness value of 1.2 nm
3. Click “Create device”
4. Click “Save” to save the device



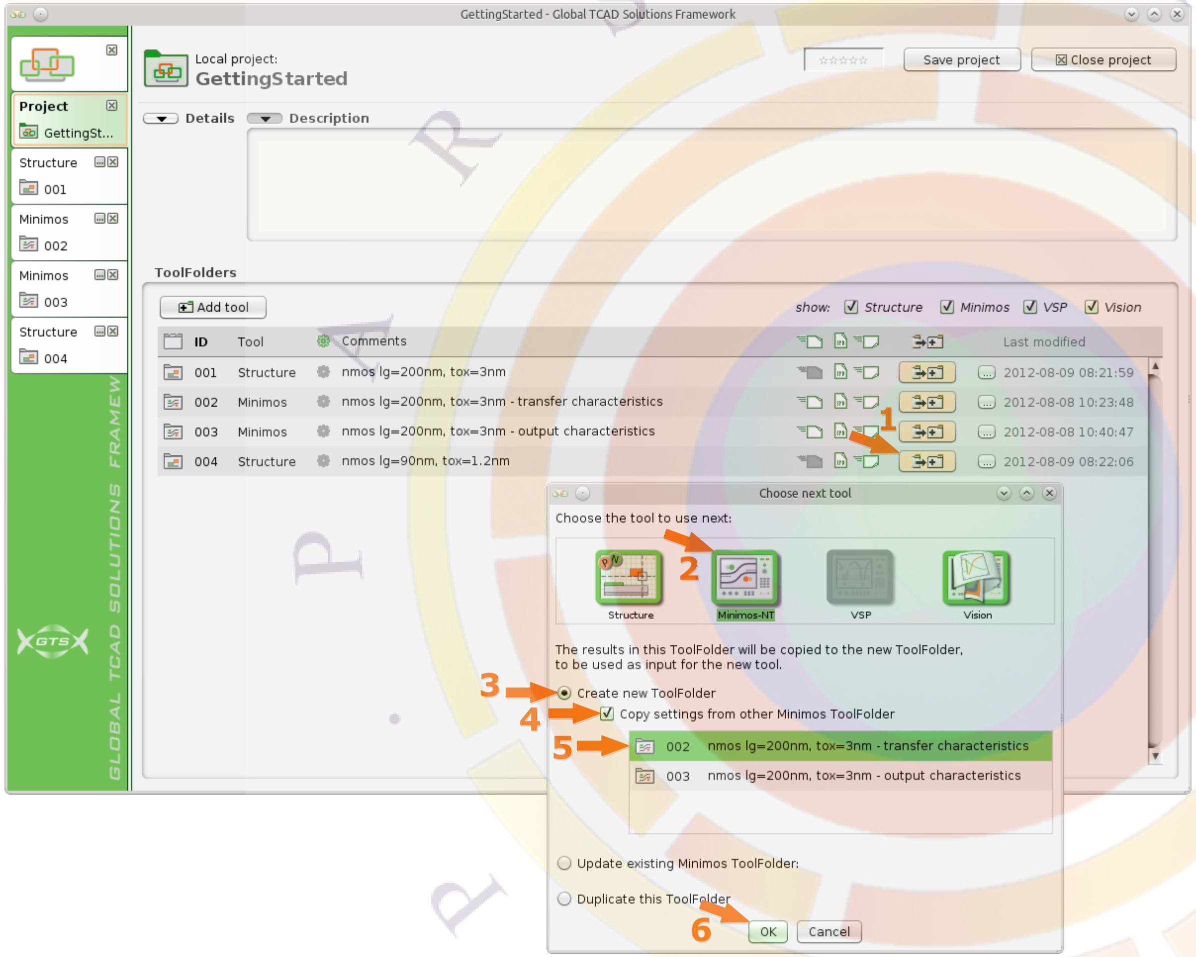
2.1.4. Project Home / Editing Comments

Many functionalities are accessible in the *Project Home* as well, which is handy if you want to use older ToolFolders that are not open at the moment. To illustrate this, we will do the next steps in the *Project Home*.

First, we can edit the comments of ToolFolders:

1. Switch back to the "Project home"
2. Right click the new GTS Structure ToolFolder
3. Choose "Edit comments" from the context menu
4. Type a comment you feel appropriate for the ToolFolder (e.g. "Scaled transistor")
5. Press Enter when done

The "Next tool" button is available in the *Project Home* as well. We will use it here in the next step, with an additional option.



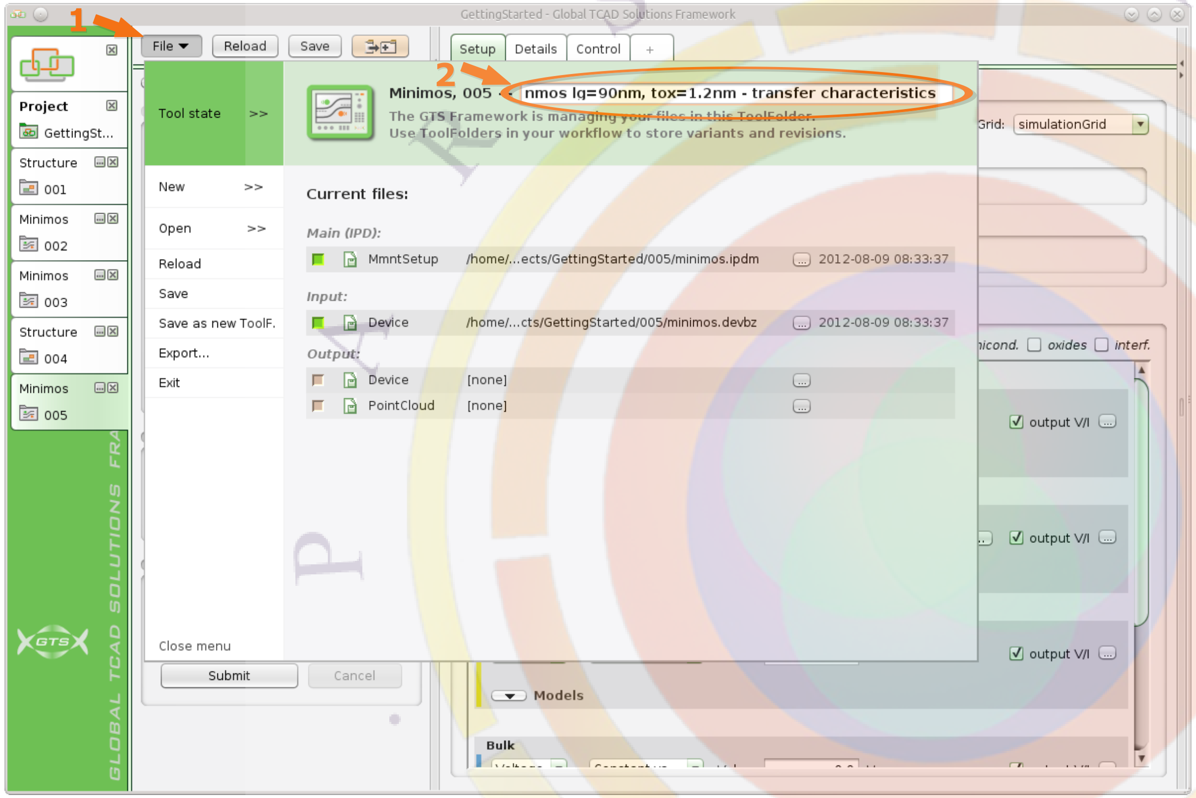
2.2. Transfer Characteristics

2.2.1. Copying Simulation Settings

GTS Framework offers a convenient way to use the same simulation settings for multiple devices. Here, we will re-use the previously created transfer characteristics setup for the modified device:

1. Press the "Next tool" button of the new GTS Structure ToolFolder (90nm)
2. In the *Choose next tool* dialog, select "Minimos"
3. Select "Create new ToolFolder"
4. Check "Copy settings from other Minimos ToolFolder"
5. Choose the ToolFolder where we previously created the transfer characteristics (notice comments)
6. Confirm with OK

GTS Framework creates a new Minimos ToolFolder, containing the 90nm device for input and the setup from the previous transfer characteristics simulation.



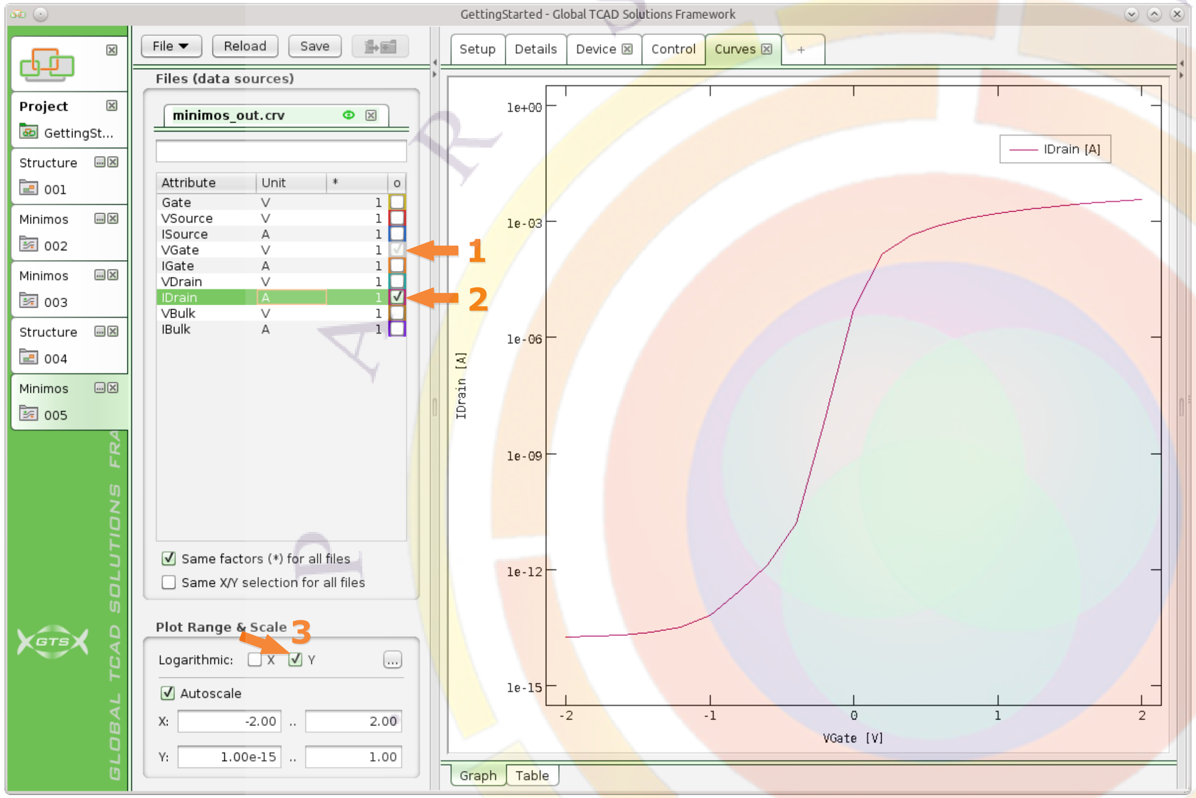
2.2.2. Editing Comments, Run

The new ToolFolder opens automatically. Let's assign a comment first:

1. Press "File" in the menu bar
2. Click on the comment to edit it, finish by pressing the Enter key, and close the menu (by clicking outside it or choose "Close menu")

Check the settings in the "Setup" page: All values have been copied from the previous simulation. So we can run the simulation without changing anything.

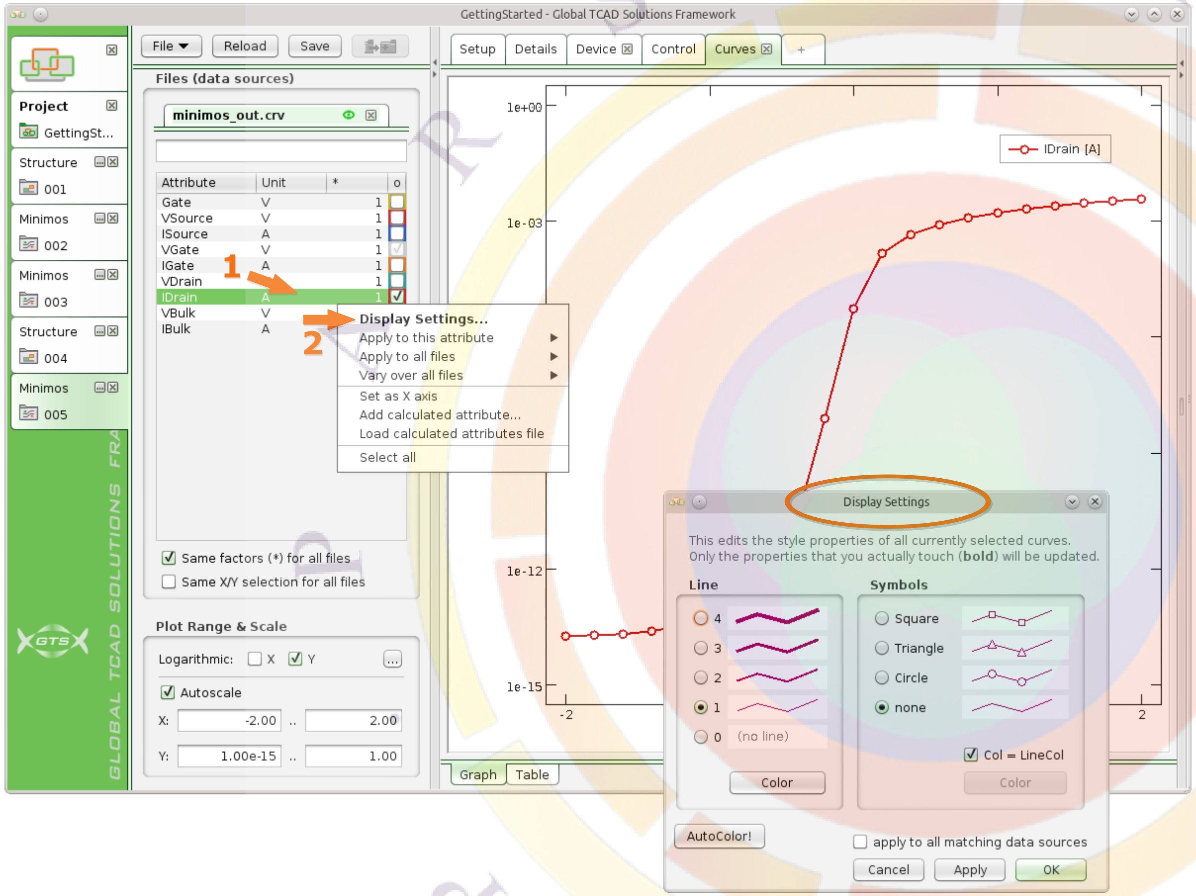
- Run the simulation by clicking the "Run" button, and confirm to save
- When the simulation is done, switch to the "Curves" page



2.2.3. Transfer Characteristics Plot

In the Control panel of the *Curves* page, we adjust the plot settings according to our needs:

1. Set the gate voltage *VGate* as X axis using its right-click context menu (leaving *Gate* would also be ok, but *VGate* is nicer in the plot)
2. Check the drain current *IDrain*
3. Set the plotting scale for the Y axis to logarithmic



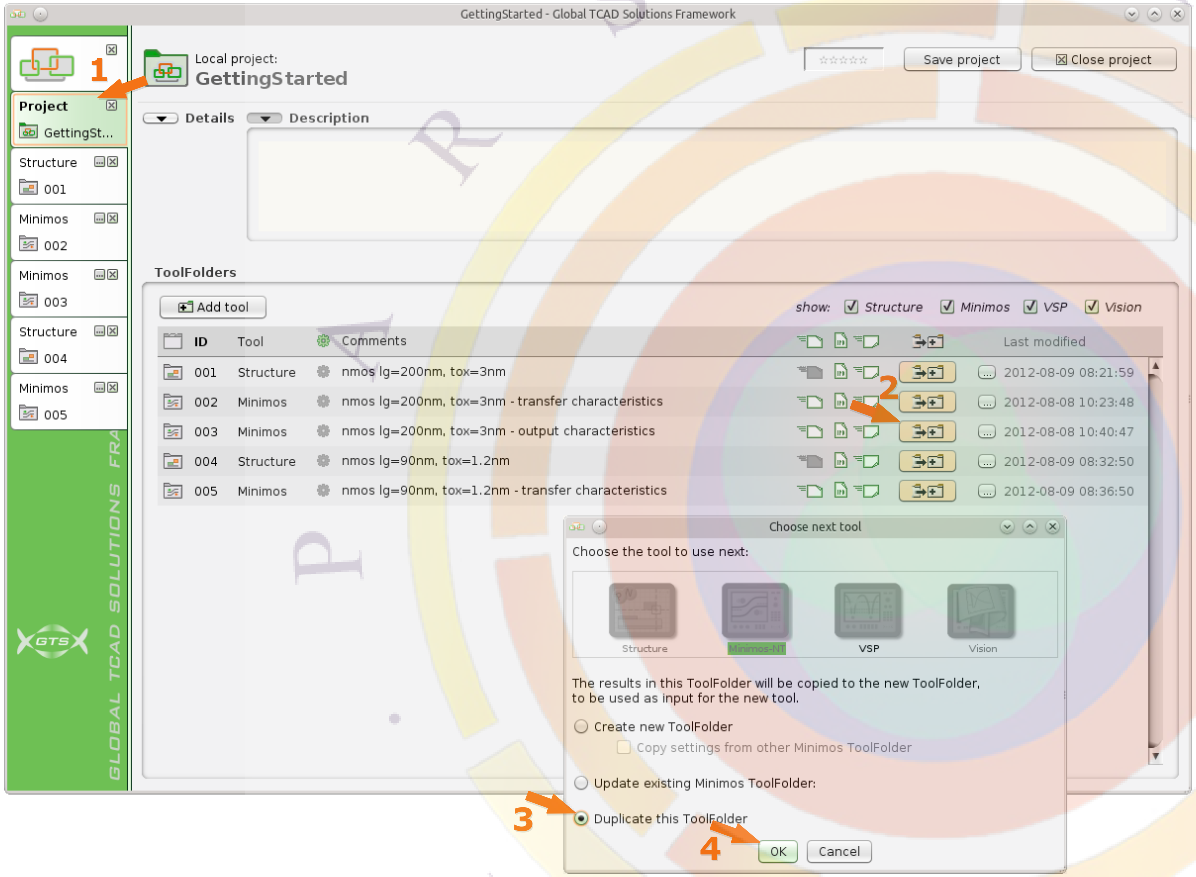
2.2.4. Display Settings

Further we can change the style of the curves:

1. Right click on the drain current quantity *IDrain*
2. Choose "Display Settings..." in the context menu

Explore various options to modify symbols, line styles and colors. (Click "Apply" to leave the dialog open while testing.)

You can apply the same color/line style to more than one curve, if you select multiple quantities before invoking the context menu (use the Shift or Ctrl keys for multi-selection).



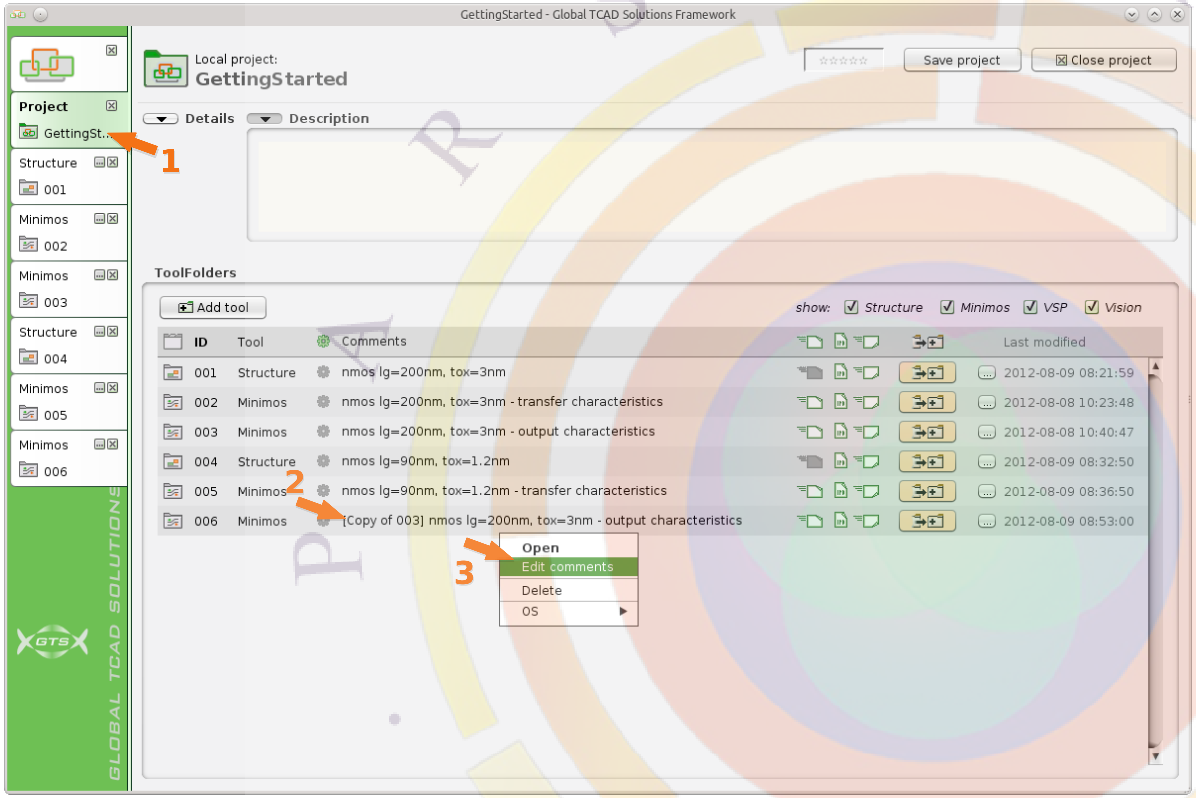
2.3. Output Characteristics

2.3.1. Duplicate Toolfolder

There is yet another possibility to run the same simulation setup on different devices, based on duplicating an existing simulation.

We will use this method now, first creating a duplicate of a Minimos ToolFolder and then exchanging the device in the duplicate:

1. Switch back to the “Project home”
2. In the list, locate the ToolFolder where we previously generated the output characteristics, and press its “Next tool” button
3. In the *Next tool* dialog, select “Duplicate this ToolFolder”
4. Confirm with “OK”

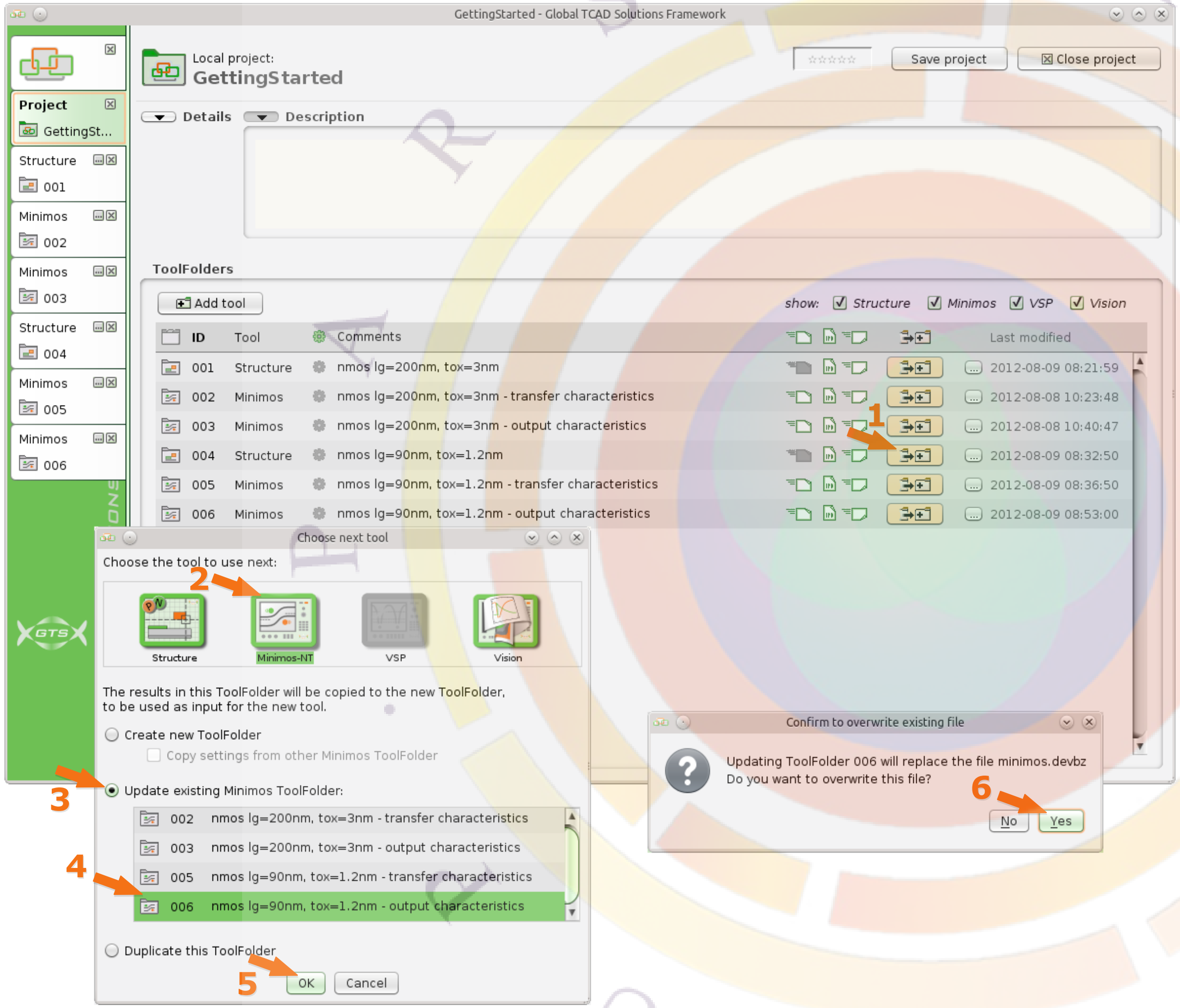


2.3.2. Editing Comments in the Project Home

The new ToolFolder is created and opened automatically. We switch back to the *Project Home*, and as usual, we want to assign a meaningful comment to the new ToolFolder:

1. Switch back to the “Project home”
2. Right click the new ToolFolder
3. Choose “Edit comments”, and edit the comment as appropriate, press Enter when done

Instead of right-clicking, the context menu is also accessible via the [...] button in each table row.

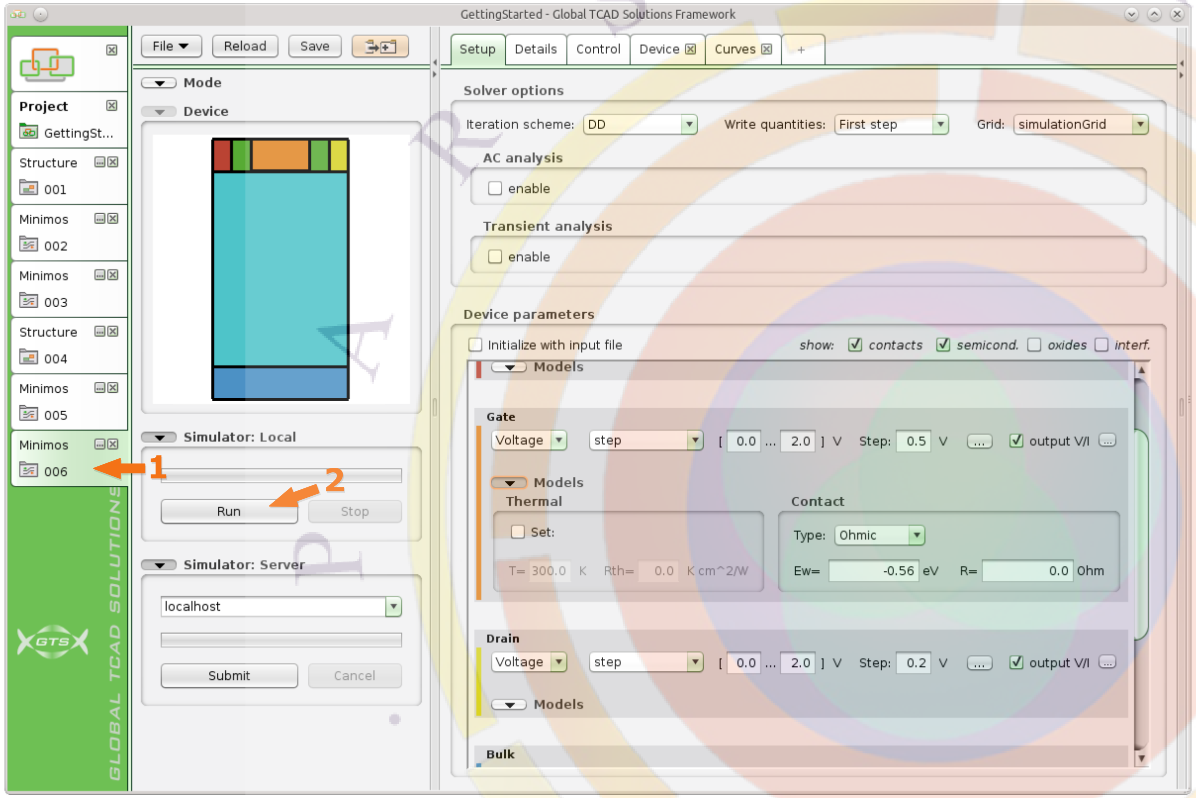


2.3.3. Updating a ToolFolder

In the newly created ToolFolder 006, the simulation settings are as required, but the input file is the 200 nm NMOS FET. But we can easily transfer the 90 nm transistor from ToolFolder 004 to our new ToolFolder 006:

1. Press the “Next tool” button of the Structure ToolFolder that contains the 90 nm transistor (probably 004)
2. In the Next tool dialog, make sure that Minimos is selected
3. Select “Update existing ToolFolder”
4. As target, choose the newly-created ToolFolder (probably 006)
5. Confirm with “OK”
6. When asked, confirm to overwrite the old `minimos.devbz` file

This was to show how you can use the result of a ToolFolder (e.g. Structure) to update the input of another ToolFolder (e.g. Minimos) that is already existing.



2.3.4. Updated ToolFolder

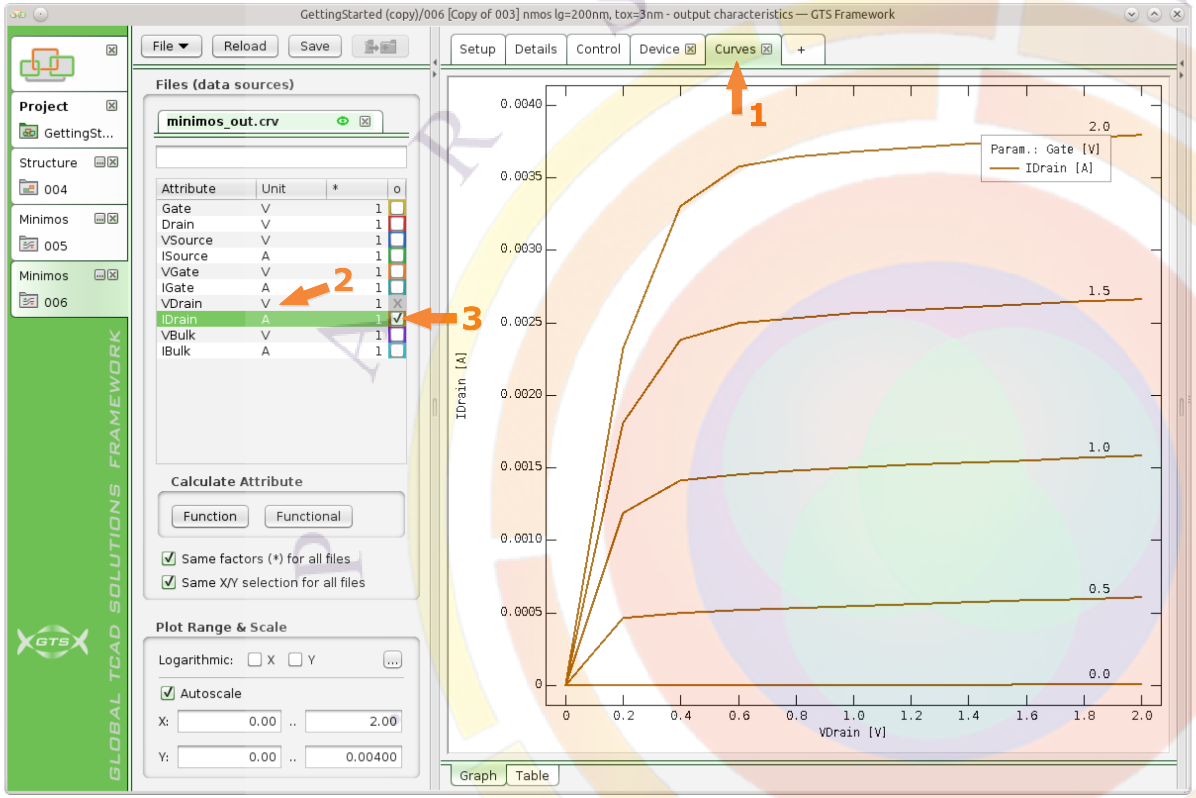
Now we go back to the new Toolfolder 006, where the setup and the device are in place.

1. Switch to the updated ToolFolder 006, click "Reload" to verify the update.

This procedure (steps 2.3.1 and 2.3.3) is an alternative to the one shown in 2.2.1 and illustrates the "Update ToolFolder" functionality.

Now we can start the simulation as usual:

2. Click the "Run" button in the Control panel, confirm when asked to save.



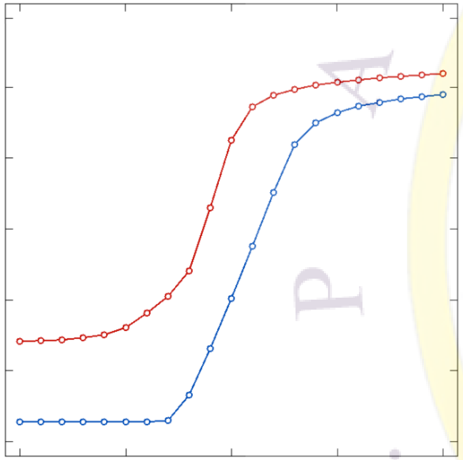
2.3.5. Output Characteristics Plot

When the simulation is done, plot the results:

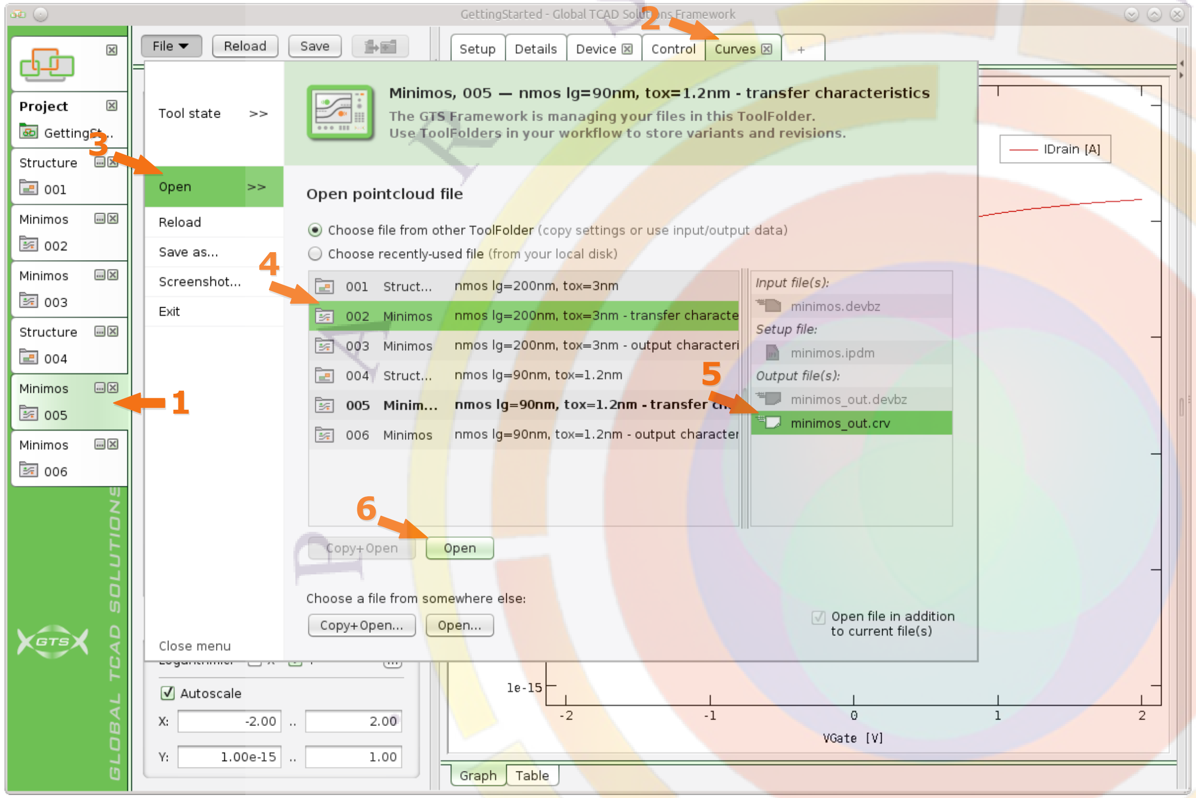
1. Switch to the “Curves” page
2. In the context menu of the drain voltage *VDrain* select “set as X axis”
3. Check the drain current *IDrain* to show on the Y axis

Part 3

Comparing Results



In the following, the results of the previous examples, i.e. the transfer characteristics of the 200 nm and the 90 nm NMOS FETs are compared.

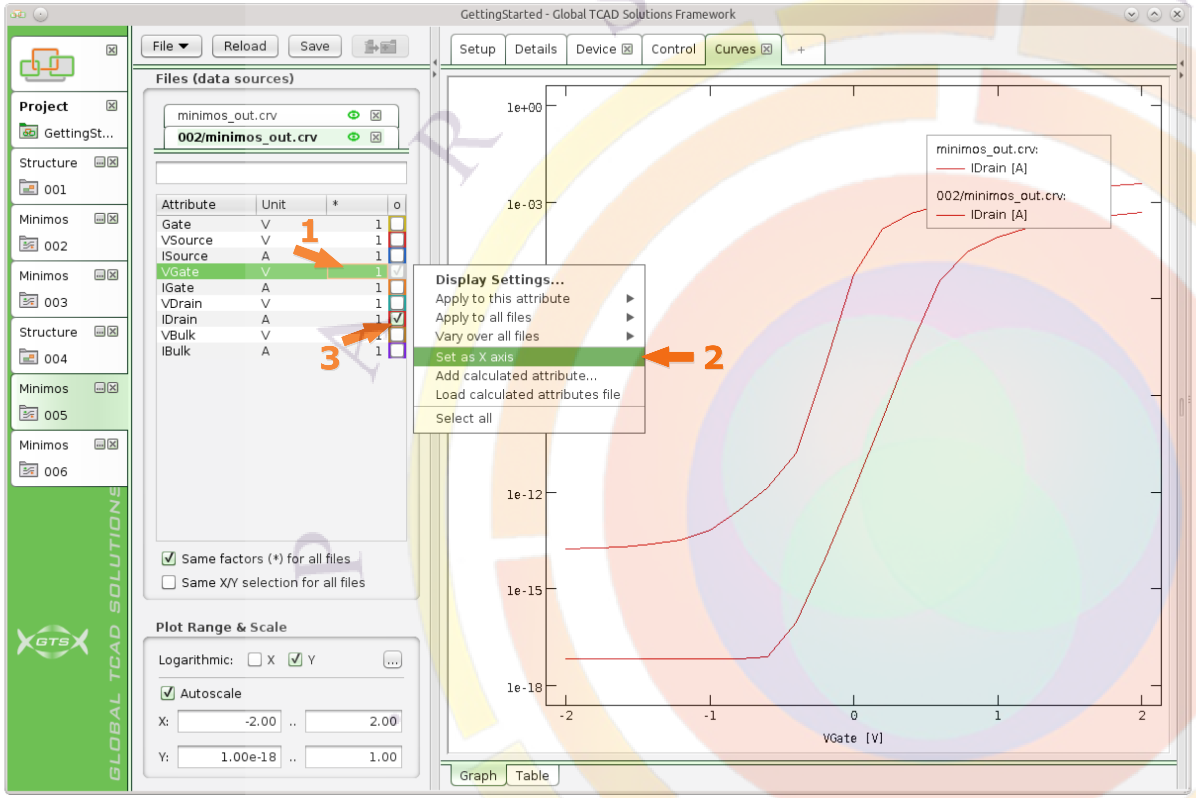


3.1. Transfer Characteristics

3.1.1. Select Simulation to Compare

Now we can compare the results of the 200nm NMOS and the 90nm NMOS:

1. Switch to the ToolFolder “005” (where we calculated the 90nm transfer characteristics)
2. Verify that the “Curves” page is active (important, because only the *Curves* page can show crv files)
3. Select “File”, “Open” (here you can open files from other ToolFolders as well as from other locations)
4. Choose the ToolFolder “002” (where we calculated the 200nm transfer characteristics)
5. Select the file `minimos_out.crv`.
6. Press “Open”



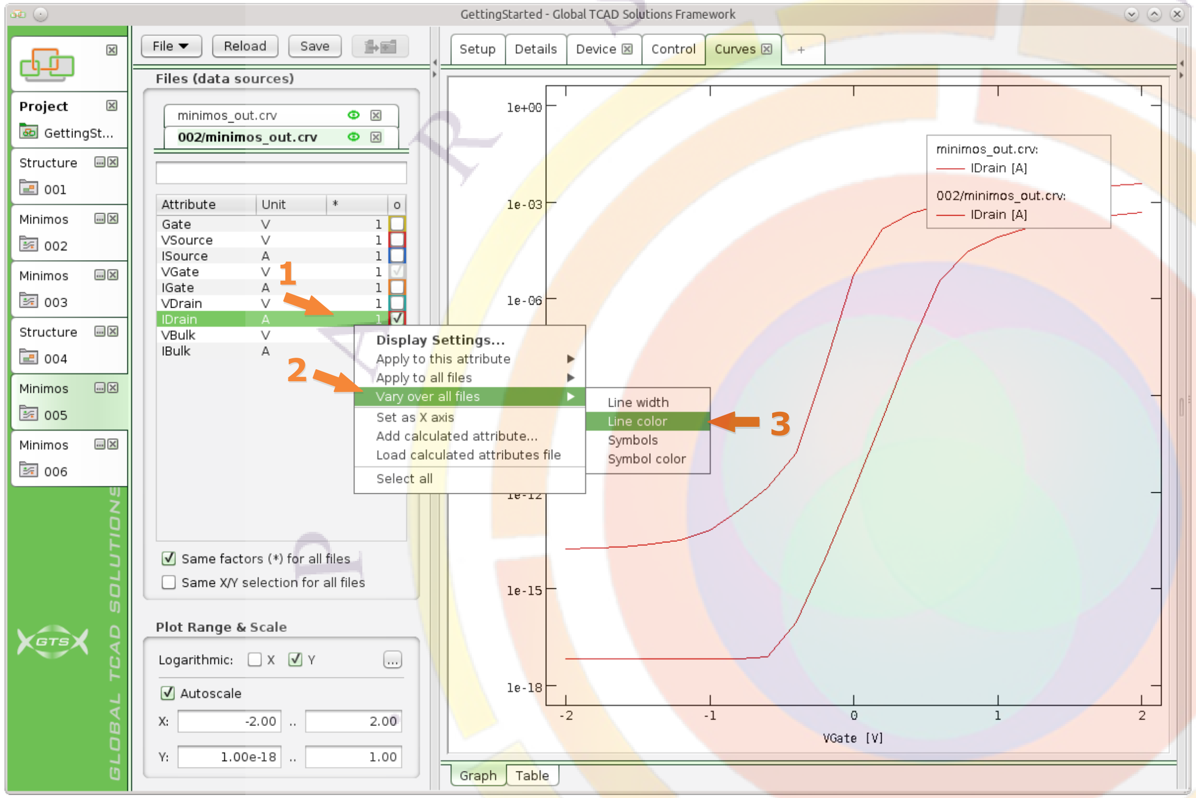
3.1.2. Plot Configuration

The data we want to compare is loaded. Note the second file entry in the *Files* group at the top of the Control panel:

- By clicking on the file name, you can switch between the files to access the attributes and display settings for each of them.
- You can click on the green eye to show/hide the respective data in the plot.

The plot should already look like in the picture. If not, go through the following steps:

1. Right click the gate voltage quantity *VGate*
2. Choose "Set as X axis" in its context menu
3. Choose the drain current *IDrain* as Y axis

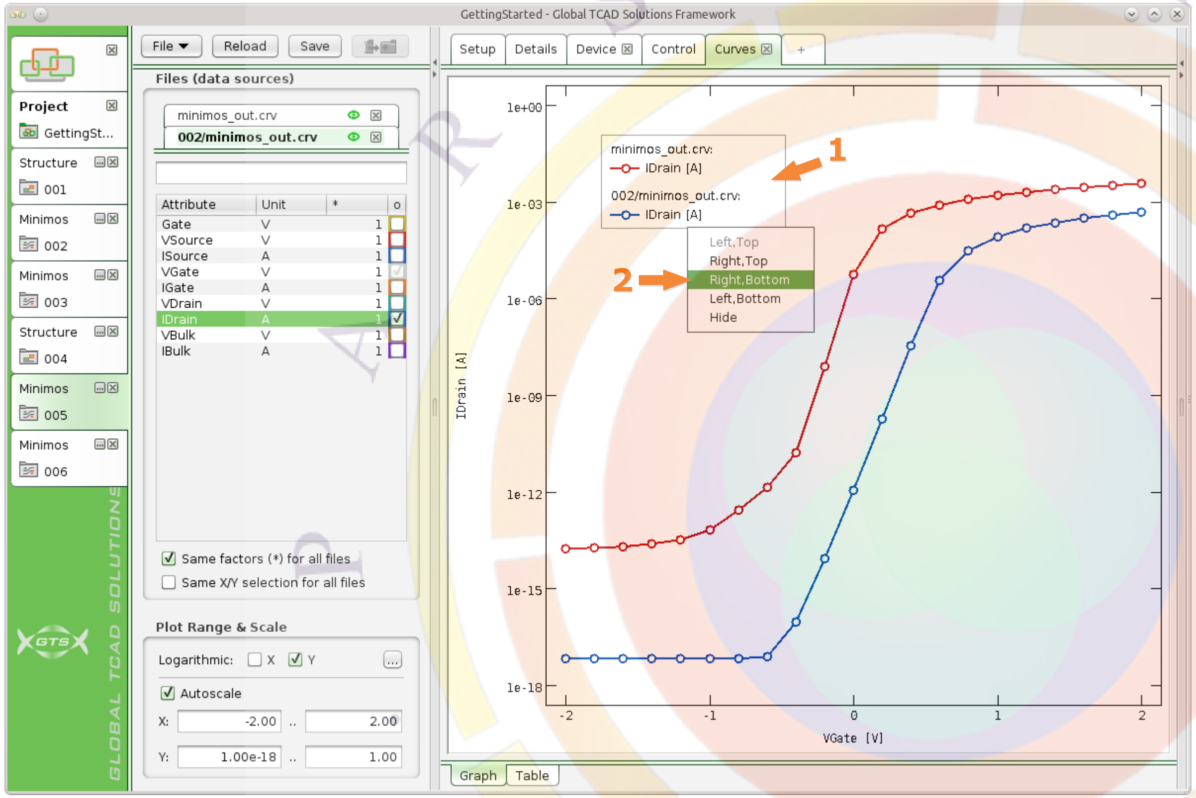


3.1.3. Display Settings for Multiple Curves

An easy way of assigning different colors to multiple curves is:

1. Right click the drain current *IDrain*
2. Choose “Vary over all files”
3. Select “Line color” in its context menu

In addition, one can change the line width, symbols, and symbol color, either for one or for all files (data sources). Analogously, you can use the “Display settings” dialog, which also has the option “Apply to all data sources”.



3.1.4. Moving the Legend Box

The legend box can be moved to the desired corner:

1. Right click the legend box
2. Select the desired location in the context menu

Conclusion

The general handling of ***GTS Framework*** has been introduced by rather basic examples. The basic uses of ***GTS Structure*** and ***Minimos-NT*** have been illustrated as well as the concept of *ToolFolders* and the related workflow functions.

We hope you have found this tutorial useful to get a first impression, and would like to invite you to take a closer look at some of the more complex examples mentioned below.

Tutorials Included In This Release

You are welcome to continue with the other tutorials included in this release – you can open them just like this one in the projects list (highlighted in yellow).

More Complex Tutorials And Application Examples

More sophisticated tutorials and examples are available online via myGTS at `https://globaltcad.com/mygts`. Downloading them to your projects folder makes them visible in ***GTS Framework***. Previews of most tutorials are viewable online at `http://www.globaltcad.com/en/solutions.html`.

For additional information, please refer to `http://globaltcad.com/` or feel free to contact us at `info@globaltcad.com`.

Appendix A

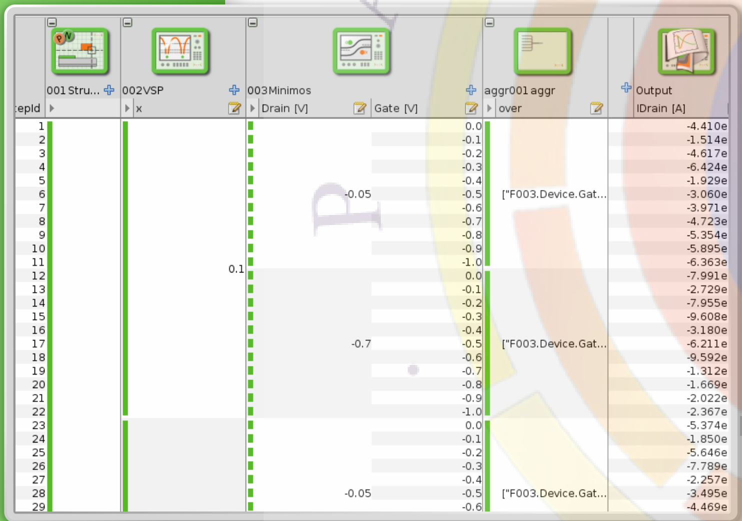
ToolFolder List

The project `GettingStarted_I` contains the following ToolFolders (TF):

TF	Tool	Description
T01	Structure	nmos lg=200nm, tox=3nm
T02	Minimos	nmos lg=200nm, tox=3nm, transfer characteristic
T03	Minimos	nmos lg=200nm, tox=3nm, output characteristic
T04	Structure	nmos lg=90nm, tox=1.2nm
T05	Minimos	nmos lg=90nm, tox=1.2nm, transfer characteristic
T06	Minimos	nmos lg=90nm, tox=1.2nm, output characteristic

GLOBAL TCAD SOLUTIONS — GTS FRAMEWORK — TUTORIAL

GTS FRAMEWORK



Simulation Flow with DOE, Process and Device Splits

Getting Started II Tutorial

global TCAD solutions®

Simulation Flow with DOE, Process and Device Splits – Getting Started II Tutorial,
GTS Framework Release 2017.03
Revision of November 9, 2017

Global TCAD Solutions GmbH
Bösendorferstraße 1/12
1010 Wien
Austria / Europe

Phone: +43 1 9255049
Email: info@globaltcad.com

Additional information is provided at: http://www.globaltcad.com/

Cover & layout: prausedesign

Contents

Introduction

Using This Tutorial

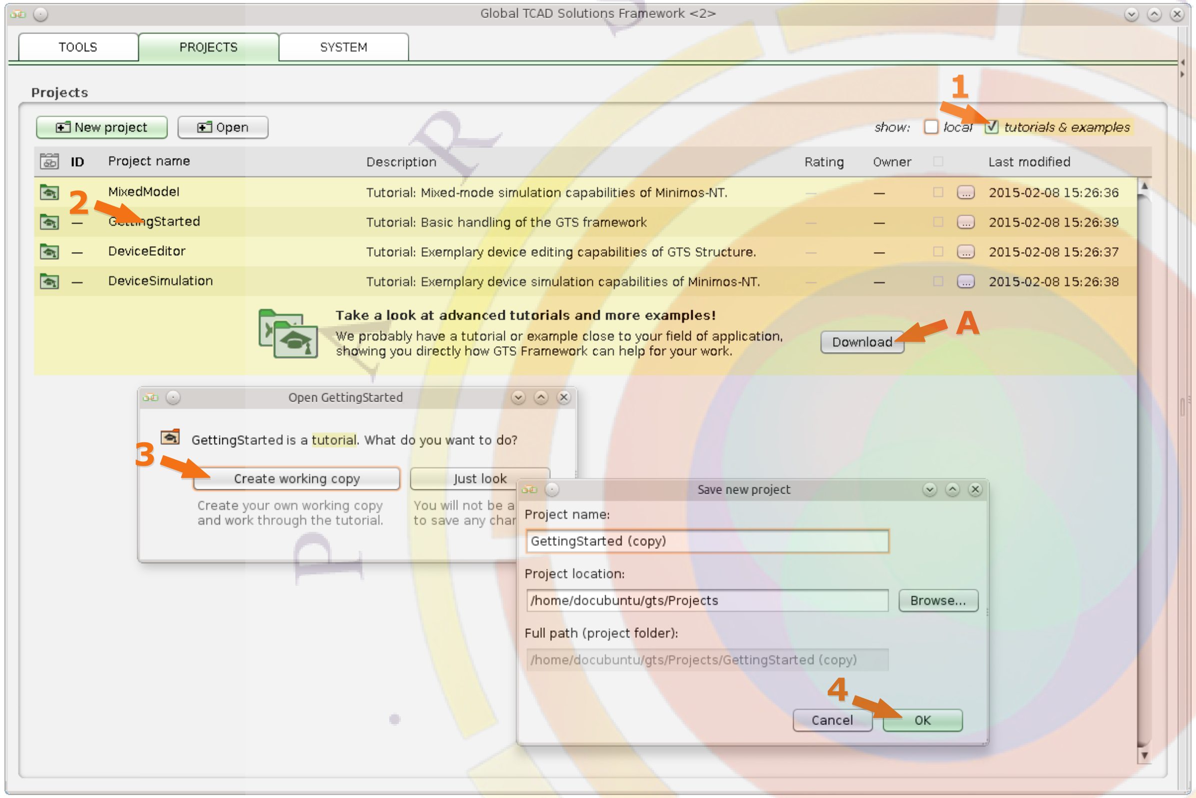
GTS *tutorials* generally consist of a written description (this text) and a GTS *project* (simulation data) which you can open in ***GTS Framework*** (yellow items in project list). As shown below, we recommend to create a working copy from the project, and proceed step by step, guided by this text. If not yet familiar, please refer to the *GettingStarted* tutorial.

Working in a copy of a tutorial project, you can open the yellow Txx ToolFolders at any time. They contain the data of the described simulations, which you can use for reference or as starting points for your own simulations.

Only a few tutorials are included with the release; the others are available via *MyGTS* at `https://globaltcad.com/mygts`. When logged in, please download the respective file and extract it to the gts projects folder on your PC. After restarting GTS Framework, the tutorial project will appear in the *Projects* list.

This is an intermediate (level 2) tutorial.

This tutorial was created using GTS Framework Release 2017.03. Other releases might need adjustments or have slightly different user interfaces.



Project / Working Copy

If you have not already done so, go to the *Framework Home* and locate the project associated with this tutorial in the *Projects* list, and create your working copy from it:

1. Check "tutorials & examples"
2. Click on the respective tutorial project: ***Simulation Flow with DOE, Process and Device Splits***
3. Choose "Create working copy"
4. Check the project name, click "OK"

The project is created and opened, so that you see the *Project Home*. — *Ready to start!*

A. If you miss the respective project, please download it via *MyGTS* (click "Download", see previous page).

Alternatively, you can start the tutorial with an empty project – just create a new project. (The advantage of the tutorial project is that you have the results at hand any time.)

Part 1

DOE Workflow

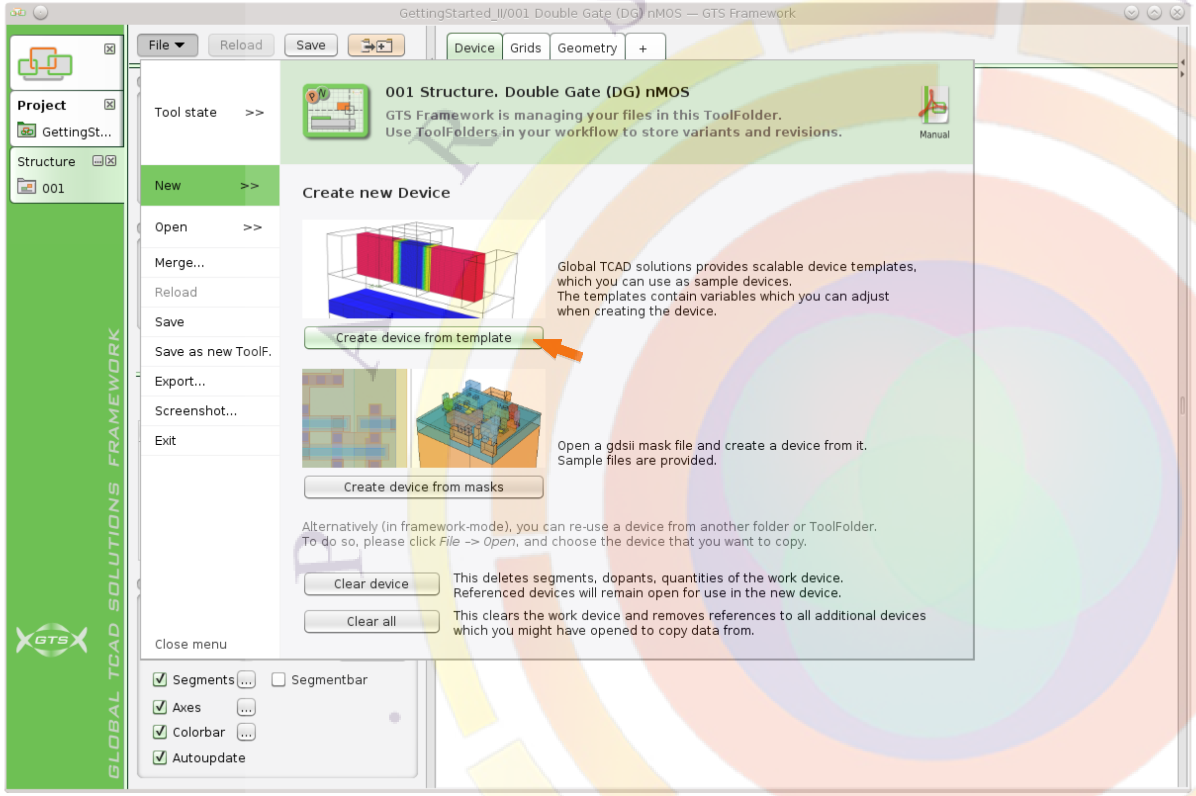
This tutorial demonstrates the use of the Design of Experiment. A workflow with structure generation, device simulation and parameter extraction is shown. For better comparison of various technologies, Off-current normalization is applied.

It is assumed that you are already familiar with the main concepts of GTS Framework and the idea of projects and ToolFolders, as explained in the “Getting Started Part I” tutorial.

1.1. Simulation Setup

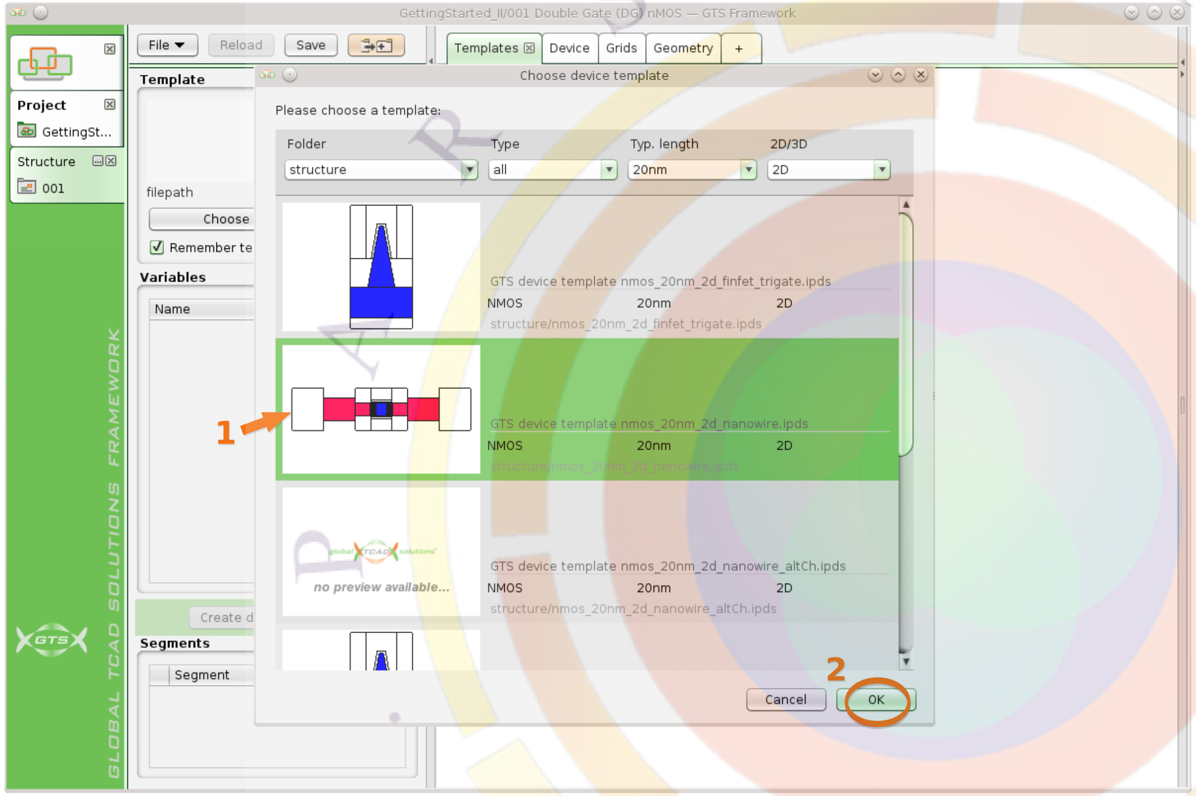
1.1.1. Create Structure

- Click the Add tool
- Add a new Structure
- Press "OK"



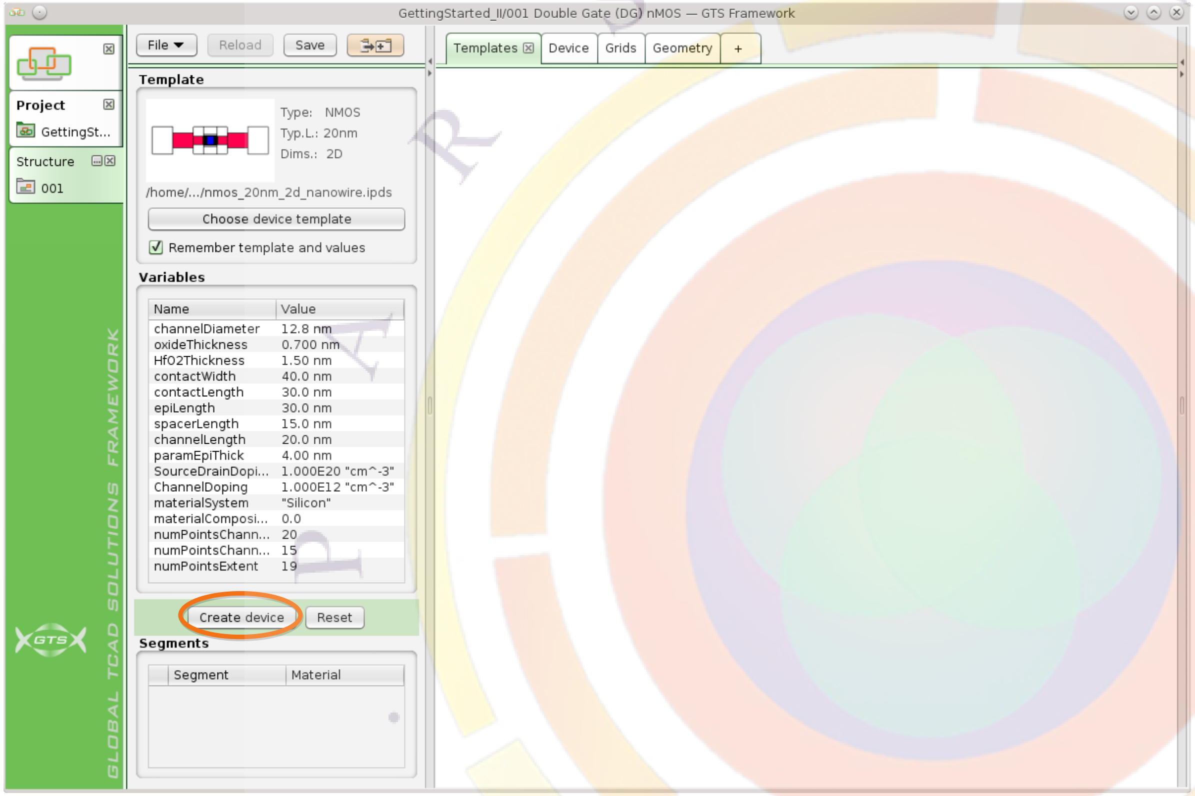
1.1.2. Device Templates

- Click "Create device from template" to open the templates dialog



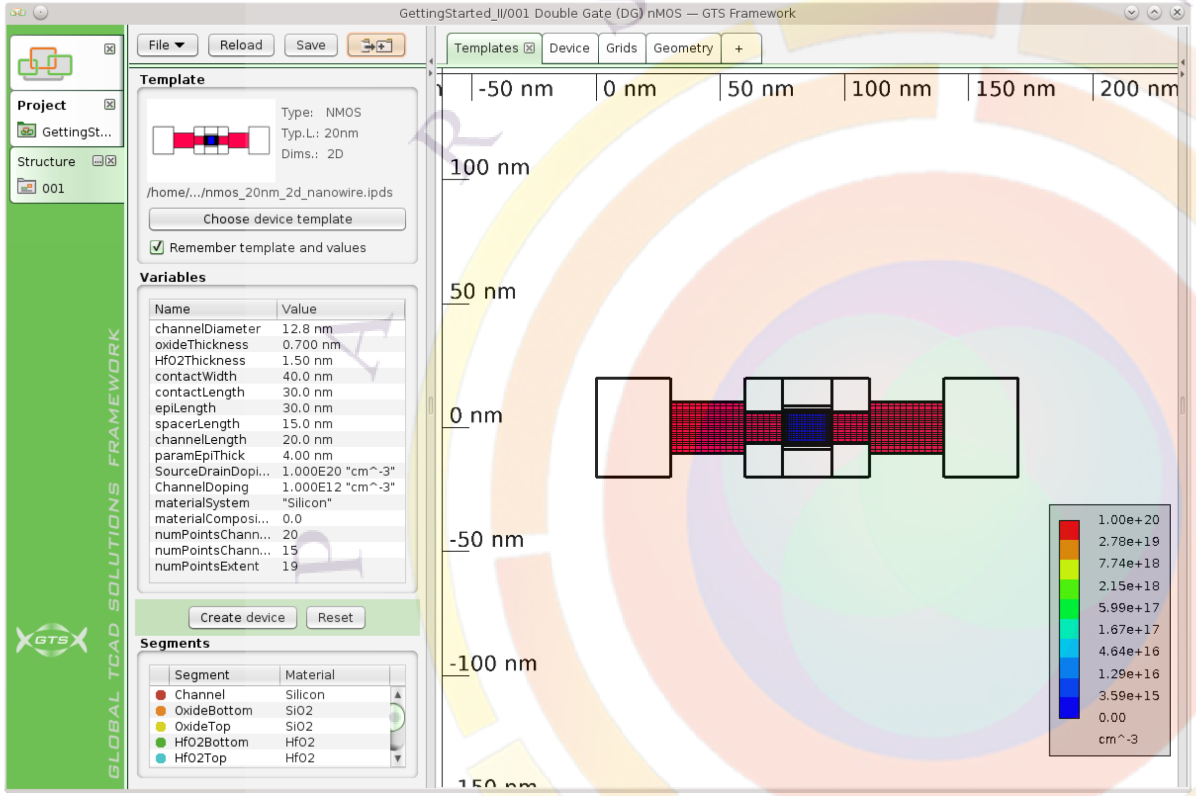
1.1.3. Nanowire Structure

- From the device templates, select `structure/nmos_20nm_2d_nanowire.ipds`
- Press "OK"



1.1.4. Create Device 1

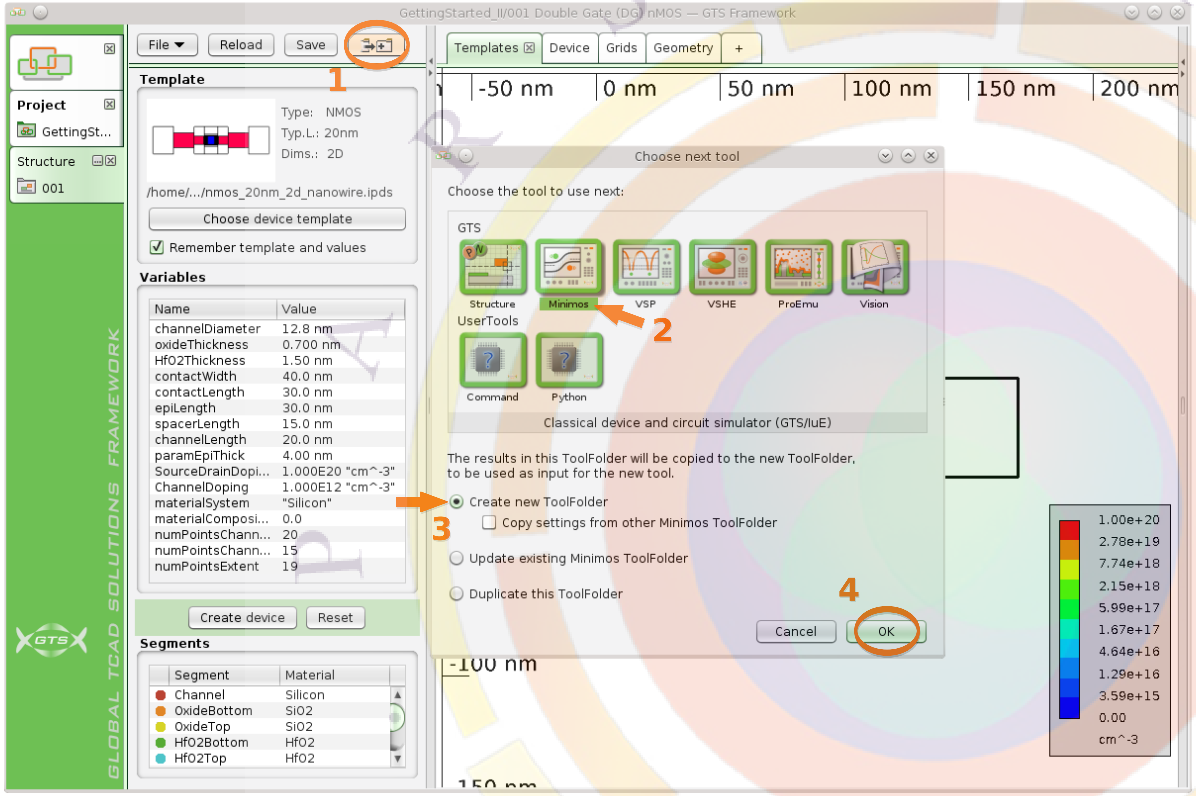
- Press "Create device"



1.1.5. 20nm 2D Nanowire Structure

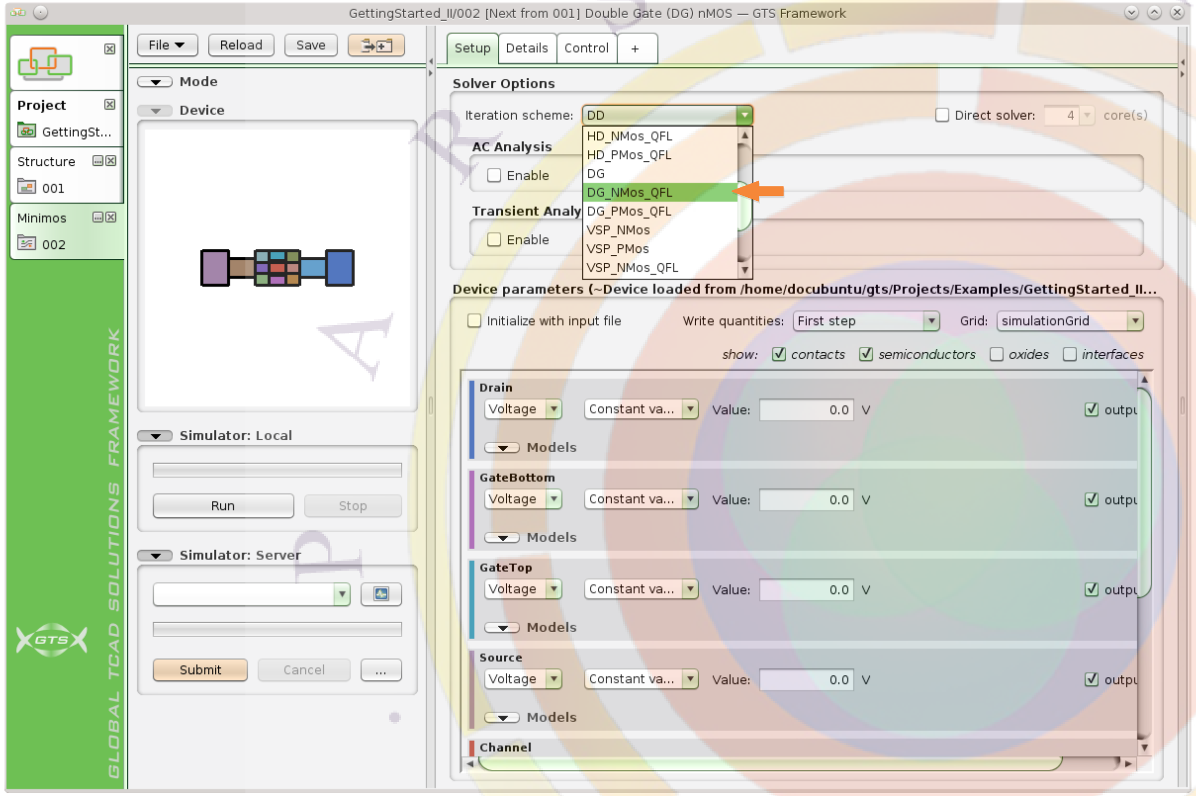
- The device structure is shown

You can modify the parameters and click "Create device" again.



1.1.6. Minimos

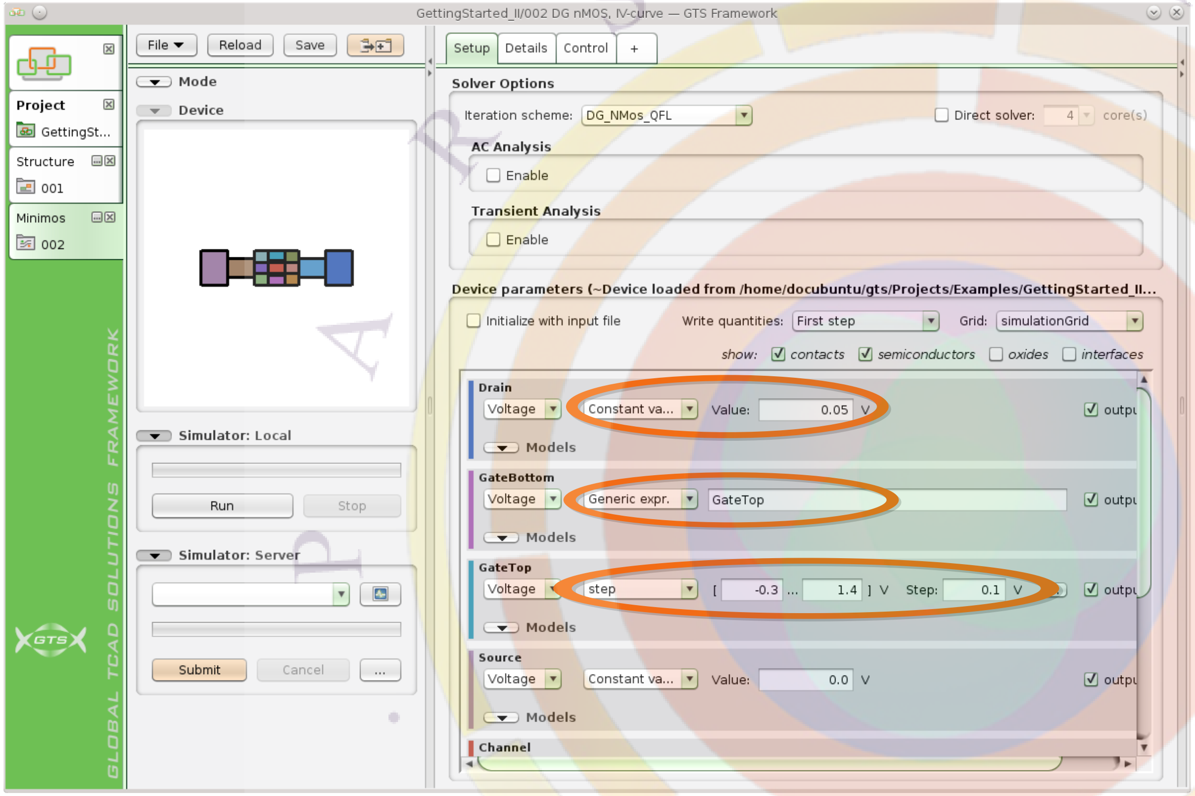
1. Press the "next tool" button and press "Yes" to save the structure.
2. Select "Minimos" and "Create new ToolFolder"
3. Press "OK"



1.1.7. Simulation Setup 1

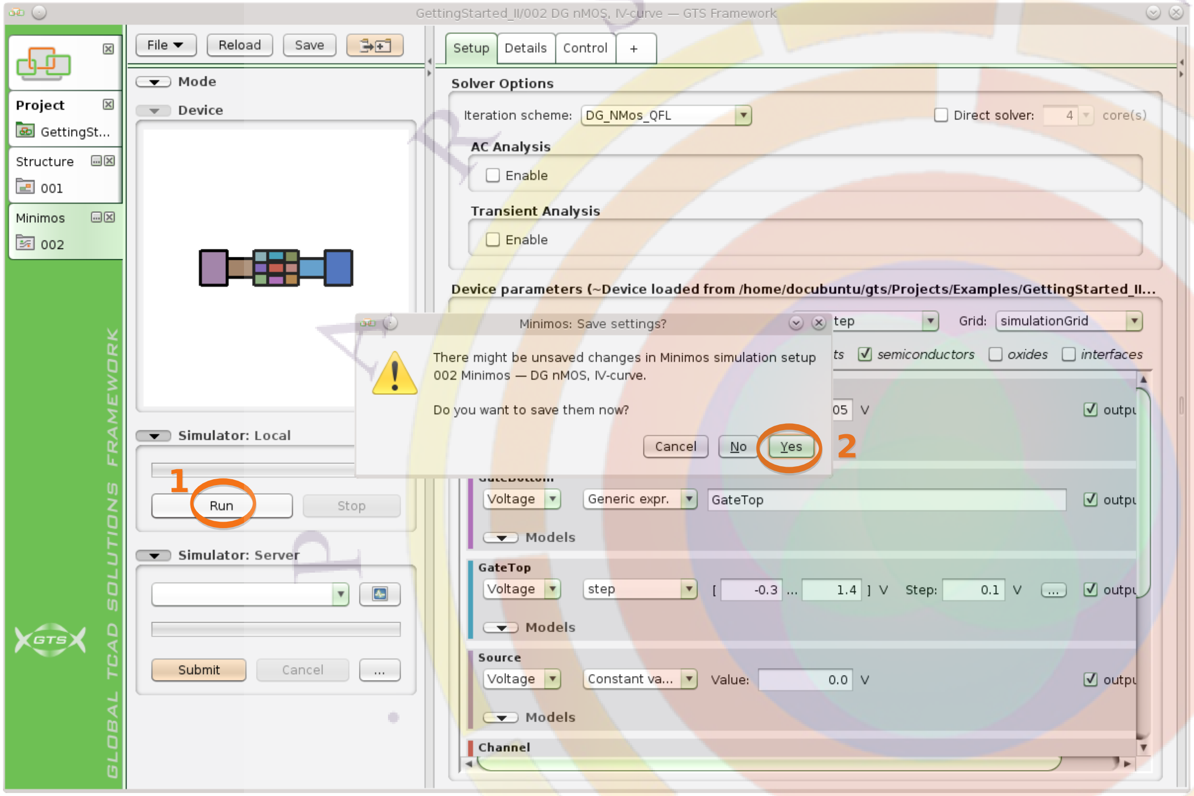
A new Minimos ToolFolder opens.

- Select the iteration scheme `DG_NMos_QFL`



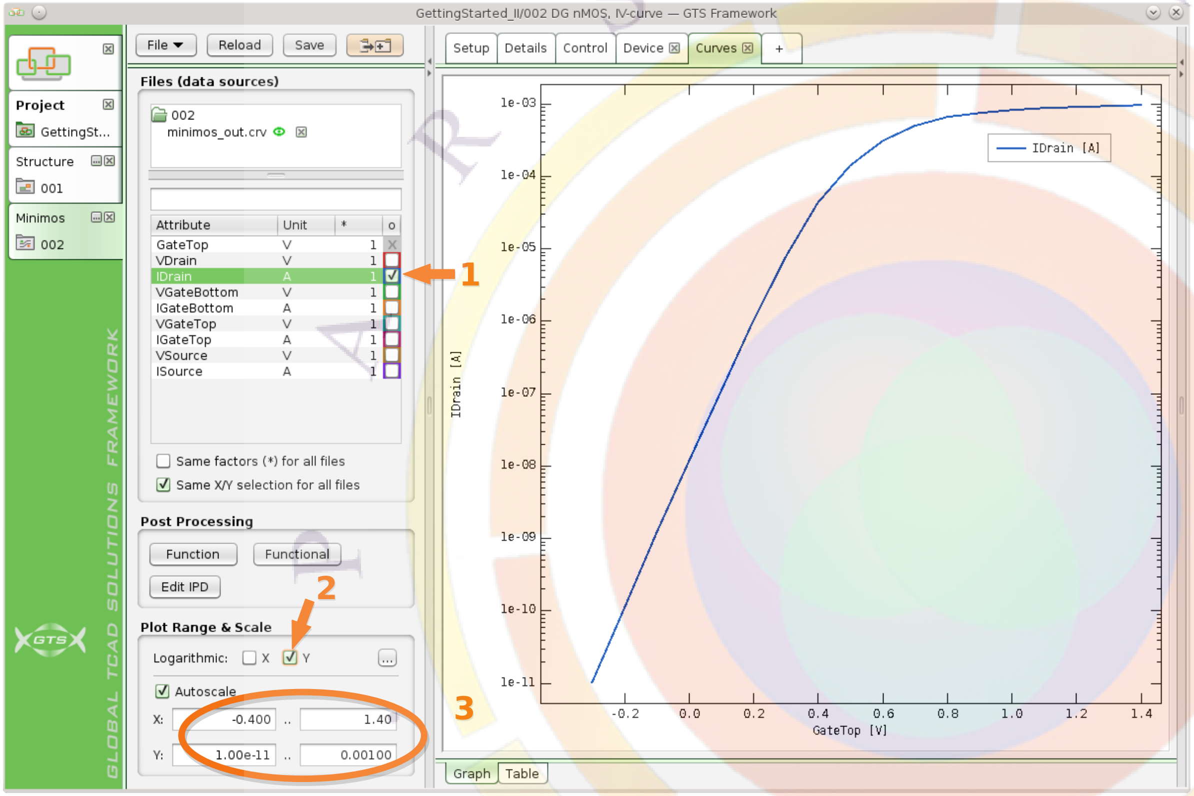
1.1.8. Simulation Setup 2

- Set up the Simulation Settings for Drain, GateBottom and GateTop as indicated in the figure.



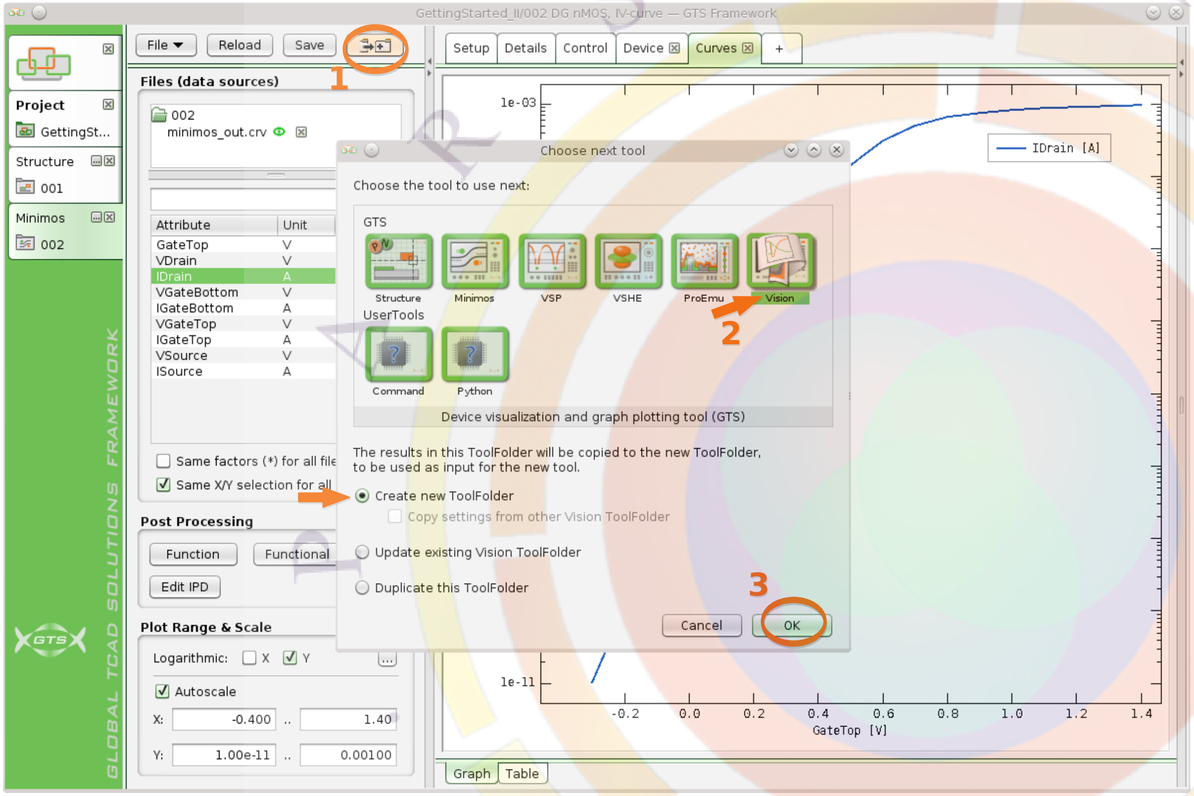
1.1.9. Run Simulation

- Press "Run"
- Press "Yes" to confirm saving and start the simulation



1.1.10. Device Characterics

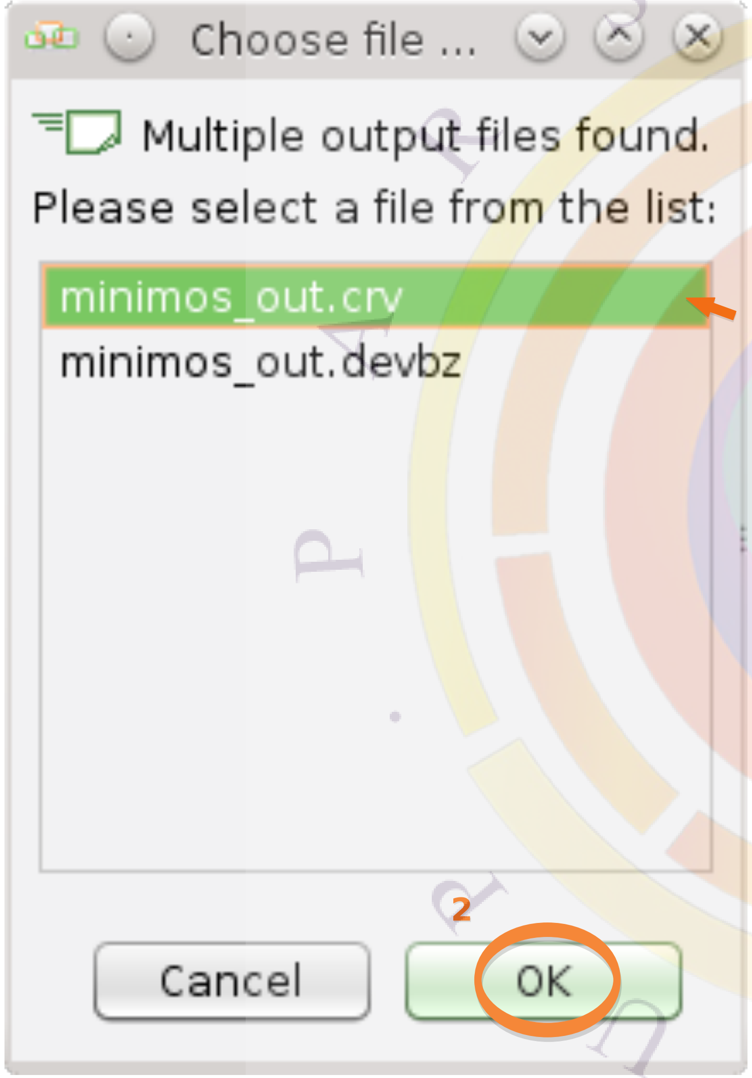
- Go to the "Curves" page
- Select "IDrain"
- Select Y-axis in logscale
- Show the scale of plot



1.2. Parameter Extraction

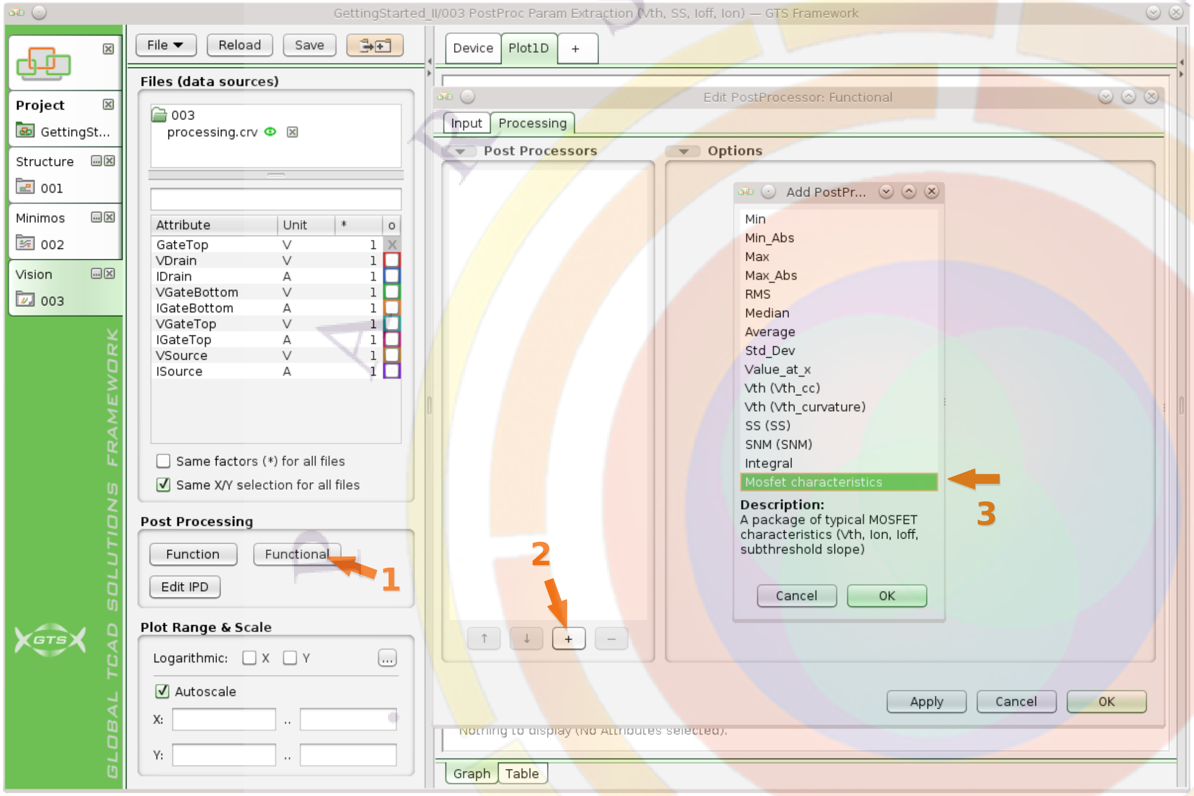
1.2.1. Create Vision

- Use the "Next Tool" button to create a new ToolFolder
- Select "Vision"
- Press "OK"



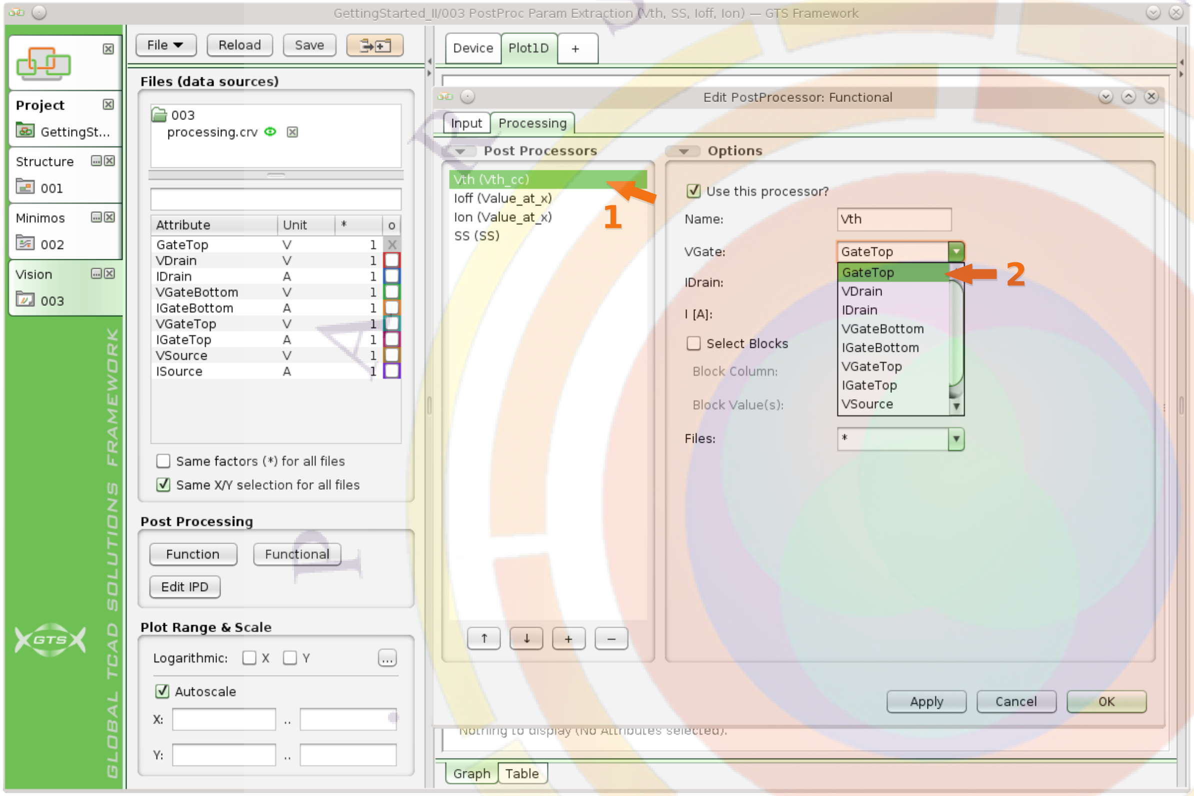
1.2.2. Select Output File

- For the file to transfer to the new ToolFolder, select `minimos_out.crv`
- Press "OK"



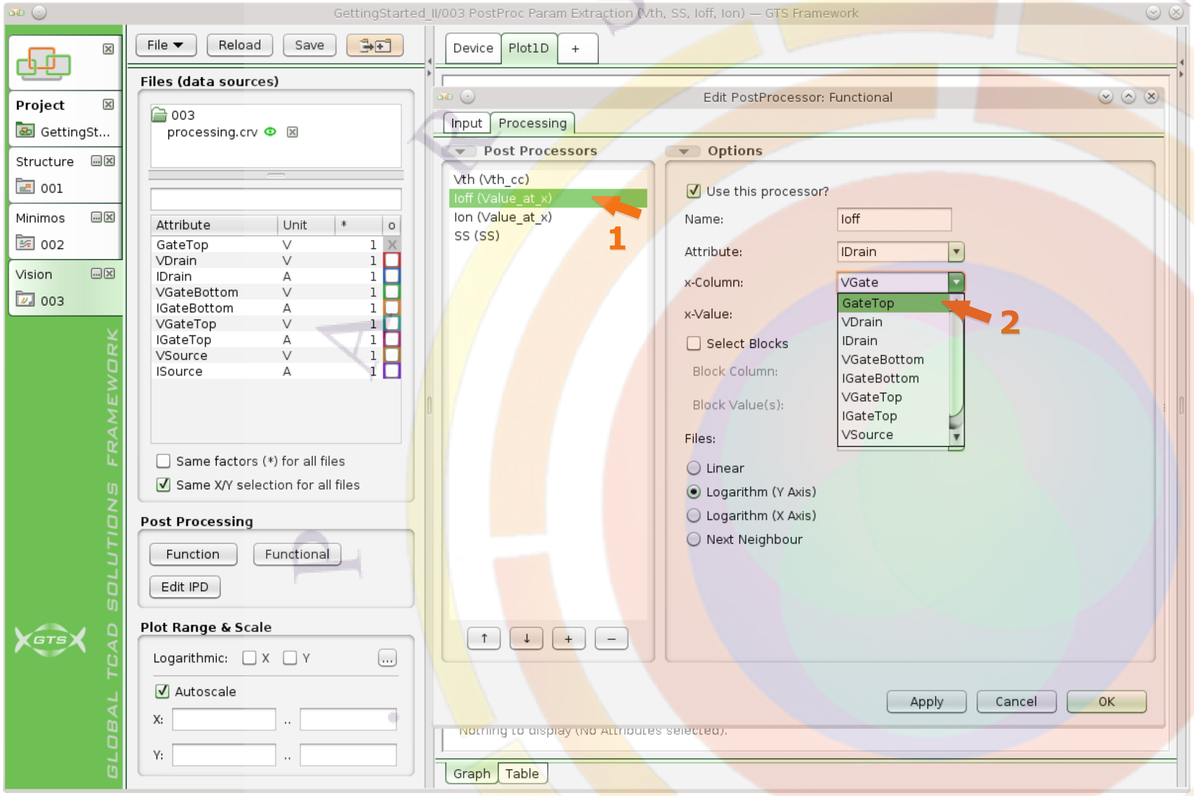
1.2.3. Definition Settings

- Click "Functional"
- Click "Add"
- Select "Mosfet characterics"



1.2.4. Settings - Vth

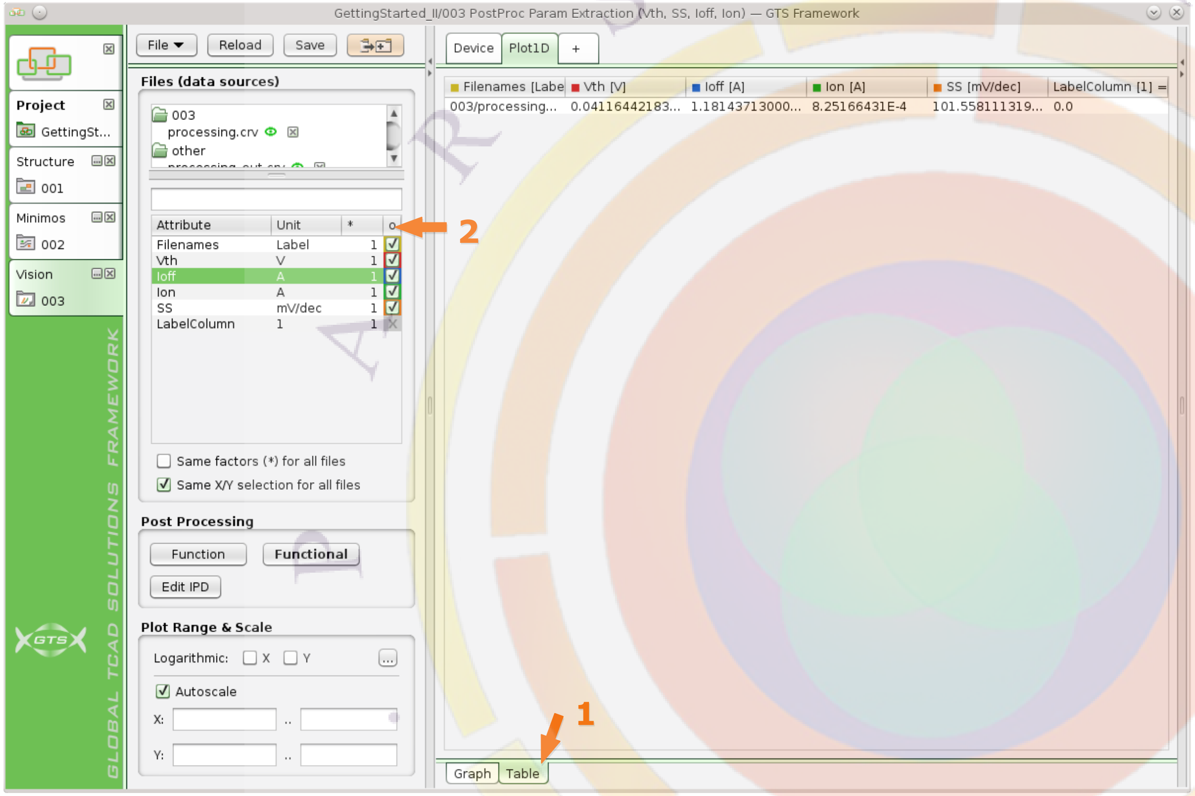
- Notice the four pre-defined postprocessors
- Choose the post-processor "Vth"
- Select GateTop for "VGate"



1.2.5. Settings - Ioff, Ion, SS

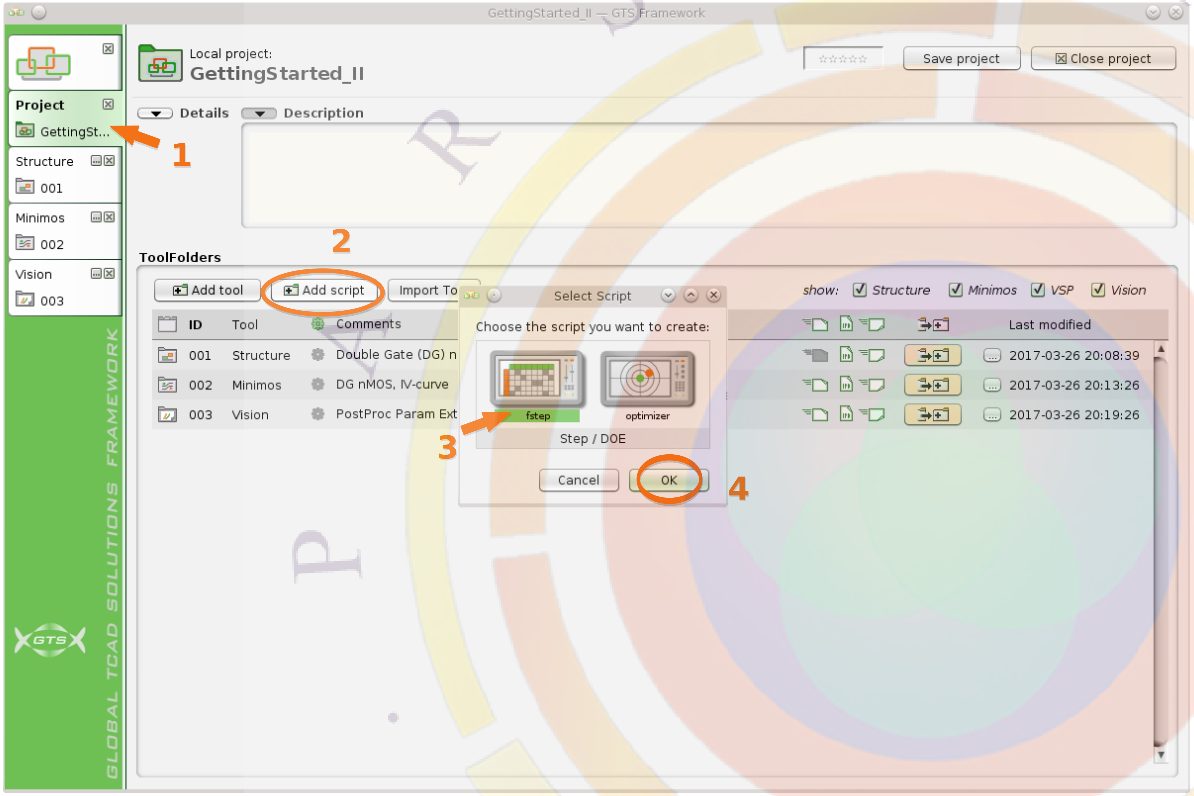
Similar to the previous steps:

- Choose the post-processor "Ioff"
- Select "GateTop" in "x-Column"
- Choose the post-processor "Ion"
- Select "GateTop" in "x-Column"
- Choose the post-processor "SS"
- Select "GateTop" in "VGate"



1.2.6. Parameters Extraction

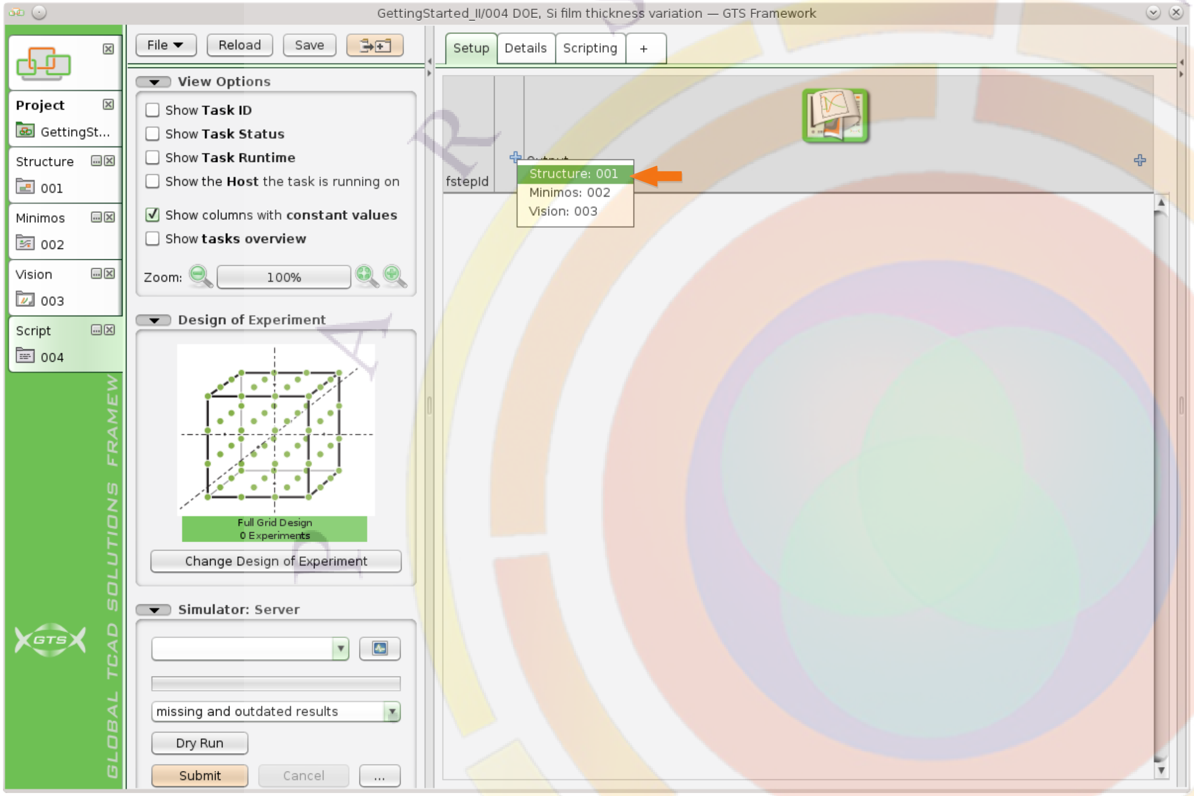
- In the "Plot1D" page, choose "Table"
- Quickly select all parameters by clicking on the table header
- All parameters are shown in the table



1.3. Design of Experiment Setup (DOE)

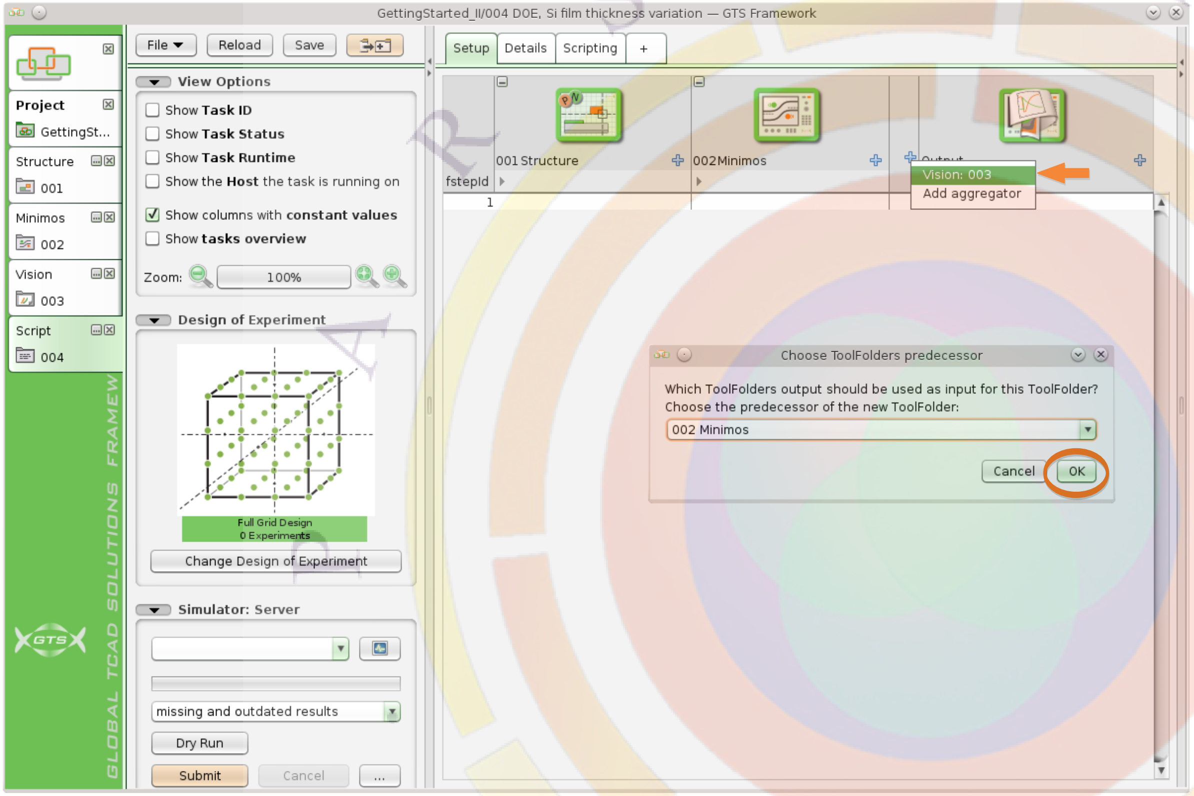
1.3.1. Create DOE Script

- Go to "Project"
- Click "Add script" to add a script
- Choose fstep
- "OK"



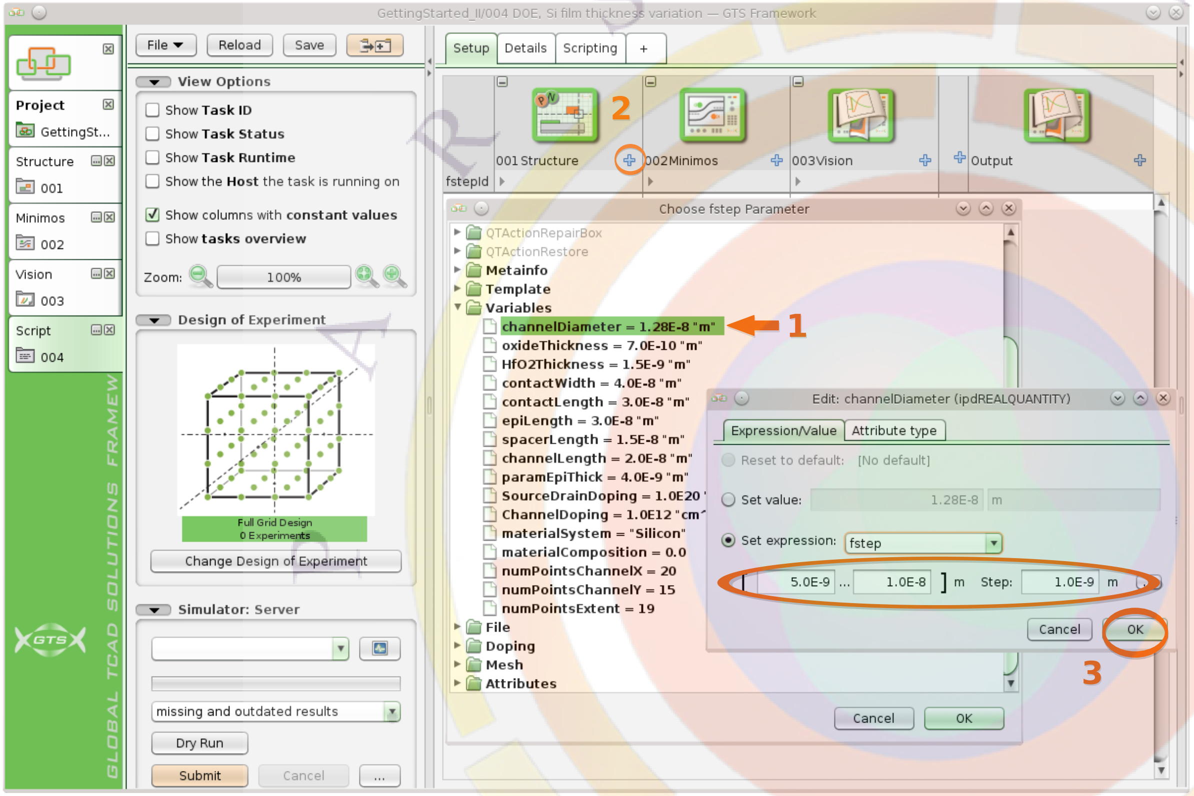
1.3.2. Add Tool 1

- Click "Add tool" button to add "Structure: 001"



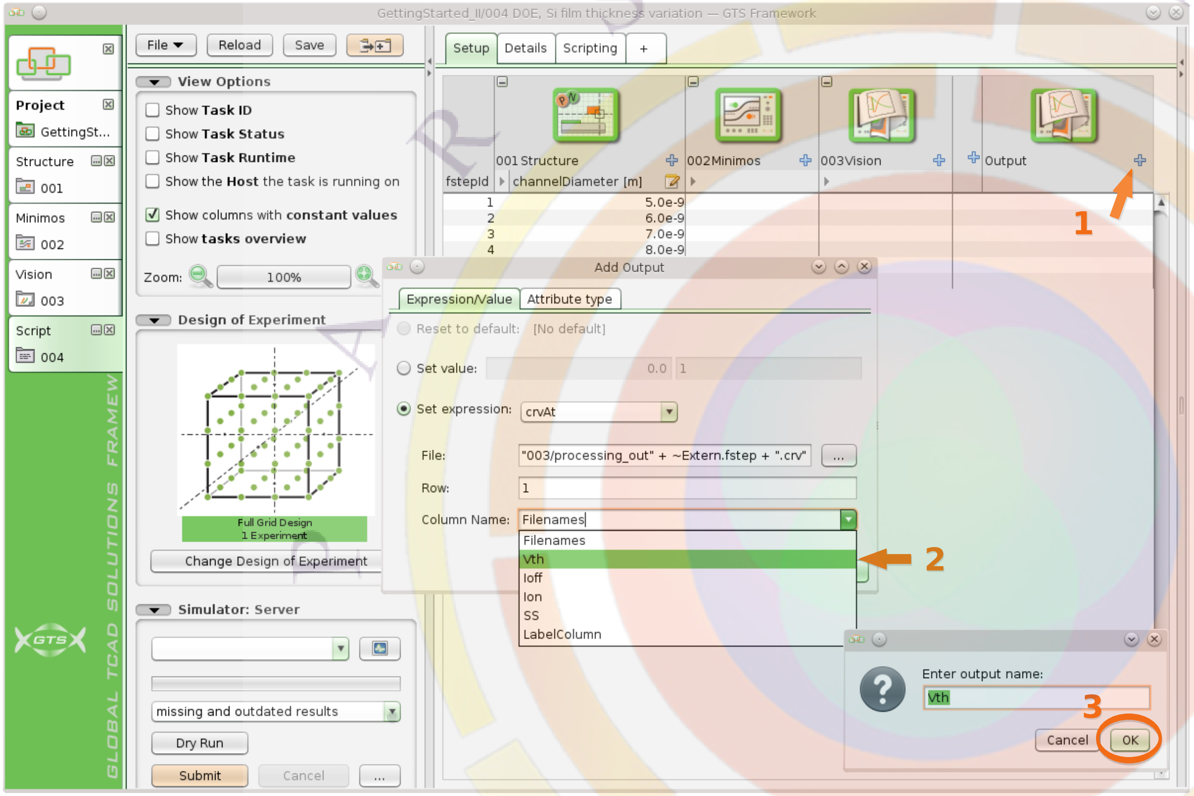
1.3.3. Add Tool 2

- Add "Minimos: 002" and "Vision: 003"
- Press "OK"



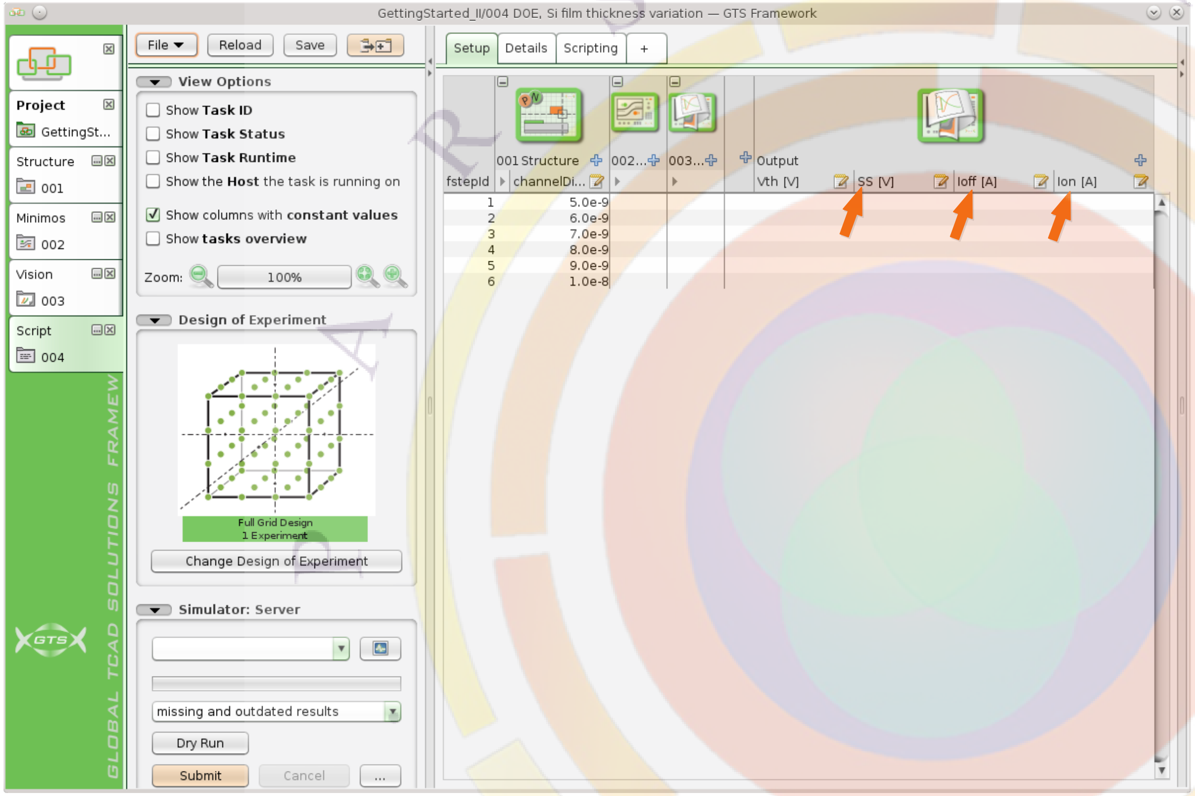
1.3.4. Edit Structure Folder

- Click "Add fstep parameter" in "001 Structure" to add parameters
- Double click "channelDiameter" in Variables to edit it
- Set up the settings as shown in the dialog.



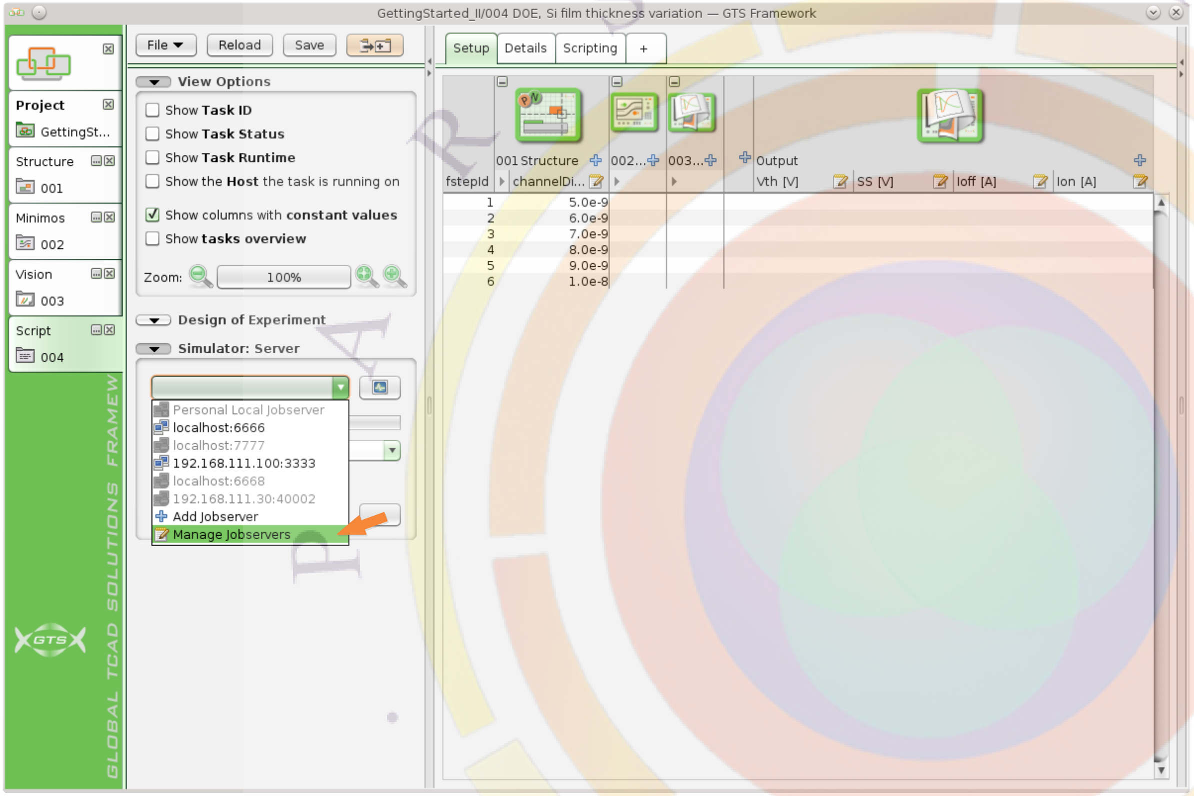
1.3.5. Add Output Parameters 1

- Click "Add output parameter" in "Output" to add parameters
- Add the "Vth" parameter as indicated in the figure



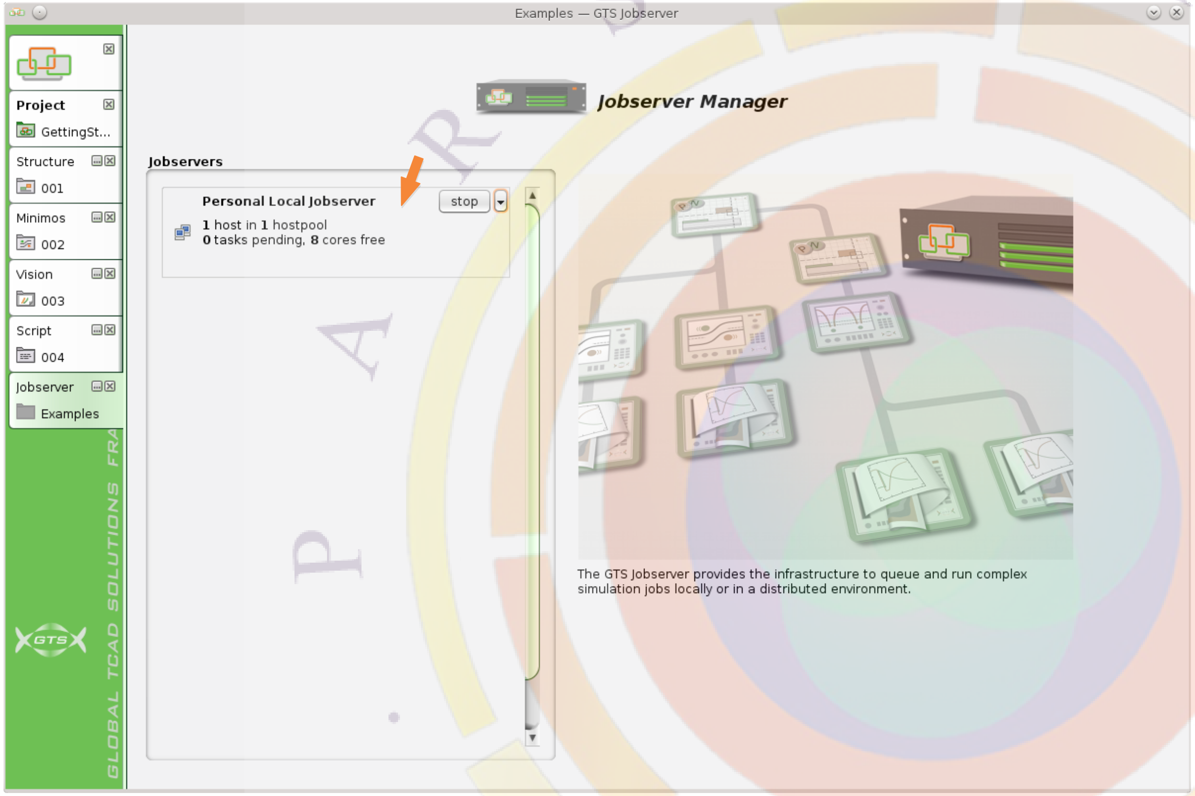
1.3.6. Add Output Parameters 2

- Repeat the previous steps to add SS, Ioff and Ion



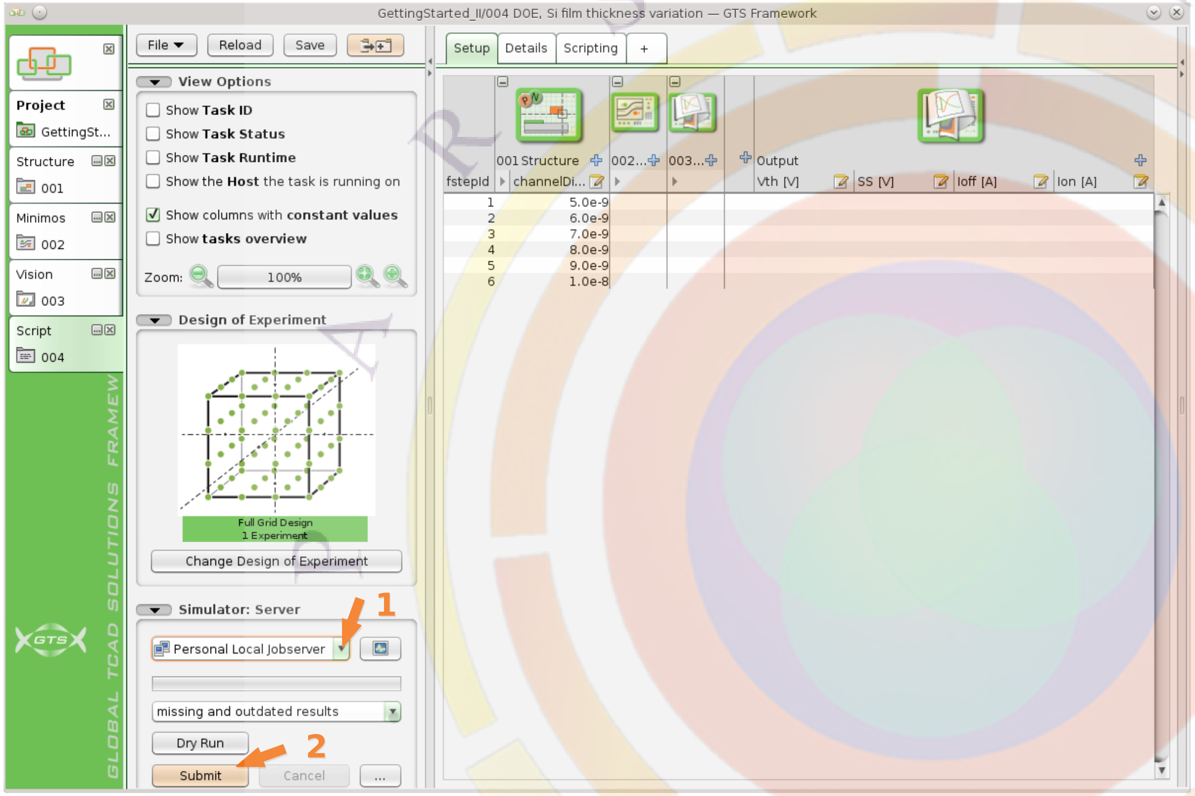
1.3.7. Jobserver 1

- Select "Manage Jobservers" as indicated in the figure



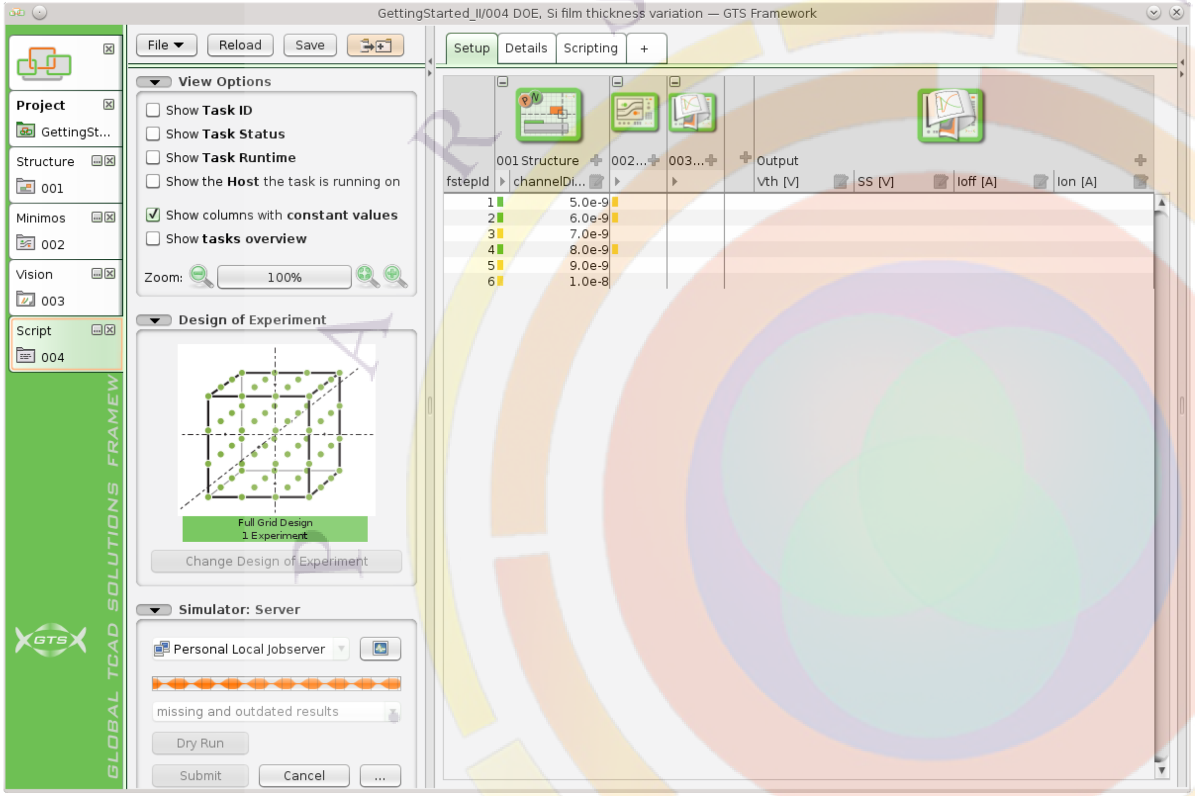
1.3.8. Jobserver 2

- Press the "start" button to run the local job server.



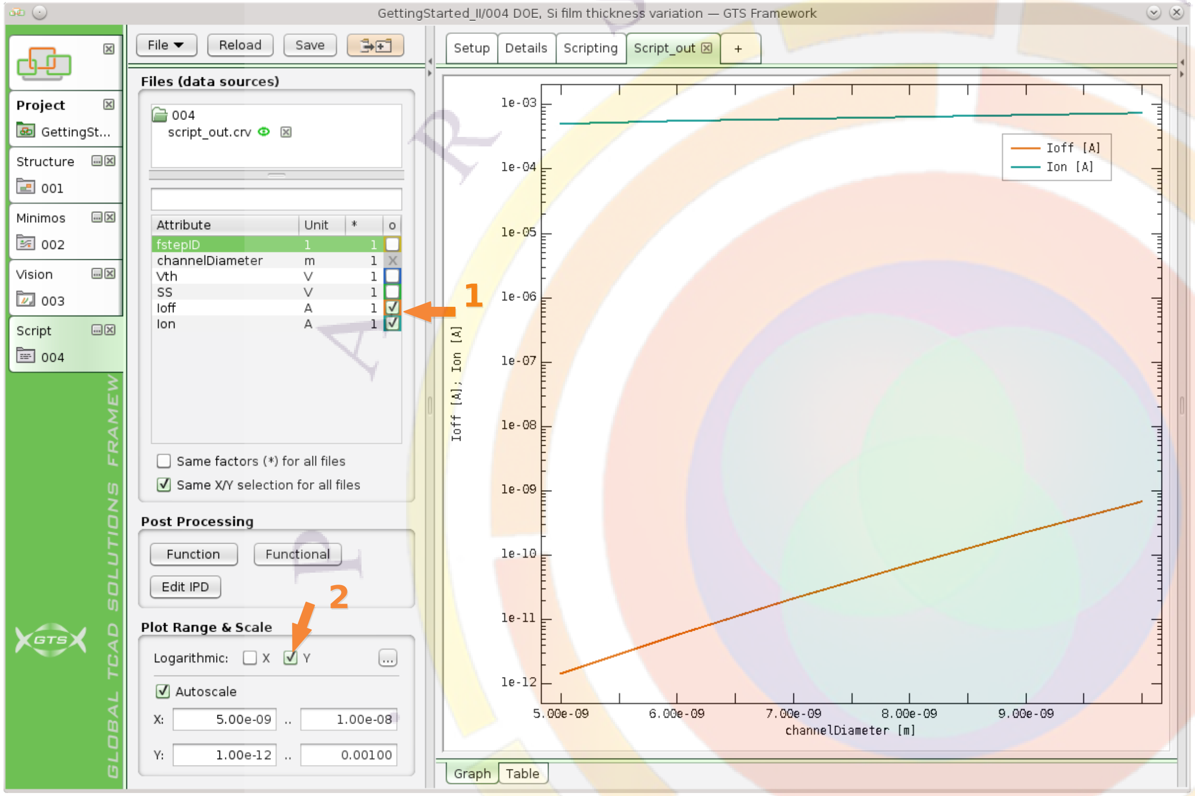
1.3.9. Submit Jobs

- Switch back to the Script ToolFolder
- Select "Personal Local Jobserver".
- Press "Submit"
- Press "Yes"



1.3.10. Simulation Status

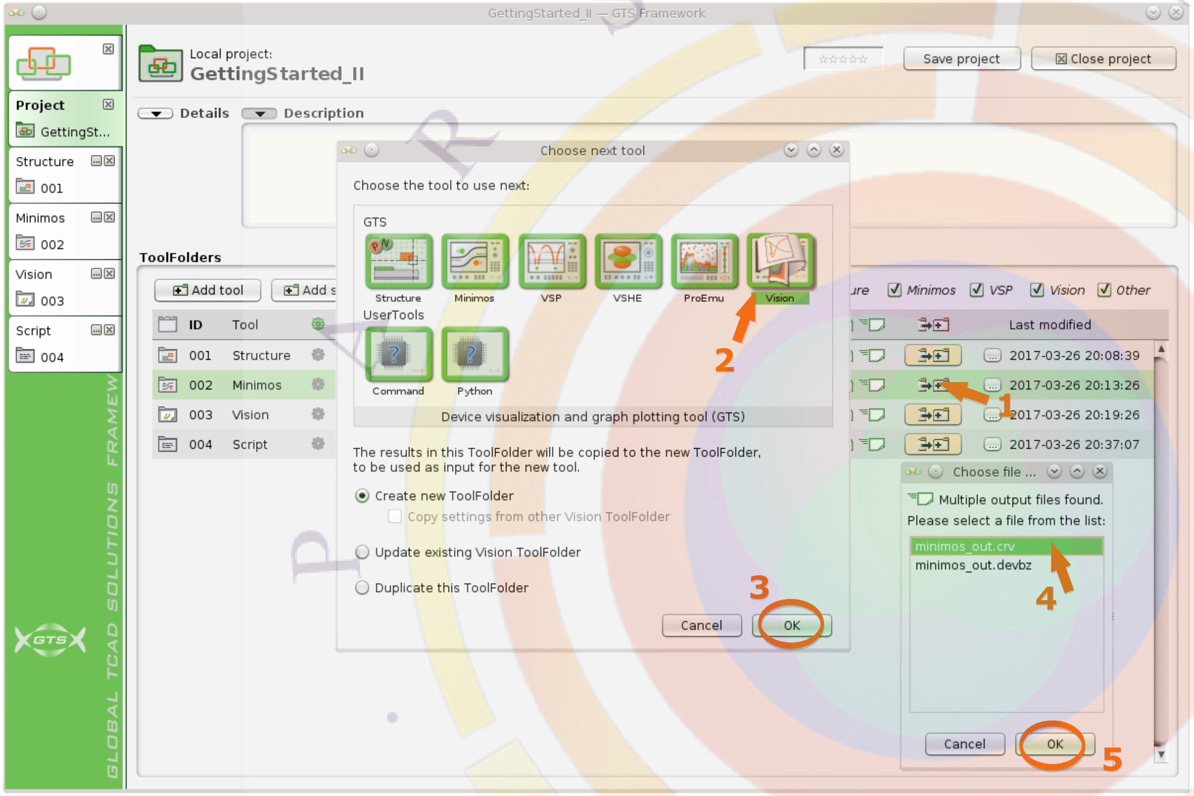
- Watch the status of the simulation tasks
- Wait for the simulation to finish



1.3.11. Simulation Result

- In the "Script_out" page, select Ioff and Ion
- Select logarithmic scale for Y

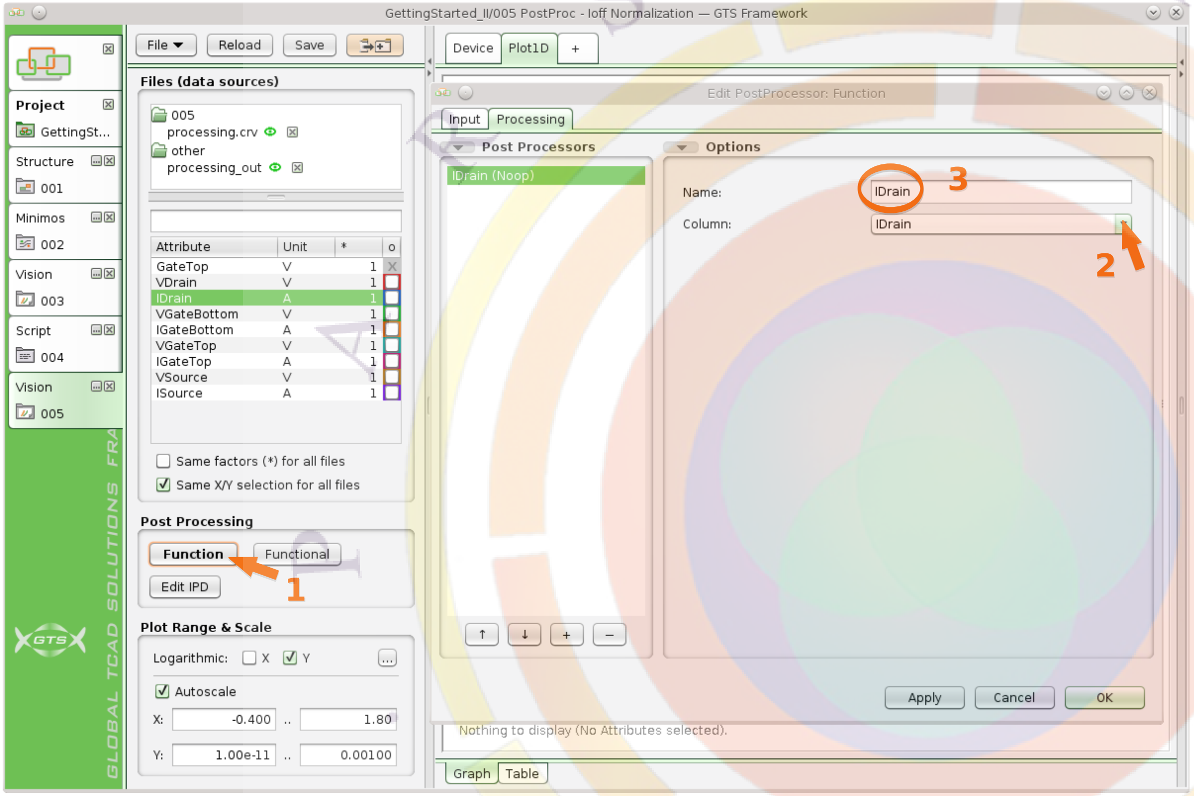
Examine the result. In the next steps, we will run the same experiment but with normalized IV curves to be able to better compare the embedded transistor performance.



1.4. Normalization

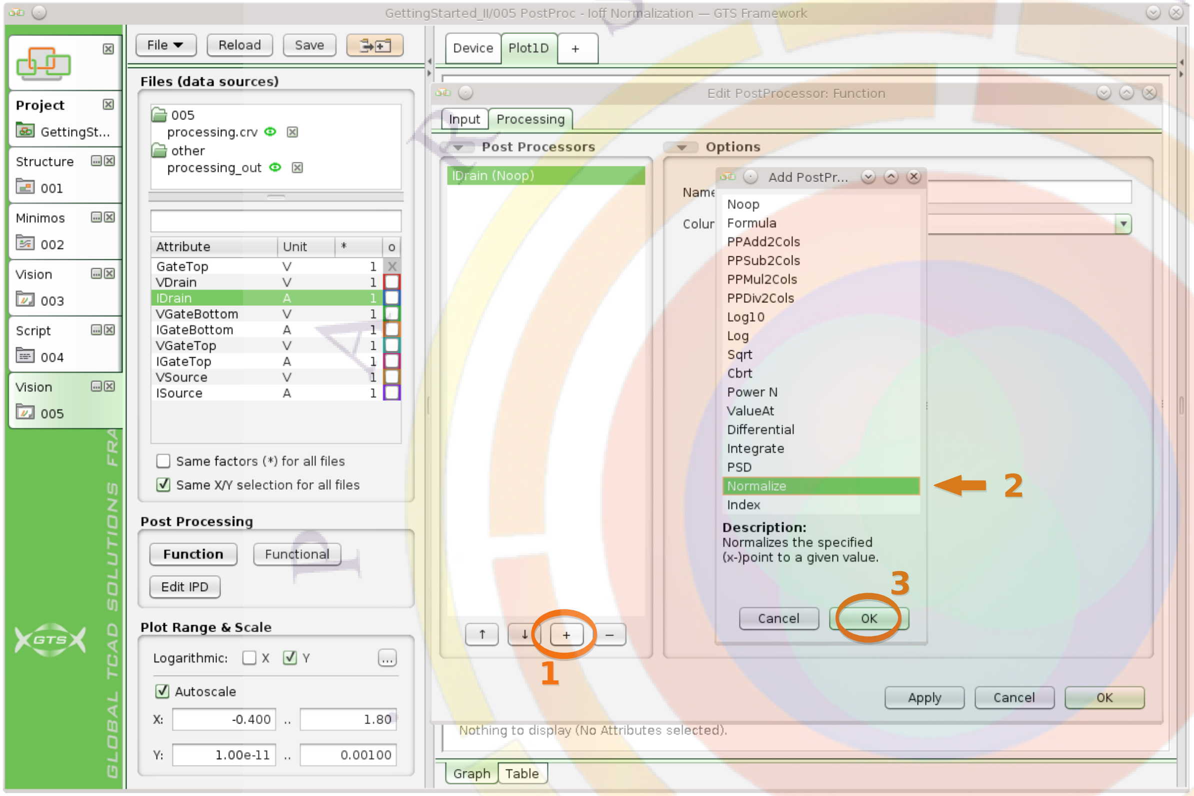
1.4.1. Postprocessing 1

- Go to Project
- Press the "Use output in the next tool" button in "002 Minimos"
- Press "OK"
- Select `minimos_out.crv` and press "OK"



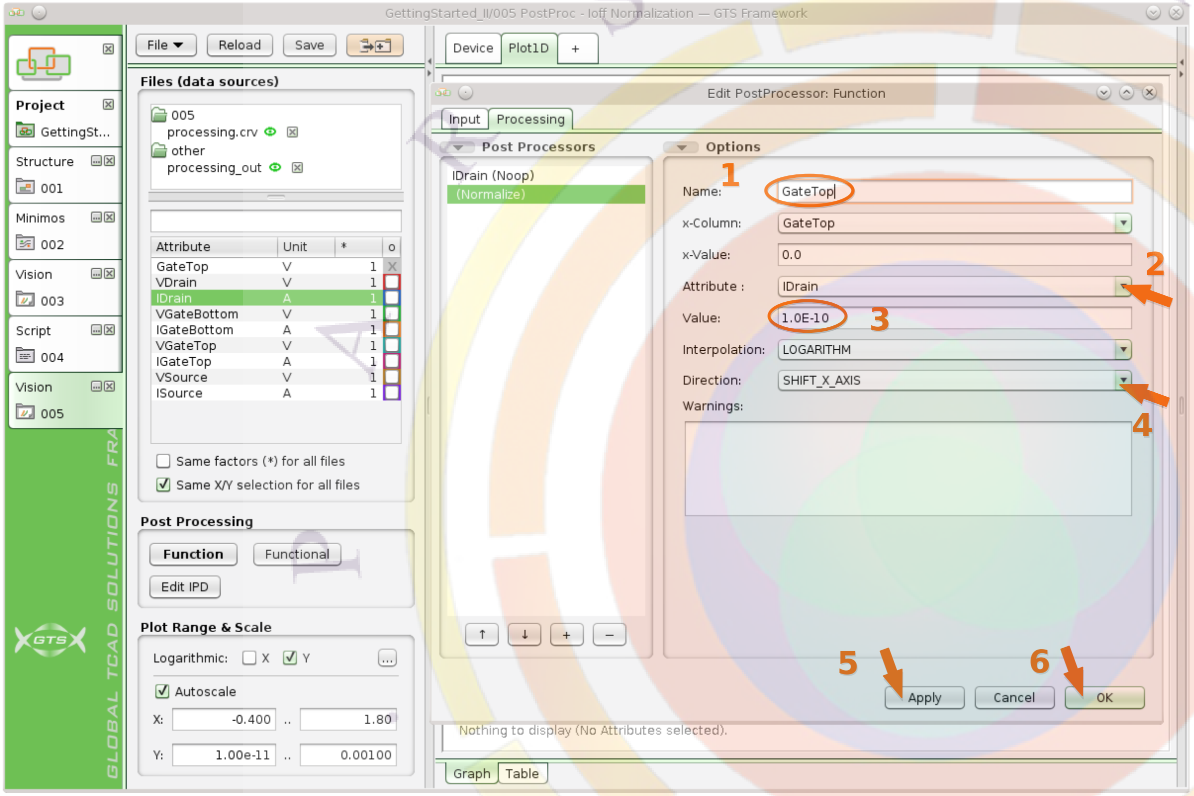
1.4.2. Postprocessing 2

- Select "Function"
- Select "IDrain"
- Rename to "IDrain"



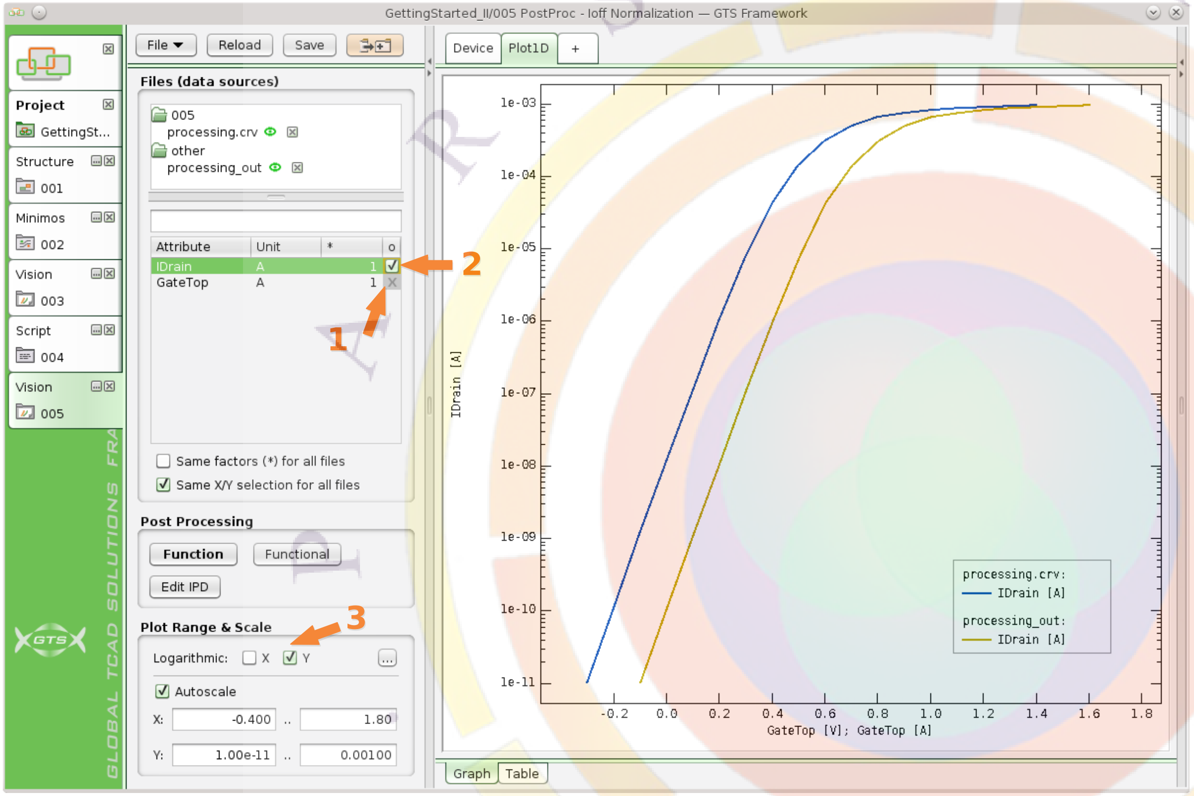
1.4.3. Postprocessing 3

- click "+" button and select "Normalize"
- Press "OK"



1.4.4. Postprocessing 4

- Write GateTop as a Name
- Select "IDrain"
- Change Value to "1.0E-10"
- Select "SHIFT_X_AXIS"
- Press "Apply" and "OK"



1.4.5. Results: Normalized IV Curve

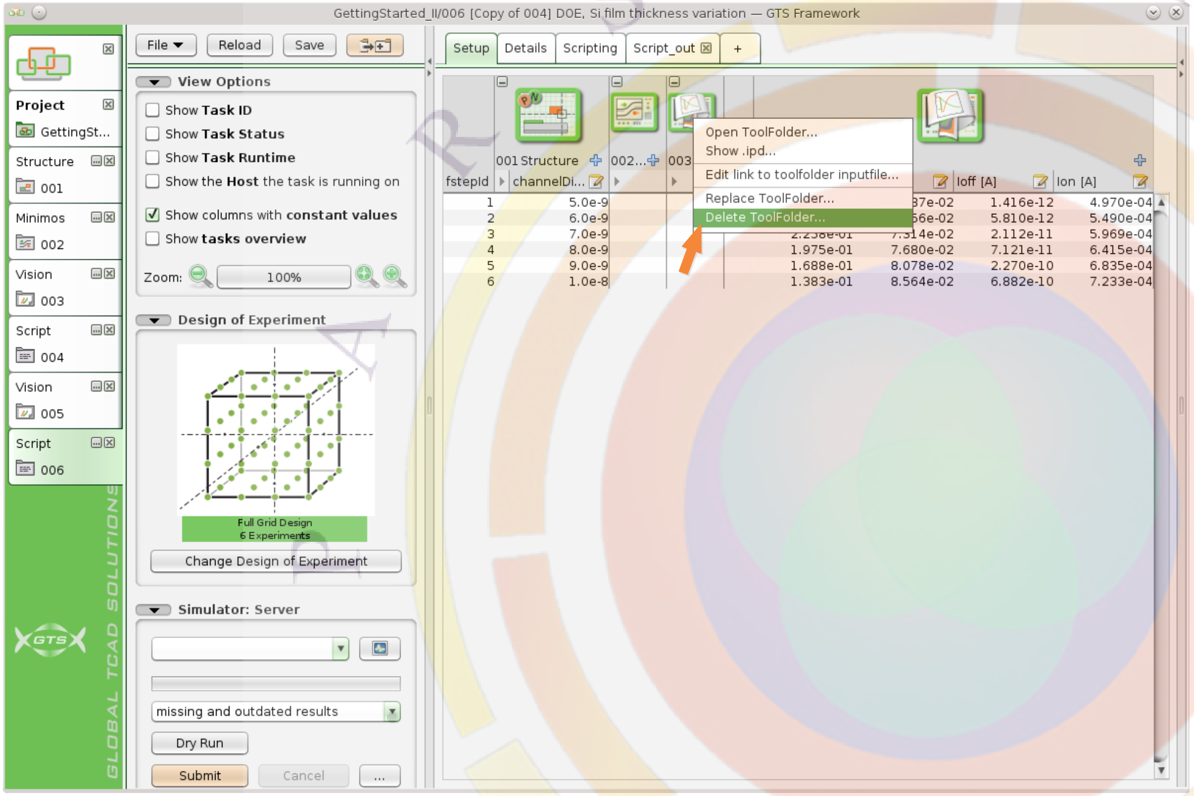
- Right click on "GateTop" and select "Set as X axis"
- Select "IDrain"
- Select logscale in Y axis
- Right click on "IDrain" to change its Line color to yellow as indicated in the figure

1.5. Design of Experiment

1.5.1. Copy DOE Script

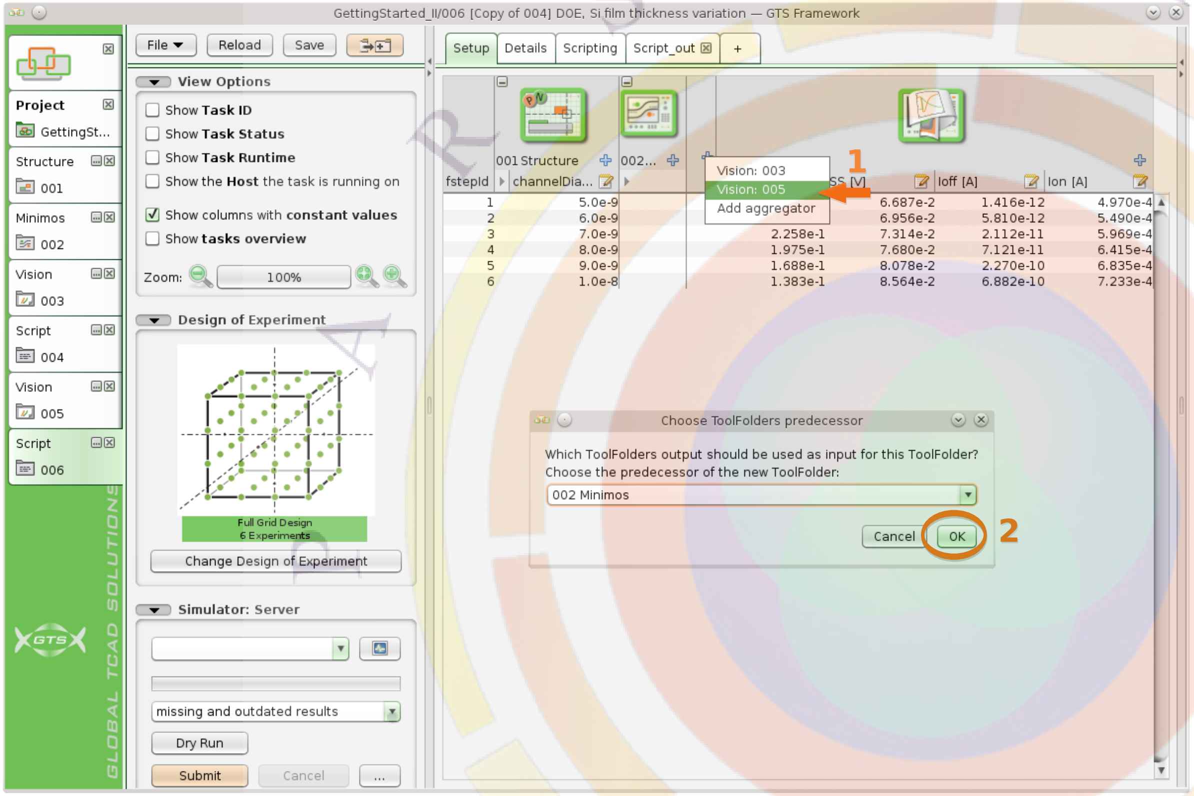
Based on the previous setup, we will now work on the normalized data:

- Go to Project
- Use button in 004 to duplicate the ToolFolder
- Press "OK"



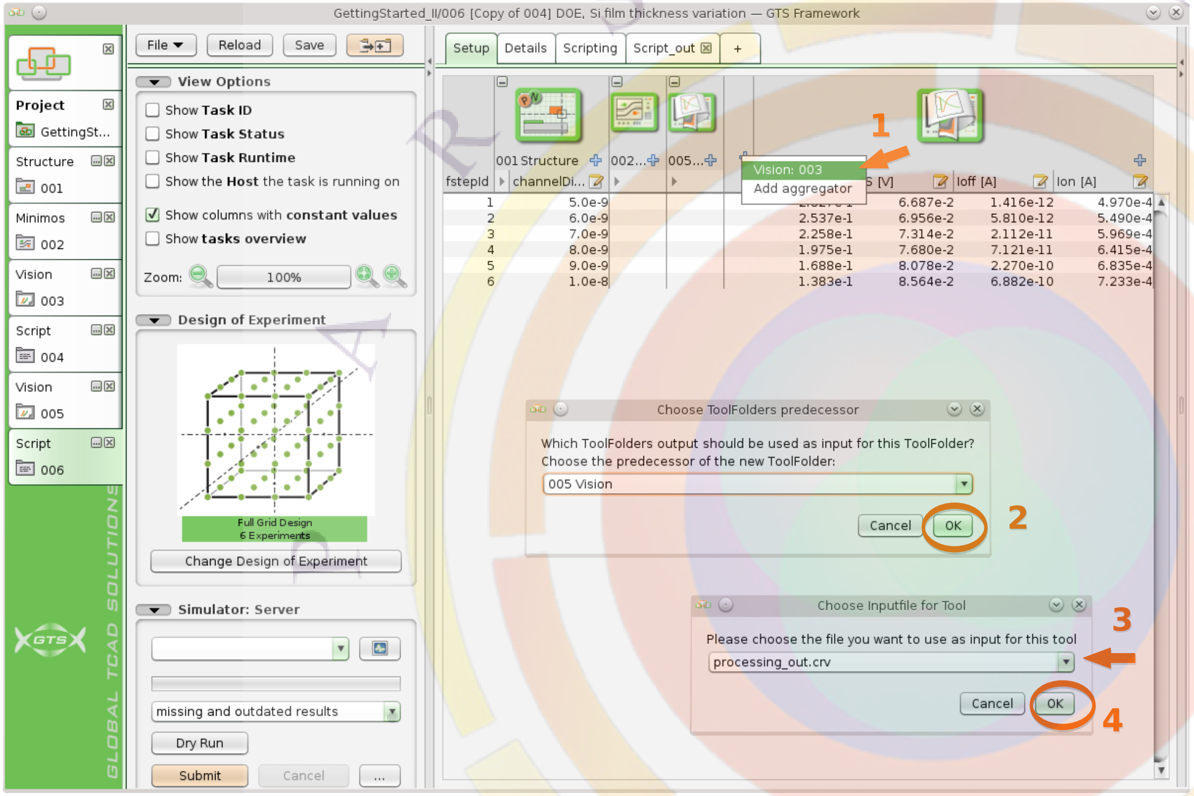
1.5.2. Edit Script 1

- Right-click on 003 and select "Delete ToolFolder"
- Confirm with "Yes"



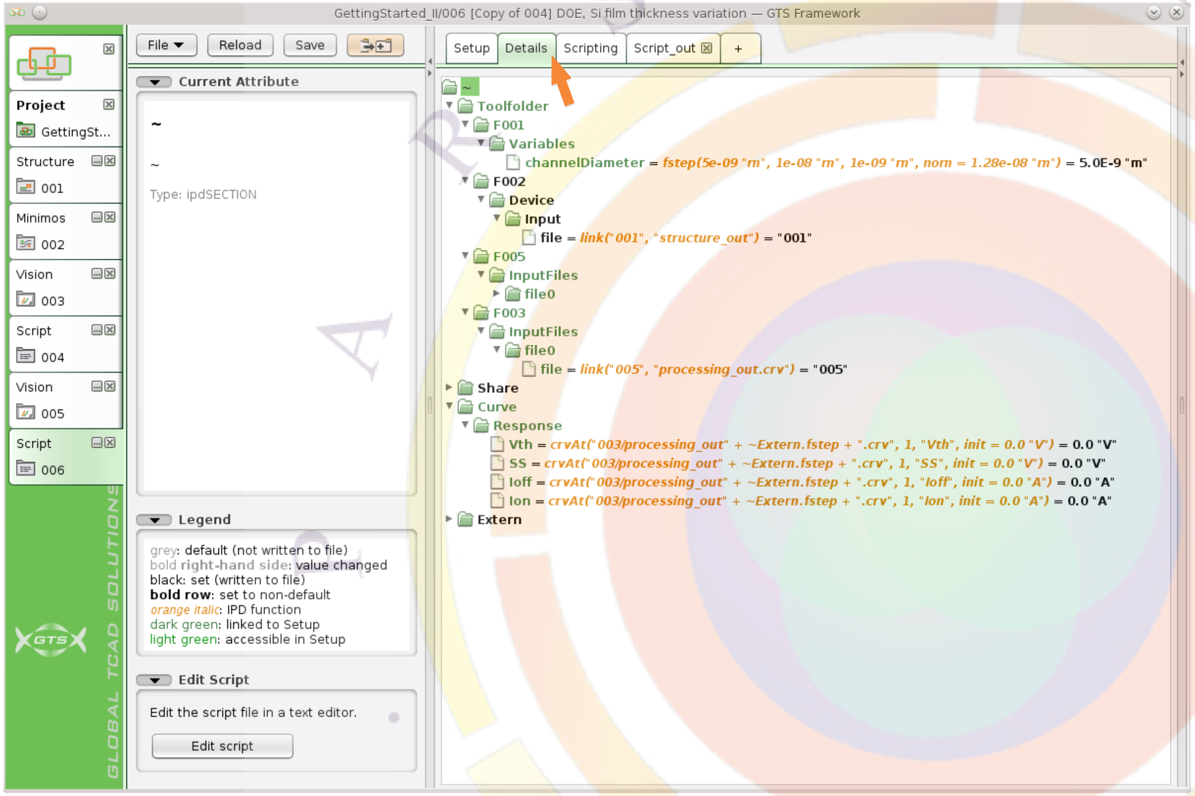
1.5.3. Edit Script 2

- Click the plus ("Add tool") to add "Vision: 005"
- Press "OK"



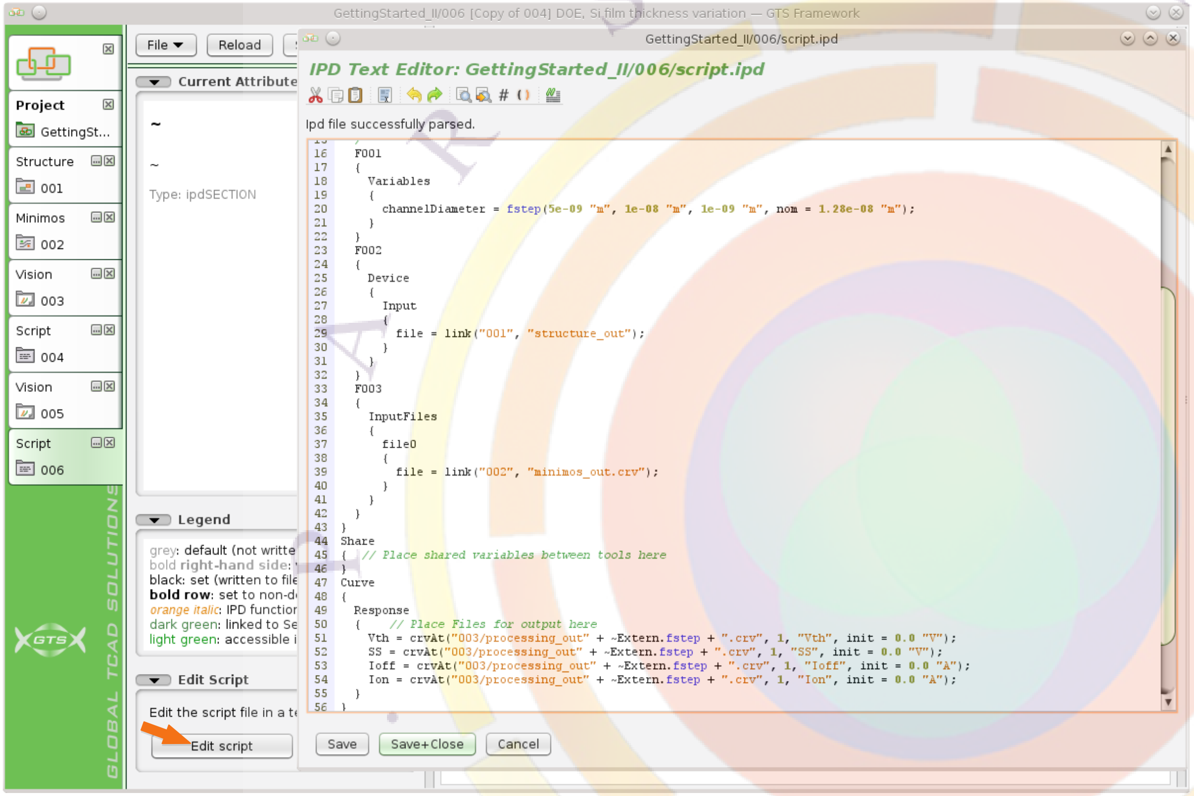
1.5.4. Edit Script 3

- Click "Add tool" to add "Vision: 003"
- Press "OK"
- Choose `processing_out.crv` and press "OK"



1.5.5. Check Setup

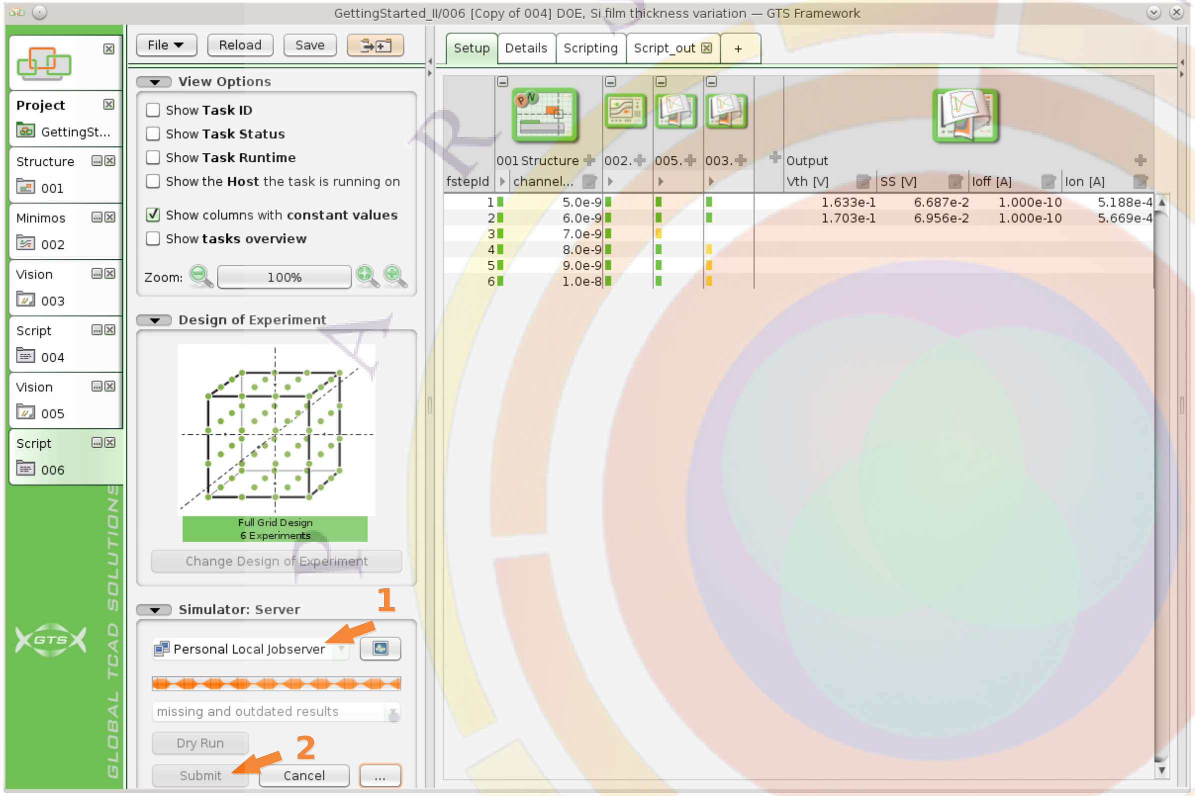
- Click Save to save the DOE Script
- Go to the "Details" page to check the flow



1.5.6. Check Setup 2

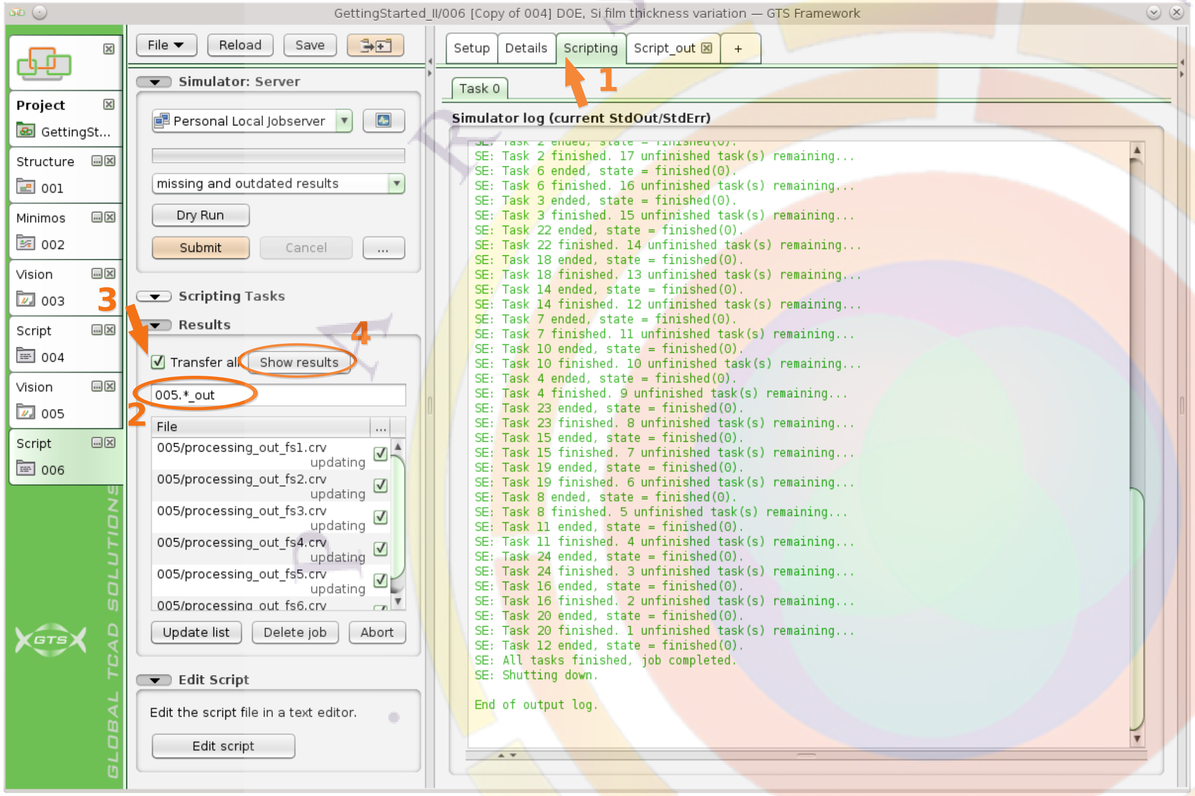
You can also take a look into the IPD file that was created:

- Go to "Scripting" page
- Open "Edit script" to check IPD file
- Press "Save+Close"



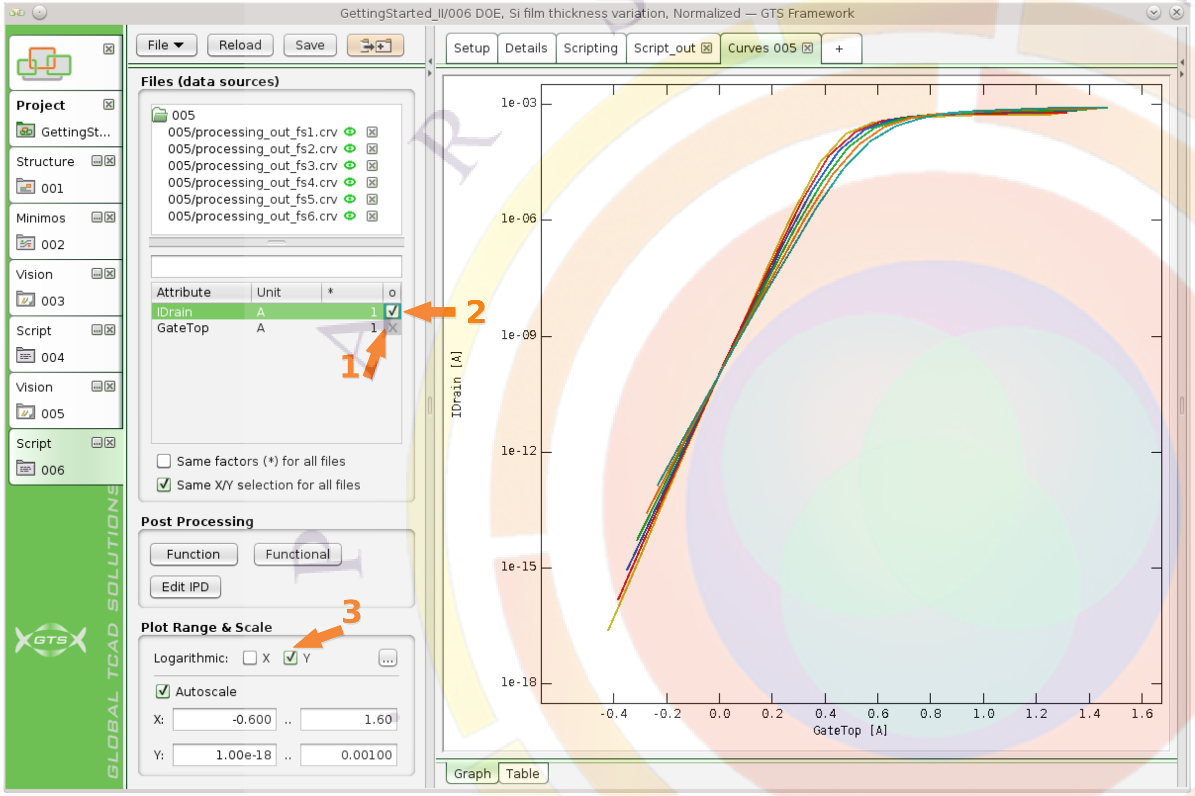
1.5.7. Run Simulation

- Go to the "Setup" page
- Select "Personal Local Jobserver"
- Press "Submit"



1.5.8. Check Results

- Go to the "Scripting" page
- Find 005.*_out
- Check checkbox of "Transfer all" to get all files
- Press "Show results"



1.5.9. Create Curve

1. Right click on "GateTop" and selct "Set as X axis"
2. Select "IDrain"
3. Select logscale in Y axis
4. Right click on "IDrain" and select "Line color" in "Vary over all files" to change colors as indicated in the figure

Conclusion

In this tutorial, we used the Design of Experiment workspace of GTS Framework. A workflow with structure generation, device simulation and parameter extraction was shown, including Off-current normalization.

Further Reading

We welcome you to have a look at further *GTS tutorials* and *examples*, which you can open in ***GTS Framework***. Next to the basic ones included with the release, you can download more sophisticated tutorials and examples from MyGTS at `https://globaltcad.com/mygts`. Extracting the archives to your projects folder makes the tutorials visible in the projects list (highlighted yellow). Previews are provided at `http://www.globaltcad.com/en/solutions.html`.

For additional information, please refer to `http://globaltcad.com/` or feel free to contact us at `info@globaltcad.com`.

Appendix A

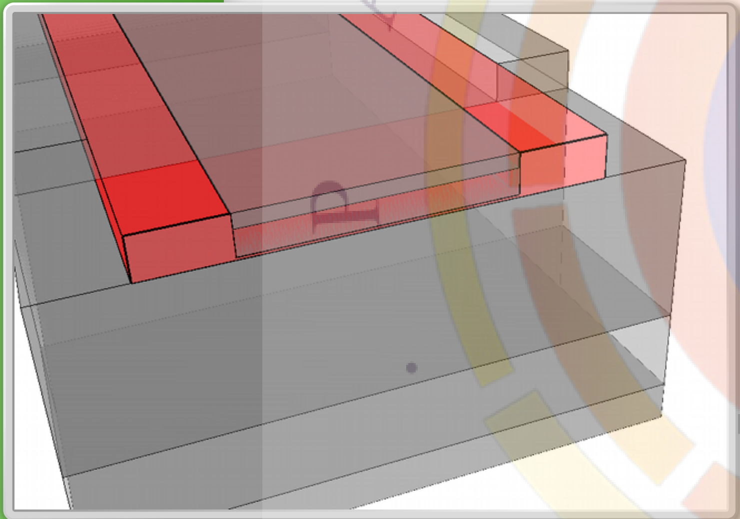
ToolFolder List

The project **GettingStarted_II** contains the following ToolFolders (TF):

TF	Tool	Description
T01	Structure	Double Gate (DG) nMOS
T02	Minimos	DG nMOS, IV-curve
T03	Minimos	PostProc Param Extraction (Vth, SS, Ioff, Ion)
T04	Minimos	DOE, Si film thickness variation
T05	Minimos	PostProc - Ioff Normalization
T06	Minimos	DOE, Si film thickness variation, Normalized

GLOBAL TCAD SOLUTIONS — GTS FRAMEWORK — TUTORIAL

GTS FRAMEWORK



Device Editor

Tutorial

global TCAD solutions

Device Editor – Tutorial, GTS Framework Release 2016.09
Revision of September 15, 2017

Global TCAD Solutions GmbH
Bösendorferstraße 1/12
1010 Wien
Austria / Europe

Phone: +43 1 9255049
Email: info@globaltcad.com

Additional information is provided at: http://www.globaltcad.com/

Cover & layout: prausedesign

Contents

Introduction

This tutorial introduces the device editor integratetd in ***GTS Framework***. It illustrates editing device doping profiles as well as 2D and 3D device structures.

Using This Tutorial

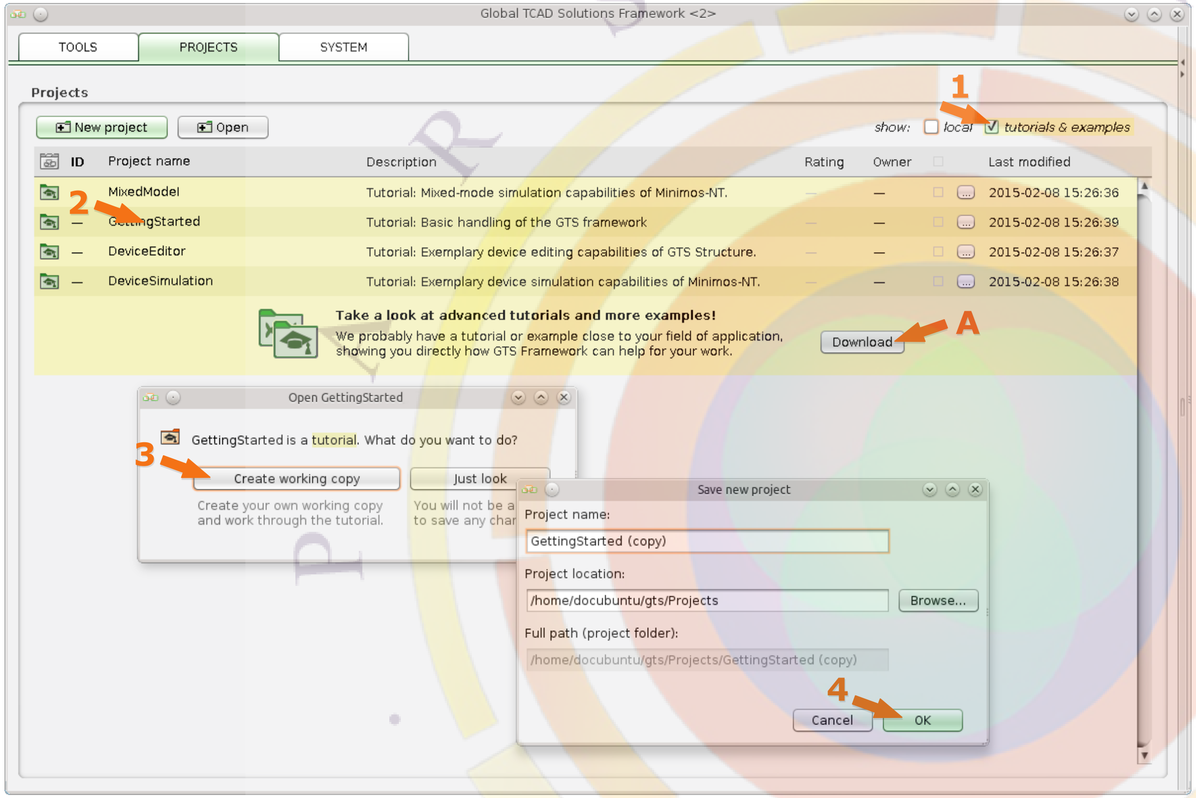
GTS *tutorials* generally consist of a written description (this text) and a GTS *project* (simulation data) which you can open in ***GTS Framework*** (yellow items in project list). As shown below, we recommend to create a working copy from the project, and proceed step by step, guided by this text. If not yet familiar, please refer to the *GettingStarted* tutorial.

Working in a copy of a tutorial project, you can open the yellow Txx ToolFolders at any time. They contain the data of the described simulations, which you can use for reference or as starting points for your own simulations.

Only a few tutorials are included with the release; the others are available via *MyGTS* at `https://globaltcad.com/mygts`. When logged in, please download the respective file and extract it to the gts projects folder on your PC. After restarting GTS Framework, the tutorial project will appear in the *Projects* list.

This is an introductory (level 1) tutorial.

This tutorial was created using GTS Framework Release 2016.09. Other releases might need adjustments or have slightly different user interfaces.



Project / Working Copy

If you have not already done so, go to the *Framework Home* and locate the project associated with this tutorial in the *Projects* list, and create your working copy from it:

1. Check “tutorials & examples”
2. Click on the respective tutorial project: ***Device Editor***
3. Choose “Create working copy”
4. Check the project name, click “OK”

The project is created and opened, so that you see the *Project Home*. — *Ready to start!*

A. If you miss the respective project, please download it via *MyGTS* (click “Download”, see previous page).

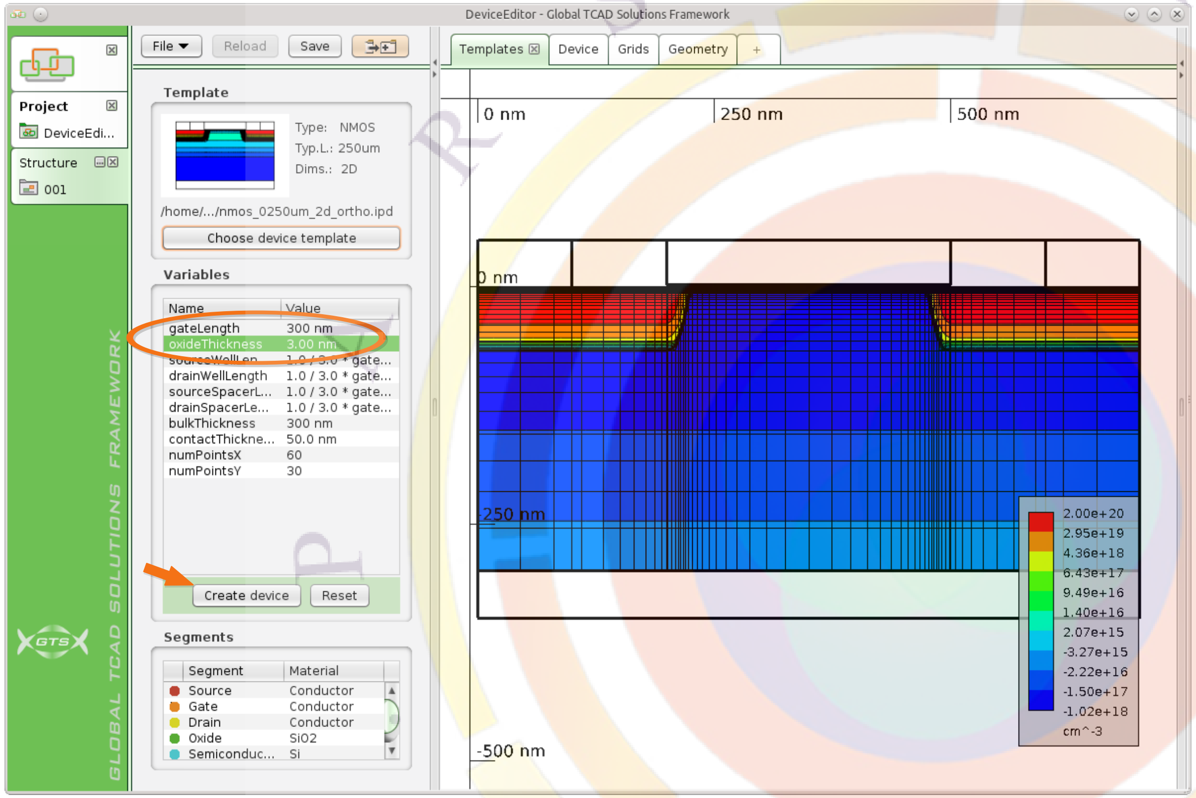
Alternatively, you can start the tutorial with an empty project – just create a new project. (The advantage of the tutorial project is that you have the results at hand any time.)

Part 1

Editing Device Doping Profiles

In this example, we will edit the doping profile of a two-dimensional device. Changing doping profiles will be demonstrated by introducing a threshold voltage doping to a nMOS template. As we want to compare the difference, induced by the doping, we will start with simulating the transfer characteristic of the original nMOS template.

It is assumed that you are already familiar with GTS Framework and the idea of projects and ToolFolders, as explained in the Getting Started tutorial.



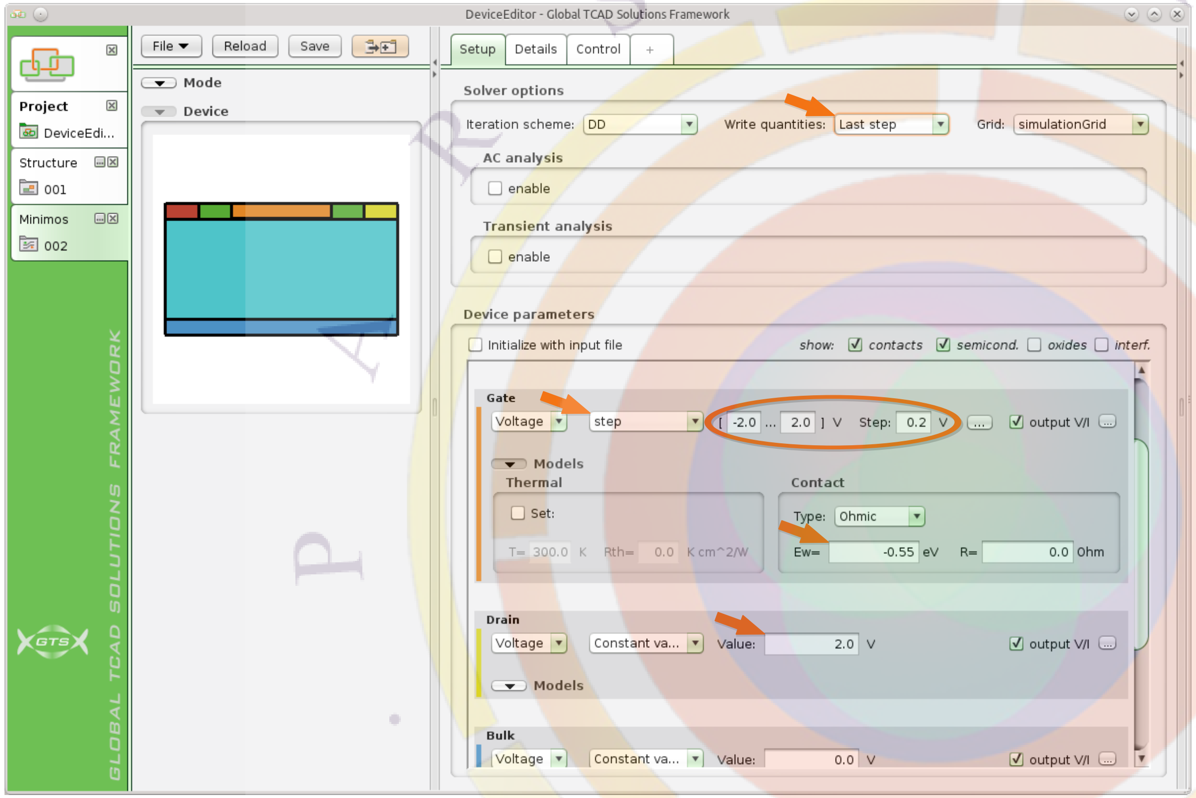
1.1. Use Device Template

1.1.1. New Structure

- In your project, add a “Structure” ToolFolder using the “Add tool” button
- We want to use a GTS-supplied device template. Click “File » New” and choose “Create device from template”
- Choose the template “`nmos_0250um_2d_ortho.ipd`”
- Set the following parameters of the template: `gateLength = 300 nm` `oxideThickness = 3 nm` as shown in the figure
- Click “Create device”

For information on projects, please refer to the Introduction (page 1).

After creating an instance of the device, the transistor will be displayed.

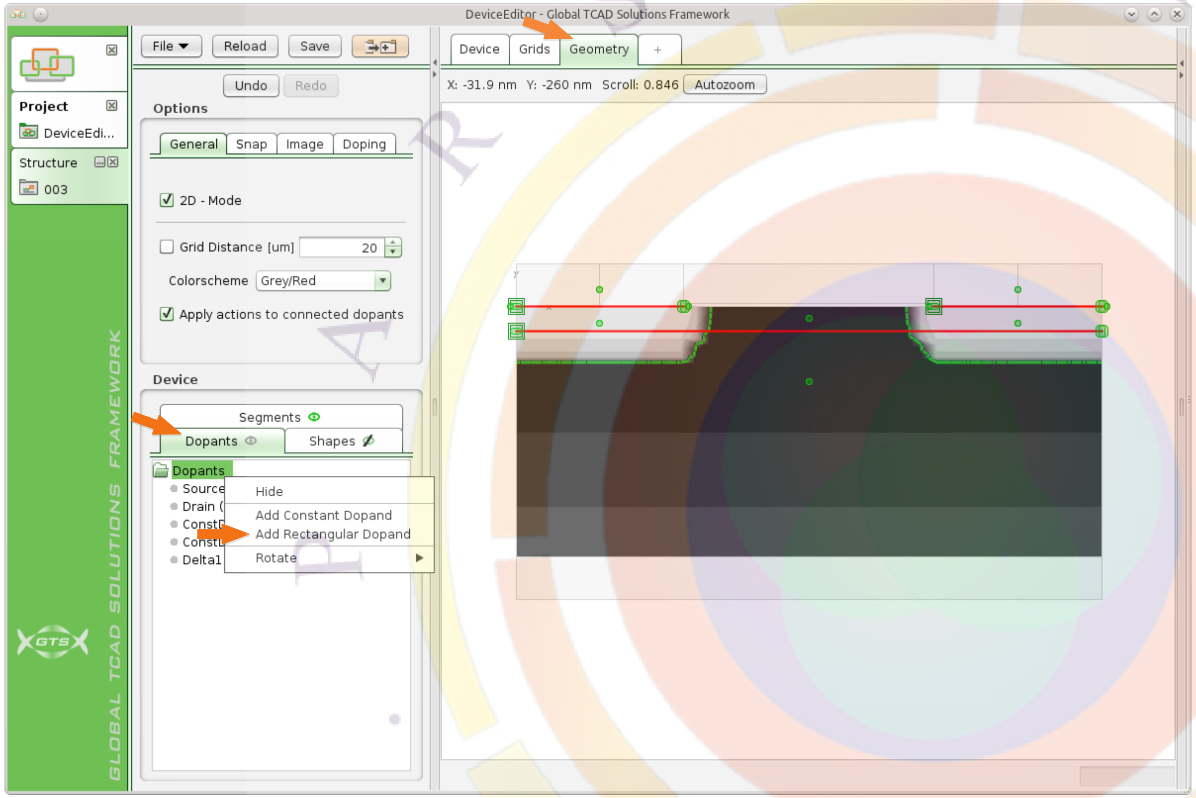


1.1.2. Transfer Characteristic

- Use the “next tool” button to create a Minimos-NT ToolFolder based on the previously generated nMOS
- Change the settings as shown in the figure
- Run the simulation in the “Control” page

The setting “`Write Quantities: Last Step`” is made to investigate quantities in on-state.

How to extract the exact threshold current is shown in the “Commandline” tutorial.



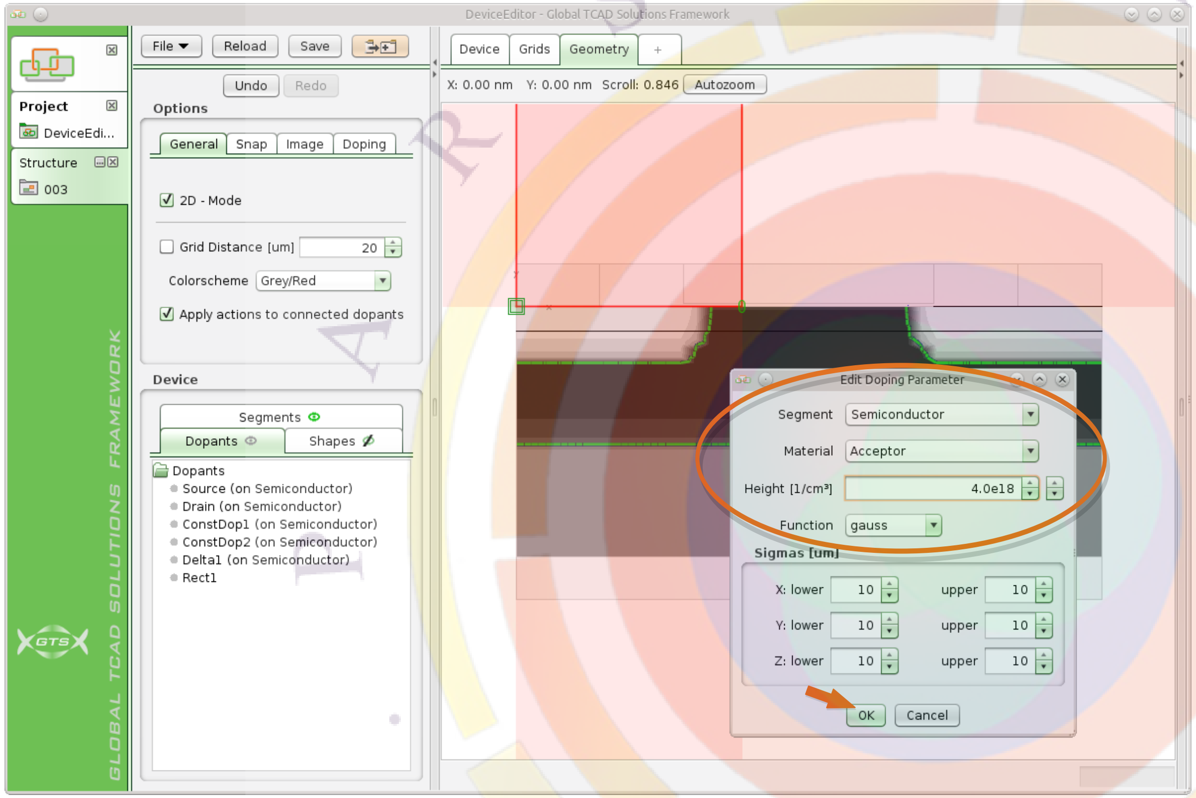
1.2. Vth Implant

1.2.1. Add Doping

Now we are going to edit the structure. As we want to analyse the effect of our changes, we will duplicate the previously generated structure tool. Changes will only be made to the new structure tool:

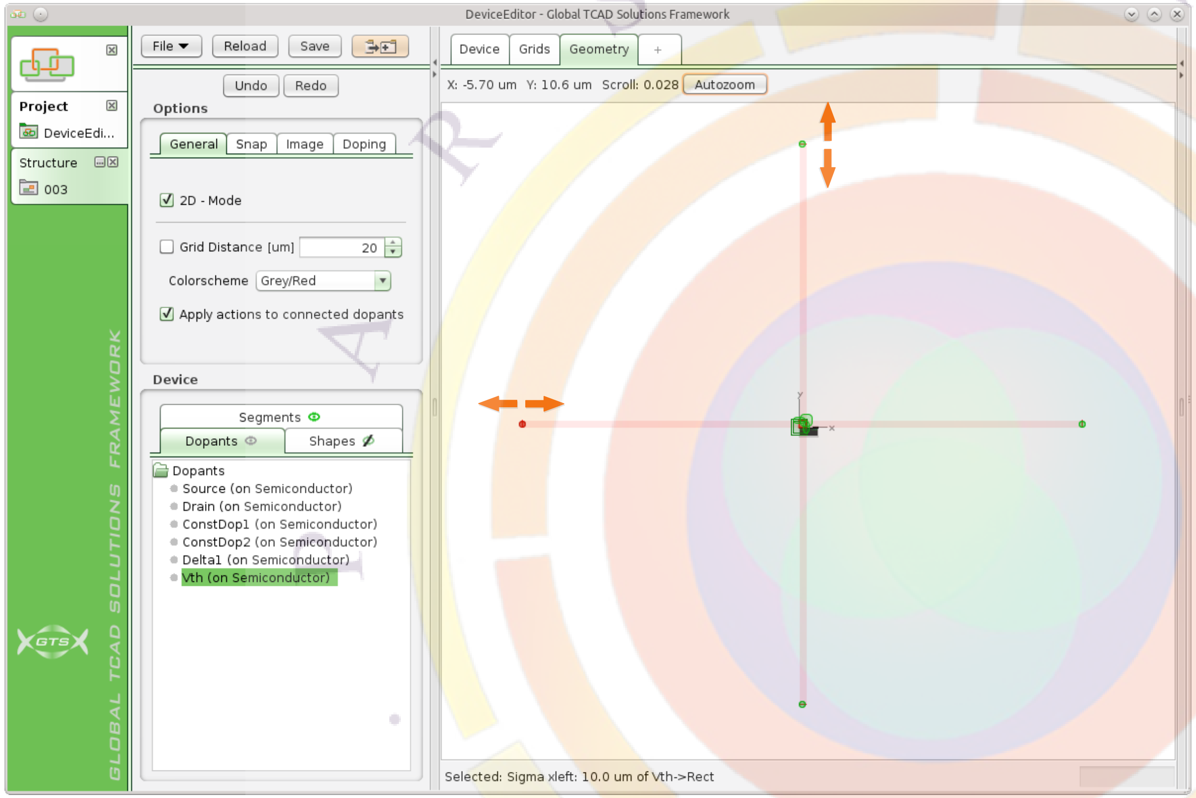
- Duplicate the old structure ToolFolder, using the "Next tool" button and open the new ToolFolder
- Switch to the "Geometry" page and choose the section "Dopants" instead of "Segments"
- Add a rectangular dopant by right-clicking "Dopants"

Don't forget to assign comments to newly generated ToolFolder.



1.2.2. Edit Doping Parameters

- Set the doping parameters as shown in the figure and confirm with “OK”
- Name the new doping area “Vth” via the option “Rename” in the context menu of the new doping area in the “Device” section of the control panel



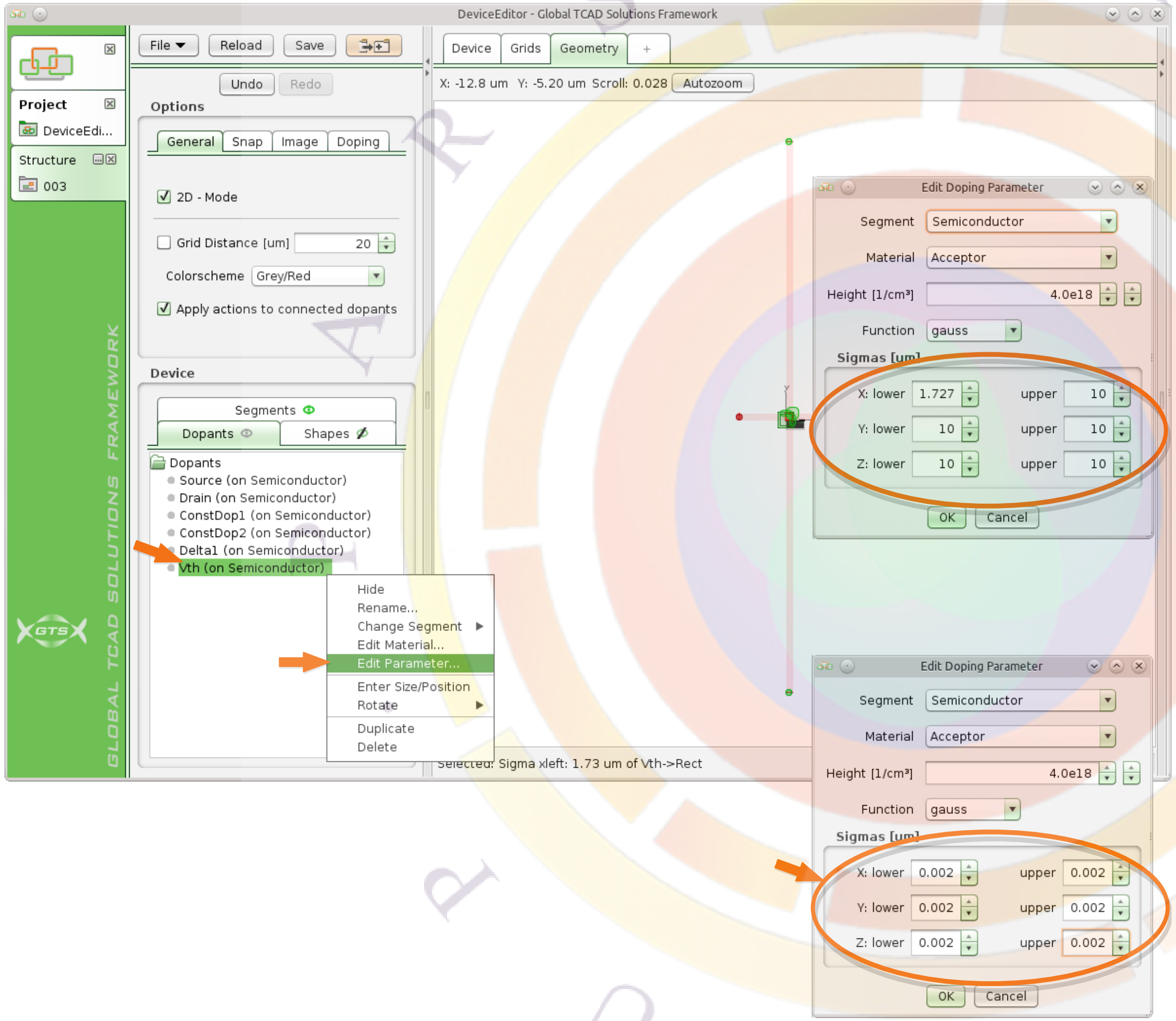
1.2.3. Edit Gaussian Distribution - Drag Points

Now we want to configure the gaussian distribution and the size and position of the new doping area using the drag points in the work area.

- Use the scroll wheel to zoom out until you see the whole gaussian distribution of the new doping area as shown in the figure
- Make sure the new doping area is selected
- The round drag points can be used to change the gaussian distribution of the doping. Click, hold and move the points with the left mouse button.

Use the “Undo” button to undo changes.

The drag points can only be moved horizontal or vertical according to the correspondant area.

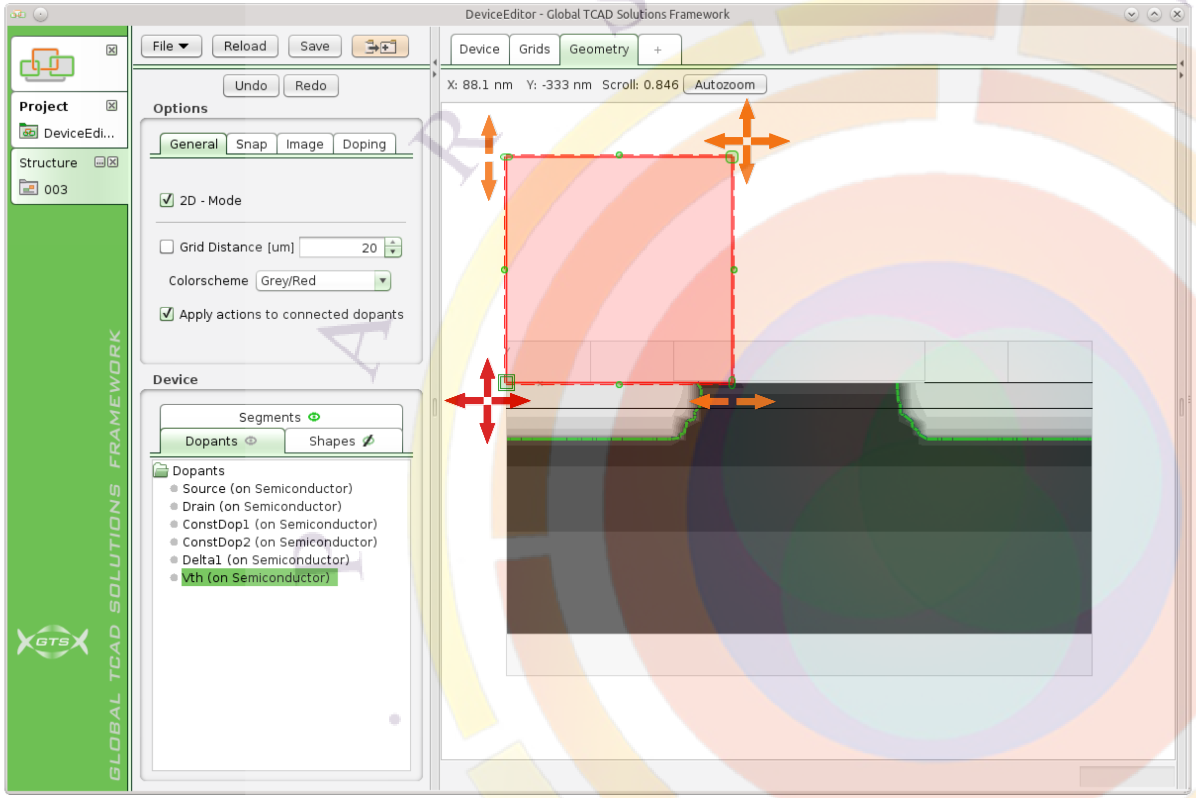


1.2.4. Edit Gaussian Distribution - Menu

In the previous step we have changed the standard deviation of the new doping area. We can analyse and change the standard deviations in the “Edit parameter...” menu as well:

- Right click the new doping area to enter its context menu
- Select “Edit parameter...”. Here you can see the change of one sigma value, as we have moved the correspondent drag point (Upper “Edit Doping Parameter” menu in the figure)
- Change the values of all sigmas to 0.002 as shown in the lower “Edit Doping Parameter” menu and confirm with “OK”

Now we have narrowed the gaussian distribution of the doping as we want to create a small Vth doping.



1.2.5. Edit Doping Area - Drag Points I

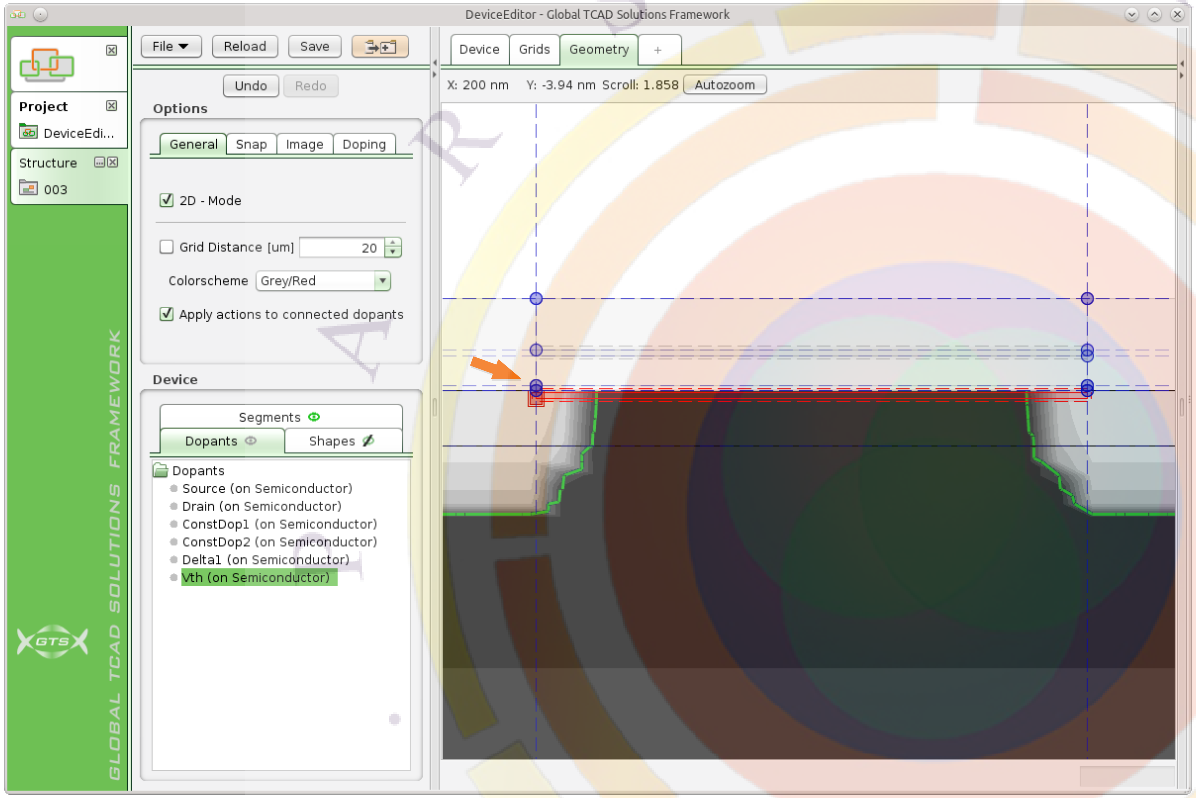
Now we want to edit the doping area itself. First we will do this using the drag points.

- Use the scroll wheel to zoom in

In addition to the previously used drag points of the gaussian distribution the doping area has four drag points, one in each corner:

- The square point, in the figure left bottom, can be used to change the position of the rectangle
- The oval point on the left top can be used to change the size of the rectangle only in vertical dimension
- The oval point on the right bottom can be used to change the size of the rectangle only in horizontal dimension
- The round point on the right top can be used to change the size of the rectangle in any dimension

To change a round or an oval drag point to an square drag point double click the desired point with the left mouse button.

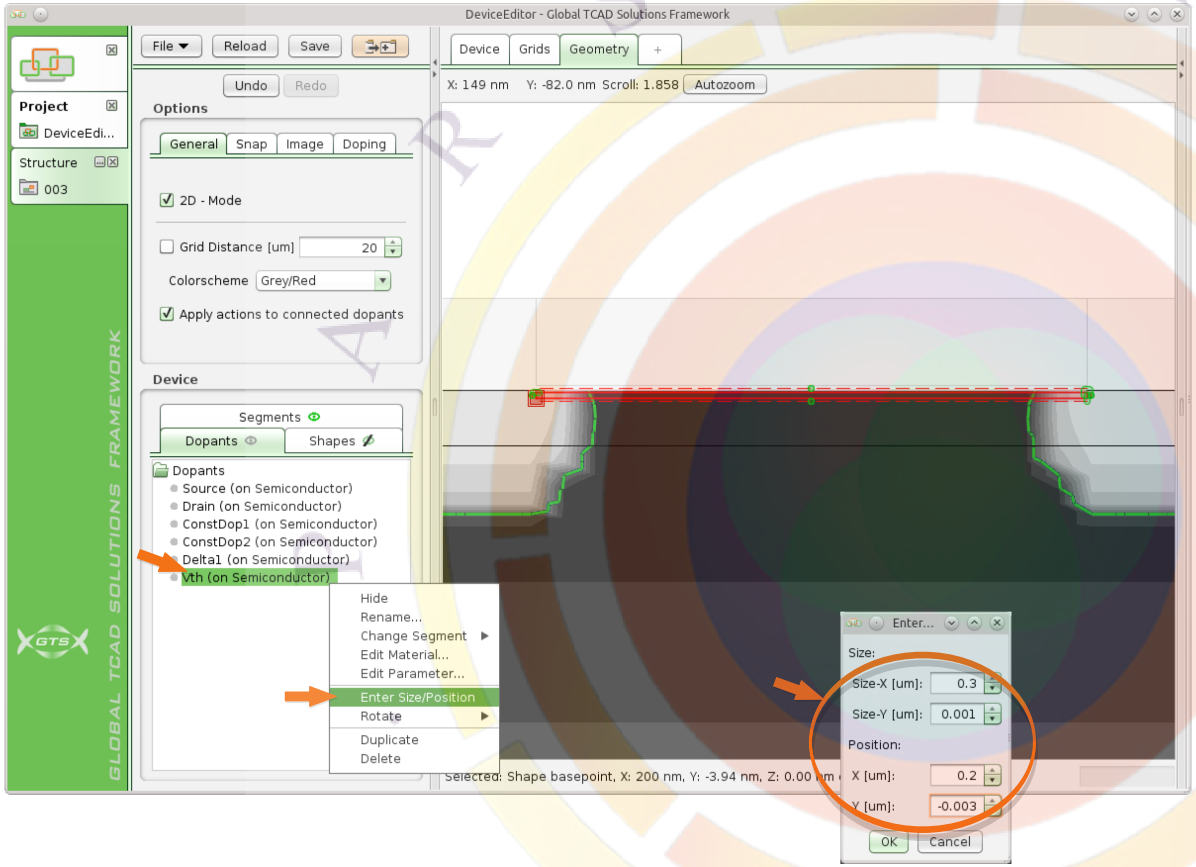


1.2.6. Edit Doping Area - Drag Points II

While you drag a round or an oval point, other significant points become blue points. If you drag a point close to one of the blue points it will snap and the point dragged will become a triangle.

- Try to get the doping area into size and position as shown in the figure. In the next step the size and position will be checked via the menu

Click and hold the right mouse button in the working area to change the area of view.

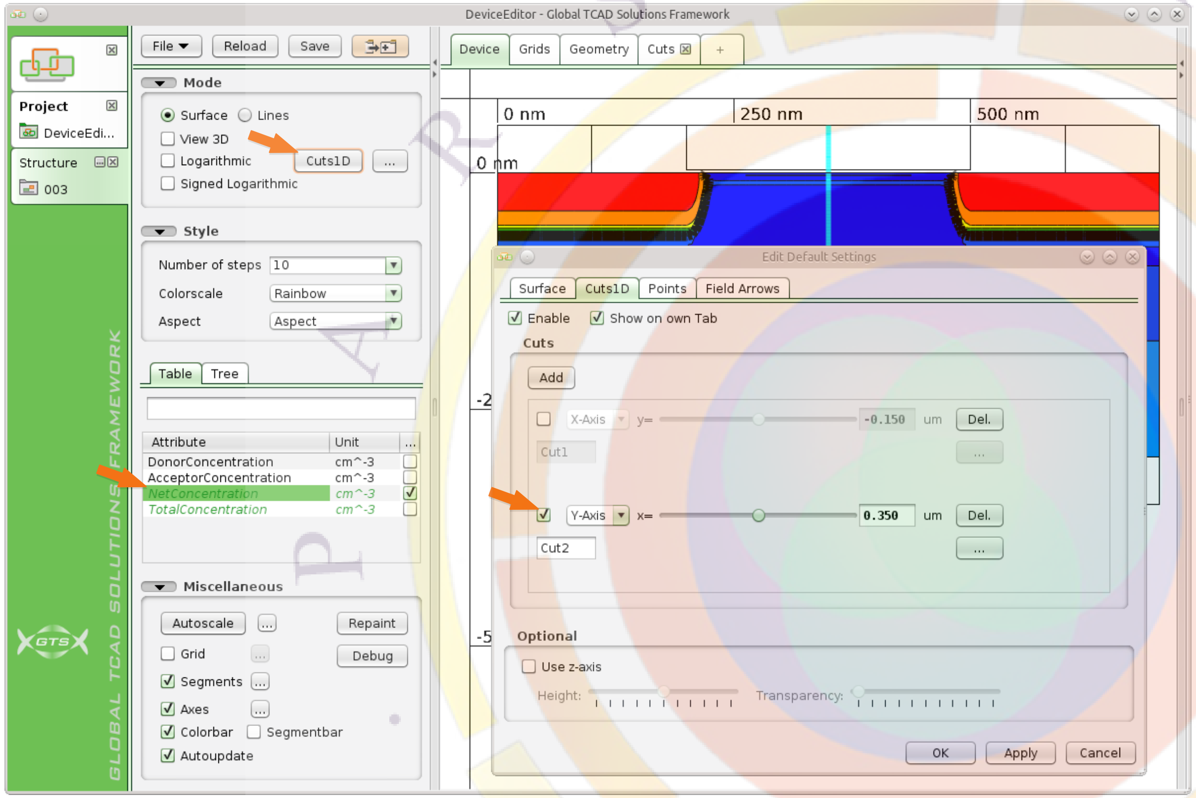


1.2.7. Edit Doping Area - Menu

We can analyse and change the size and position in the “Enter Size/Position” menu:

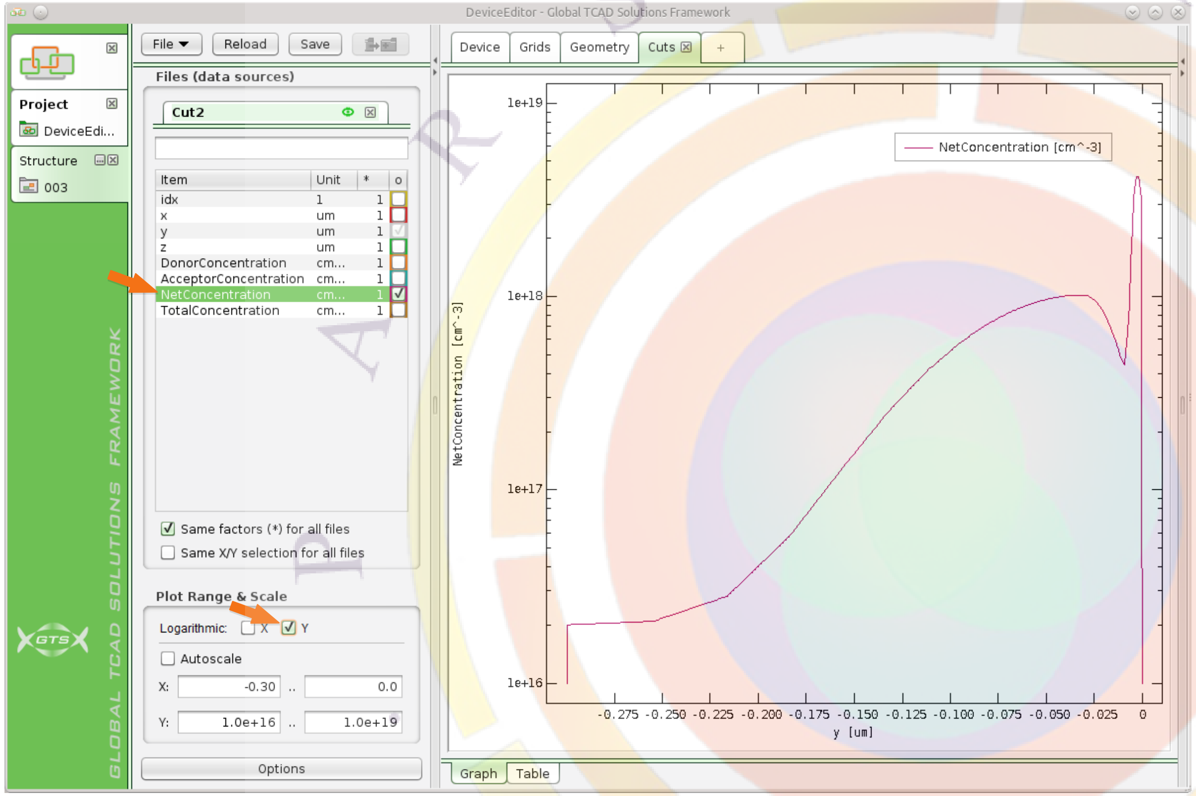
- Right click the new doping area to enter its context menu
- Select “Enter Size/Position”. Here you can see the current position of the doping area
- Make sure size and position are the same as shown in the figure and confirm with “OK”

Now the new Vth is placed and the gaussian distribution is set.



1.2.8. Vertical Cut

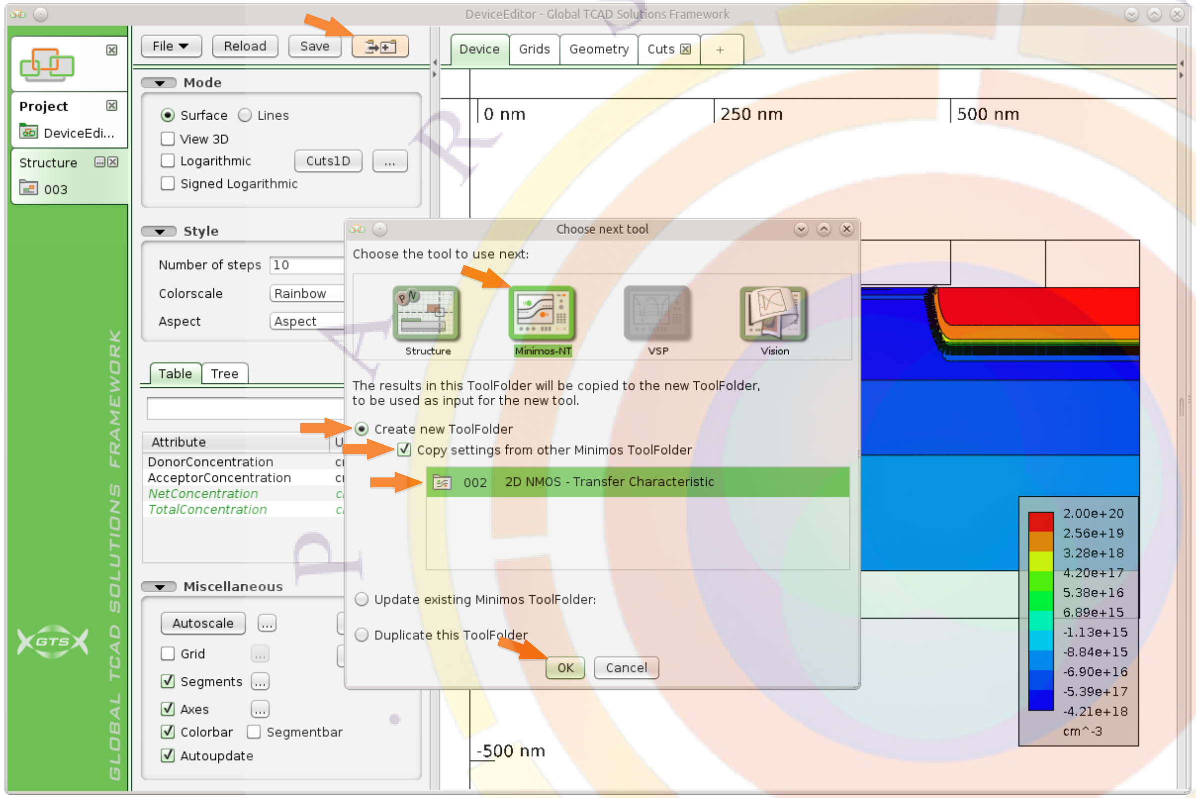
- Switch to the “Device” page and check “Net Concentration”
- Click on “Cuts1D” in the “Mode” section
- Select the Y-axis as shown in the figure and click “OK”
- The page “Cuts” will appear next to the “Geometry” page. Switch to this page.



1.2.9. Net Doping Profile

In the “Cuts” page choose again the “Net Concentration” and select “Logarithmic Y” scale. Additional to the well doping we can inspect the Vth doping close to the Gate.

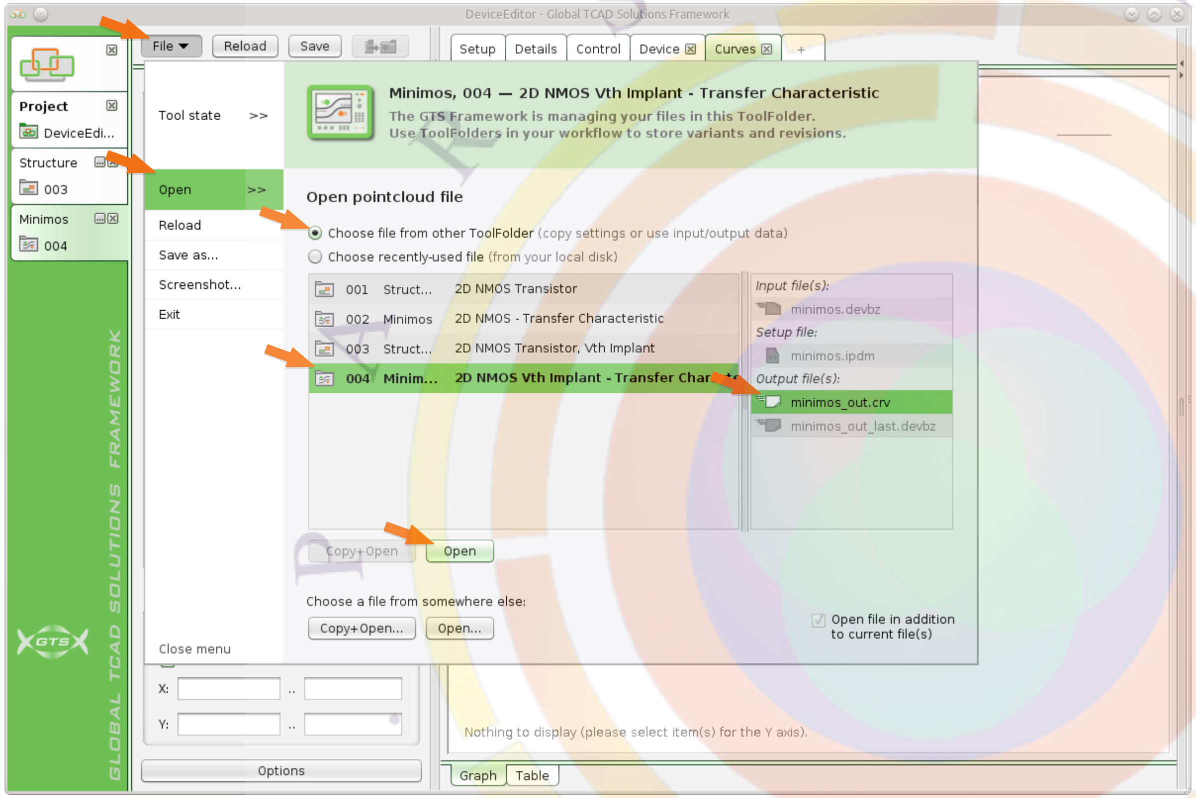
One can easily tune the doping profile using the “Geometry” and the “Cuts” page alternating. Change the doping in size, value and distribution in the “Geometry” page and check the resulting doping profile in the “Cuts” page.



1.2.10. Copy Simulation Settings

- Select the “Device” page again
- Click on the “Next tool” button, choose “Minimos-NT”, select “Create new ToolFolder” and “Copy settings from other Minimos ToolFolder”
- Select the Minimos ToolFolder we have used to simulate the transfer characteristic and press “OK”
- The settings are taken from the previous Minimos-NT run, so we can run the simulation in the “Control” page
- Switch to the “Curves” page after the calculations are done

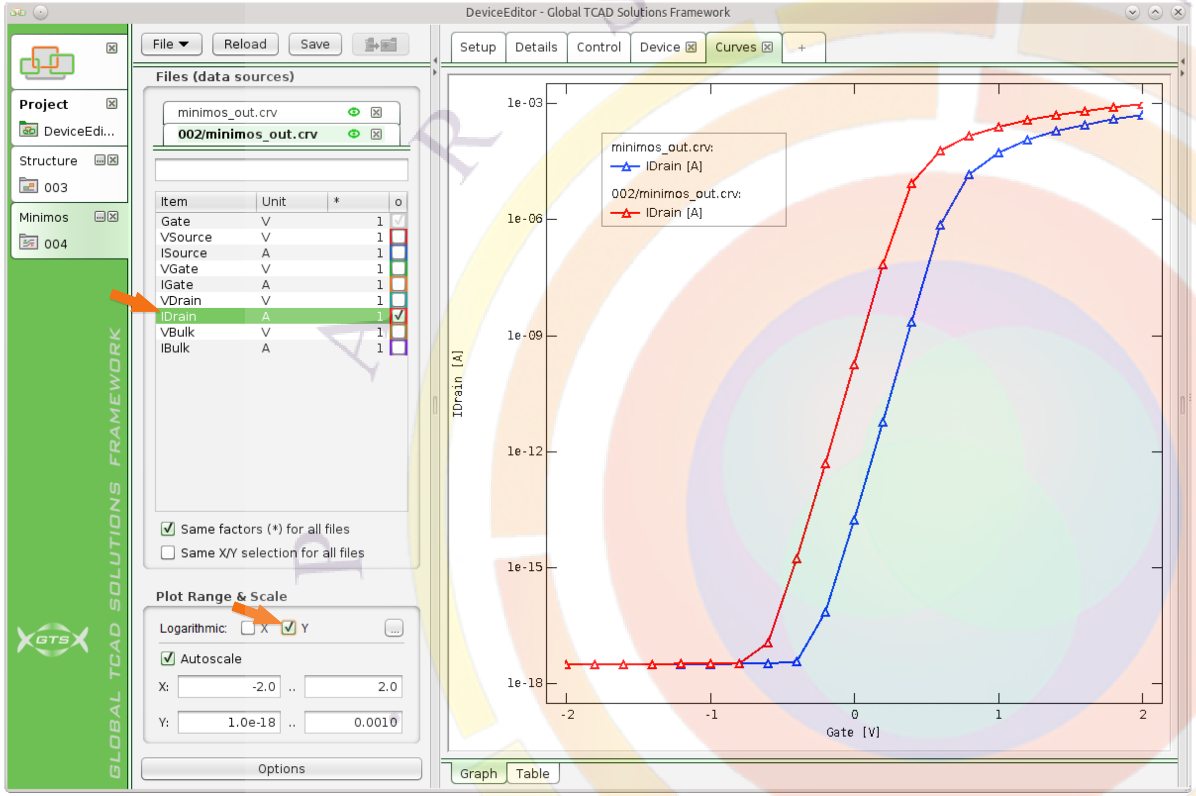
Compare the settings with 1.1.2 and start the simulation in the same way. How to run a simulation is explained in the “Getting Started” tutorial.



1.2.11. Open Files to Compare

In GTS Vision, you can easily open and compare files from different ToolFolders:

- In the "Curves" page select the menu "File » Open"
- Select "Choose File from other ToolFolder"
- In the ToolFolder list choose the one where you did the first Minimos simulation
- On the right-hand side, select the file `minimos_out.crv`
- Press "Open"



1.2.12. Compare Results

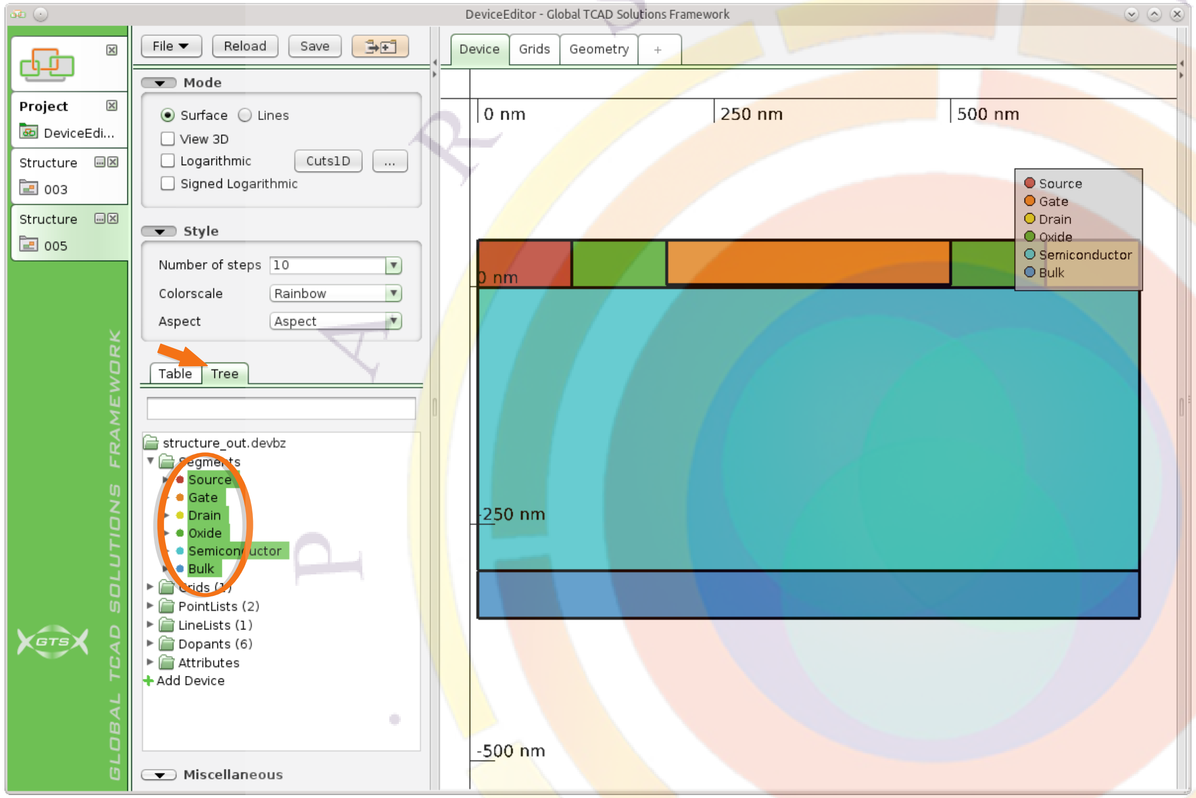
Now we just have to select the desired values and display properties in order to compare the differences in the transfer characteristic. In the figure the red curve corresponds to the output from ToolFolder 002 - the simulation of the original nMOS template.

Each opened file has its own attributes, even if they share a name. You may need to select each of the files in the file list above the attribute list and enable the attributes to be shown.

Part 2

Editing Device Structures

In this example, we will edit the structure of a two-dimensional device: Starting from the modified nMOS template with the threshold voltage doping, generated in chapter one, we will add a poly-silicon gate segment. Next, we will carry out a simple simulation on this device.



2.1. PolySi Gate

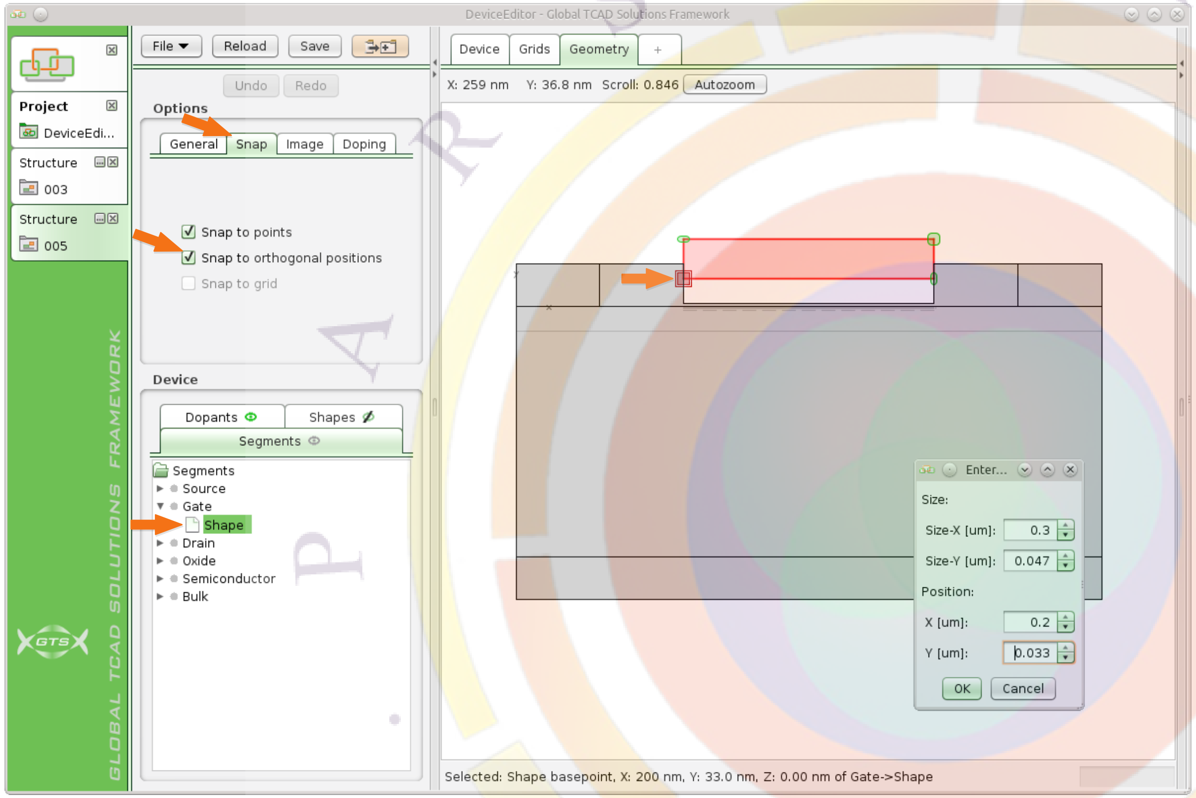
2.1.1. Structure ToolFolder

Duplicate the previous structure ToolFolder and display the existing segments of the device as shown in the figure:

- As we want to extend our modified device with the Vth implant, duplicate the structure ToolFolder “003”
- Use the tab “Details” to switch to the detailed tree view
- Double-click on a tree element to open or close the branch containing the sub-elements
- Select the segments you want to display

To select multiple items, hold the Shift or Ctrl key while clicking on an additional item.

The context menus of the segments offer various display modes.



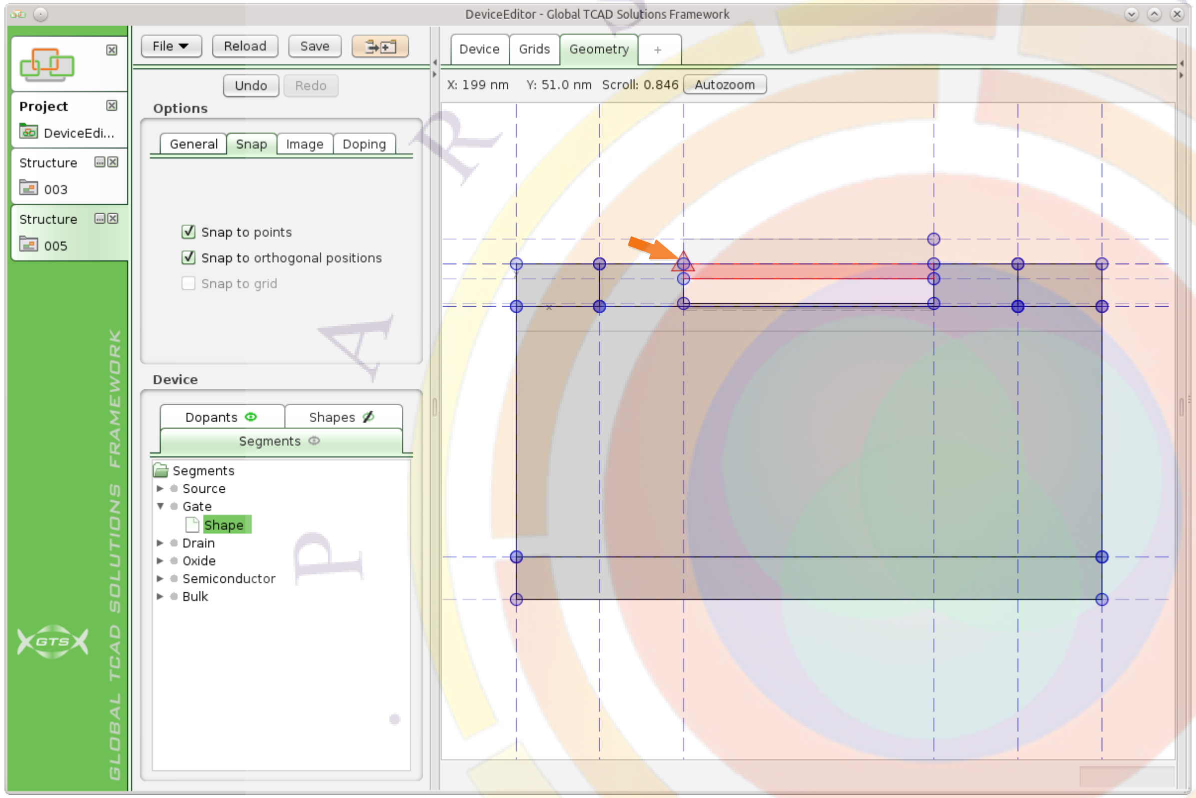
2.1.2. Edit Shapes

Move the Gate contact to make space for the poly-silicon segment:

- Switch to the “Geometry” page
- Shift the segment “Gate” 30 nm upwards by using the drag-point of the segment which is marked by a rectangle
- Make sure the segment keeps aligned with the oxide, which can be easily done by activating “Snap to orthogonal positions”

The drag point is marked by a rectangle. To change the drag-point, double-click the designated corner of the rectangle.

The size and position can also be edited by right-clicking on the rectangle an choosing “Enter Size/Position” in the context menu.

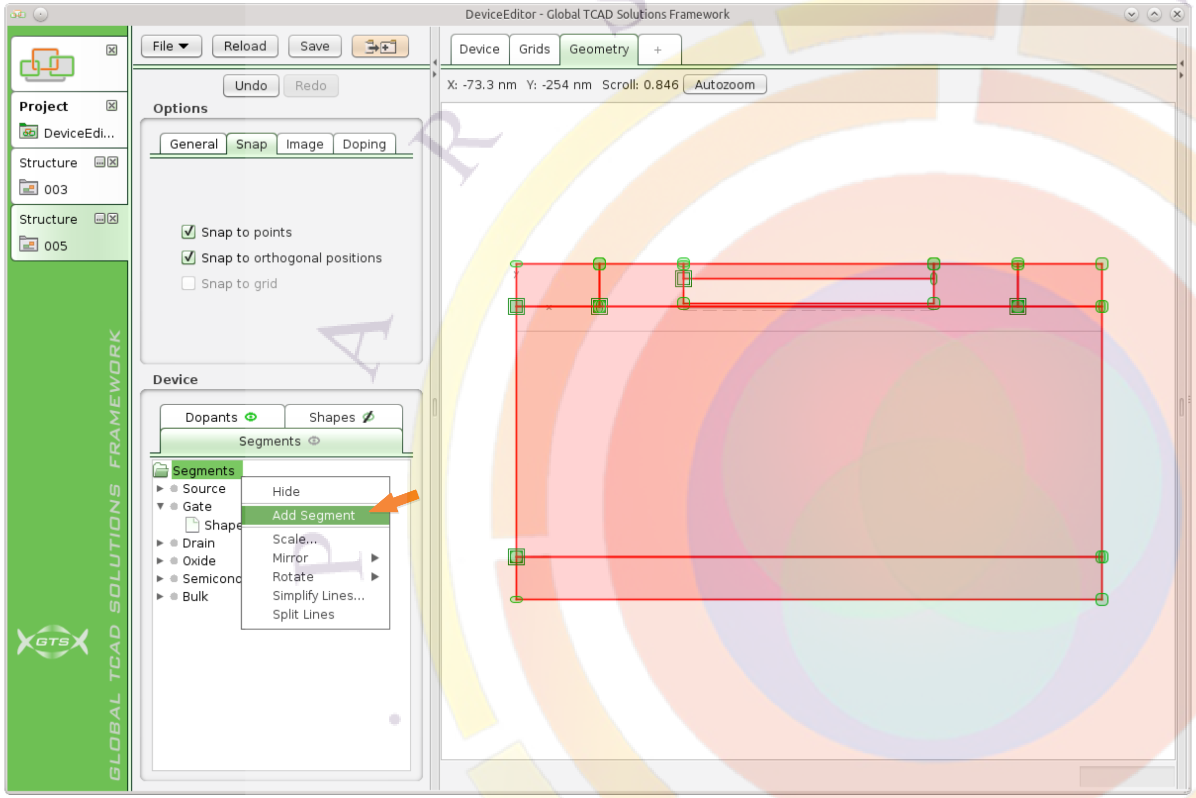


2.1.3. Change Size

Reduce the size of the gate segment shape:

- Make sure that one of the lower points is the drag-point (see hint in previous step)
- Reduce the size of the gate by dragging one of the upper points using the left mouse button

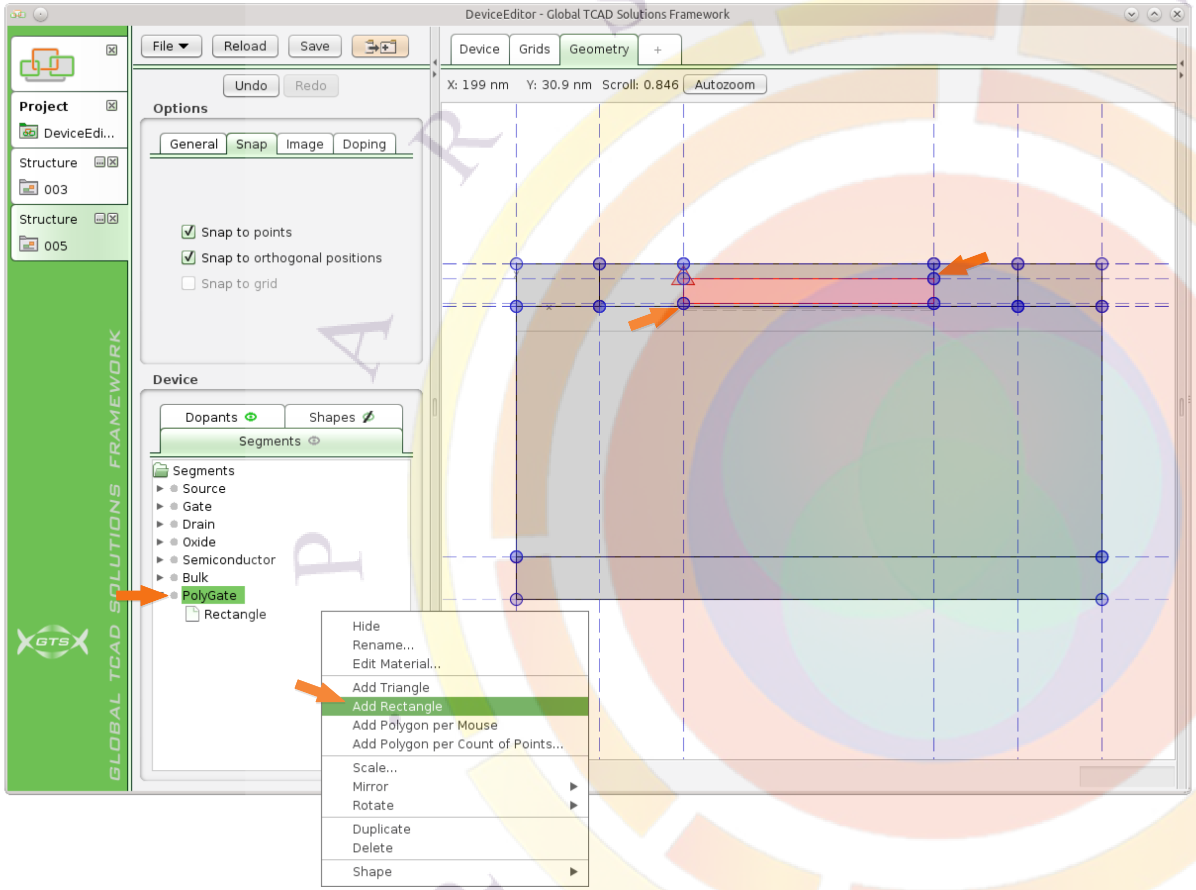
Depending on its relative position to the drag point, the points of a rectangle offer different degrees of freedom in movement as depicted by its symbol. By selecting a different drag point, the allowed moment of the other shape points gets modified.



2.1.4. Add Segment

Next, we are going to add the poly-silicon layer:

- Use the right mouse button to enter the context menu of the list entry “Segment”
- Select “Add Segment”
- Rename the newly created segment to “PolyGate” by using the “Rename” function of its context menu

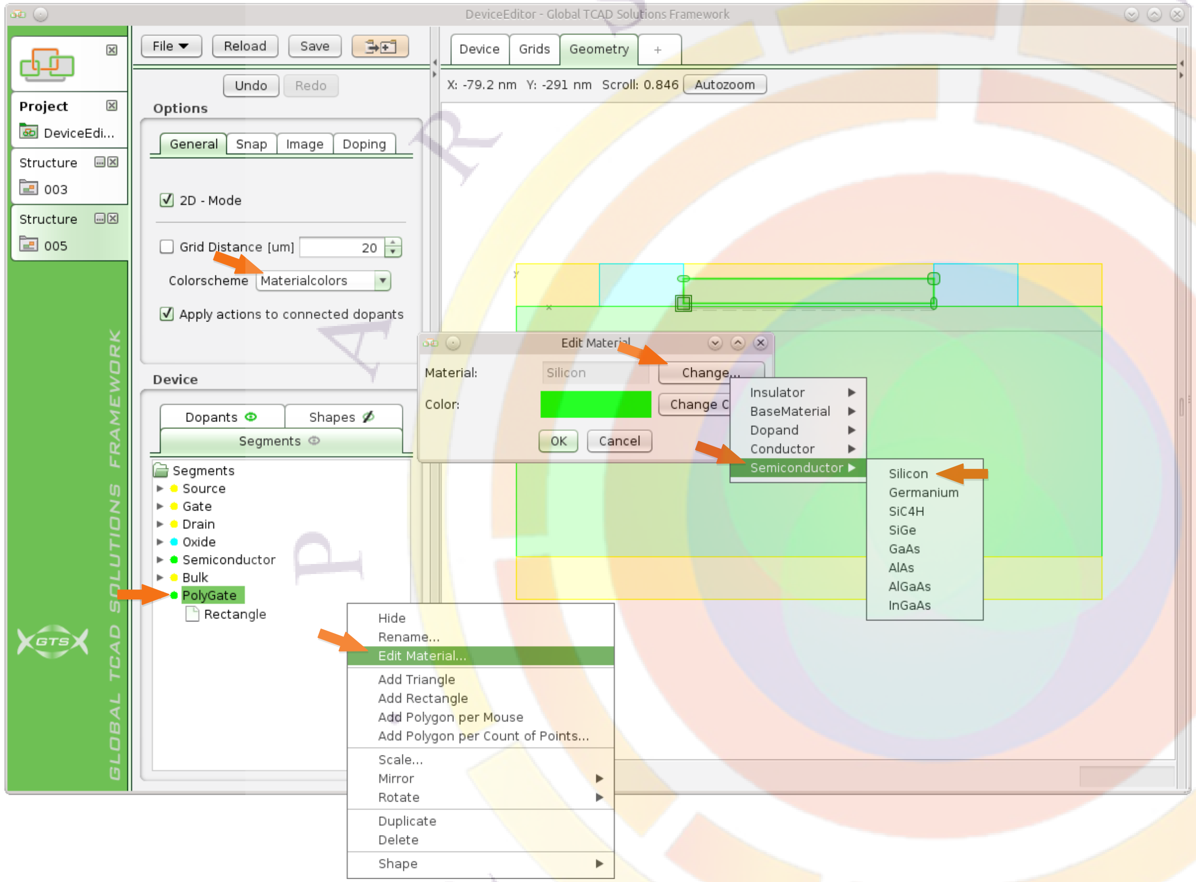


2.1.5. Add Segment Shape

Add a shape for the newly created segment:

- Use “Add Rectangle” of the “PolyGate” segment context menu
- Use the handles to resize the generated shape
- Place the rectangle as poly-silicon layer between oxide and contact
- Make sure the segment shape is aligned properly to the neighboring segments

Use the mouse wheel to zoom in or out.

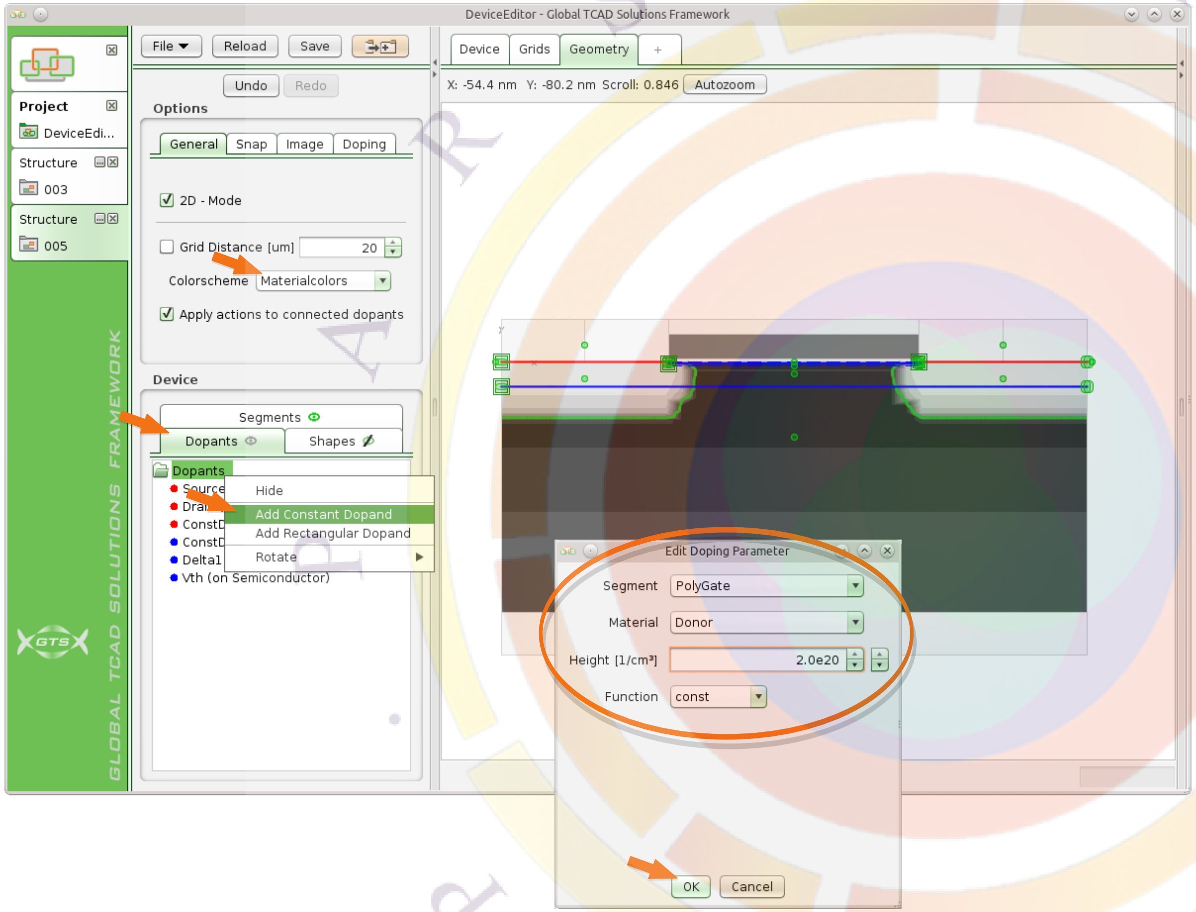


2.1.6. Set Segment Material

To set the material, do the following:

- Select the “Edit Material” menu in the context menu of the “PolyGate” segment
- Set the material to “Silicon”

Activate the Colors scheme “Materialcolors” in the tool panel of the “Geometry” page to check the material types of the segments.



2.2. Edit Dopants

2.2.1. Add Dopants

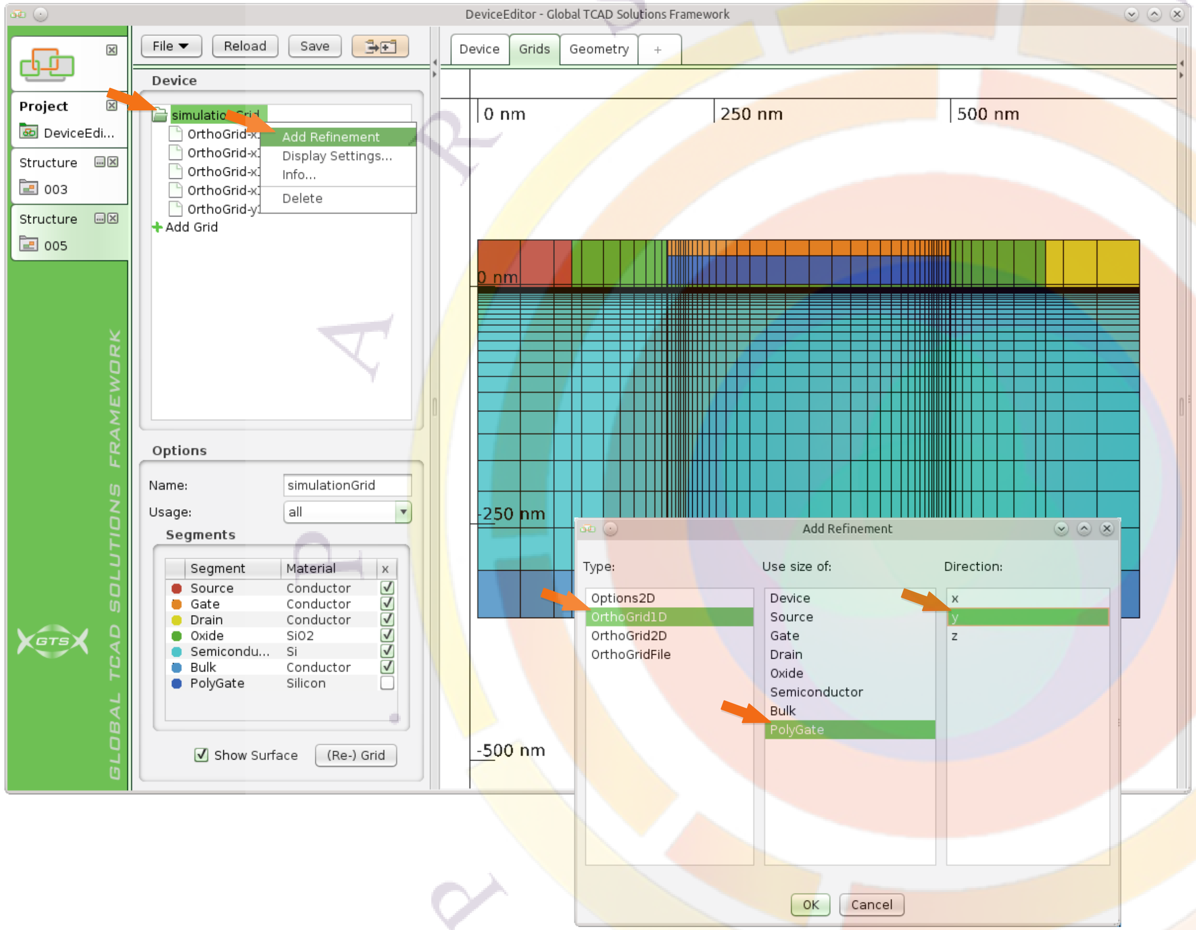
After adding the segment for the poly-silicon layer to the device, we need to apply a proper doping.

- Switch to the “Dopants” tab in the device panel
- Ensure that “Color scheme” is set to “Materialcolors”

Analytical doping profiles are defined by DopingAreaGroups of dopant material and shapes, similar to geometrical segments. The red and blue lines show existing donor and acceptor dopants, respectively. The effective net-doping profile is presented in the background, light-gray represents a high and black represents a low concentration. The green line depicts the p-n junction.

Now add the doping for the PolyGate:

- Use the context menu of the element “Dopants” in the tree and press “Add Constant Dopant”
- Choose the segment “PolyGate”
- Define the doping “Height” as $2\mathrm{e}20\,\mathrm{cm}^{-3}$

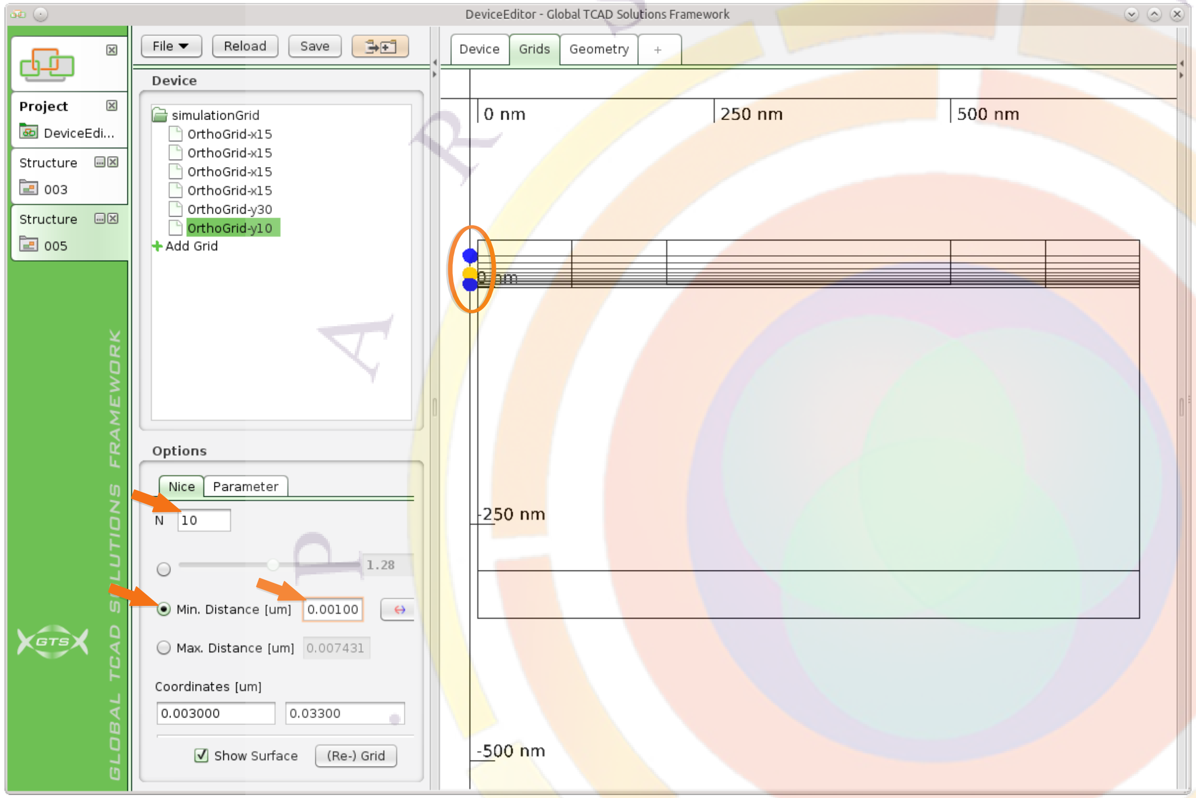


2.3. Edit Grid

2.3.1. Add Refinement

In order to simulate carrier depletion in the poly-gate, we need a sufficiently fine simulation mesh in the poly-gate at the interface to the oxide. This can be achieved by adding a further ortho-grid refinement for the Y-axis at the poly gate.

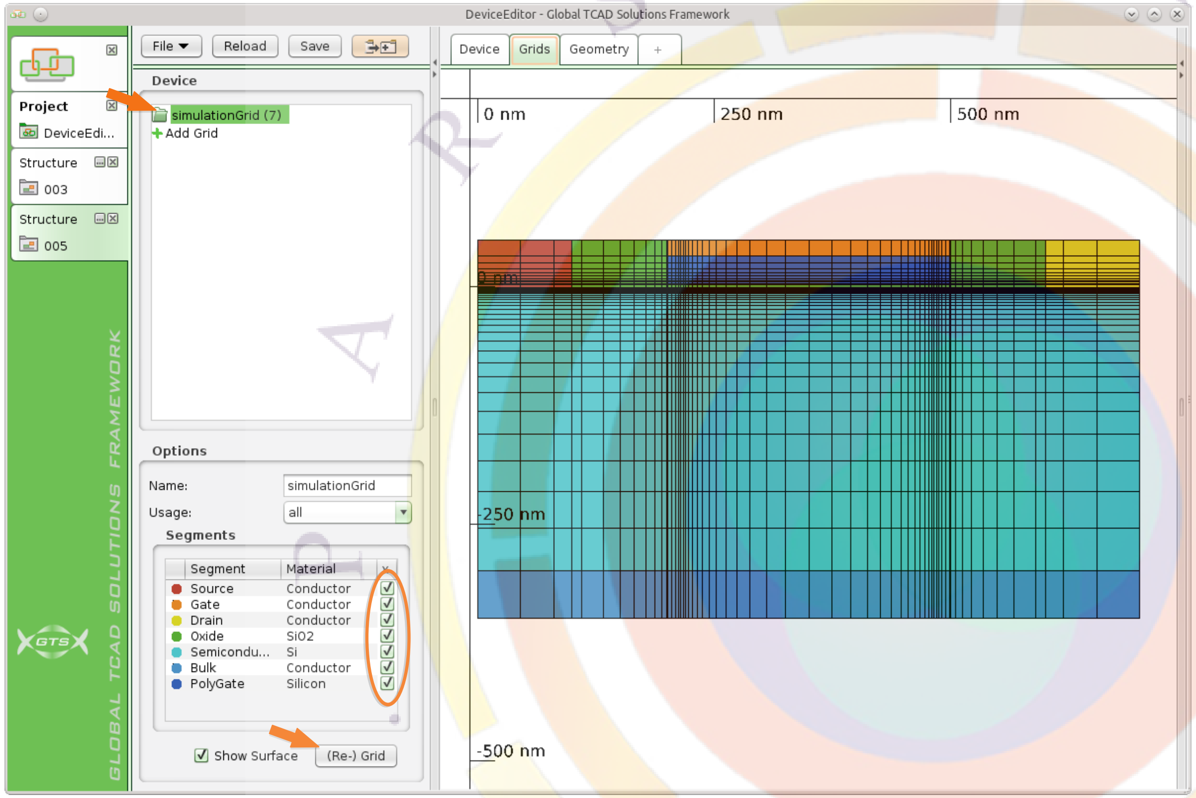
- Switch to the “Grids” page
- Double click on “simulationGrid” to check existing refinements
- Select “Add Refinement” from the context menu of the entry “simulationGrid”
- Choose “OrthoGrid1D”, with size of “PolyGate”, and direction “y”



2.3.2. Adjust Refinement

To adjust the refinement, either use the blue and yellow handles at the device axis or, for the exact definition, directly access the parameters in the options panel at the bottom left:

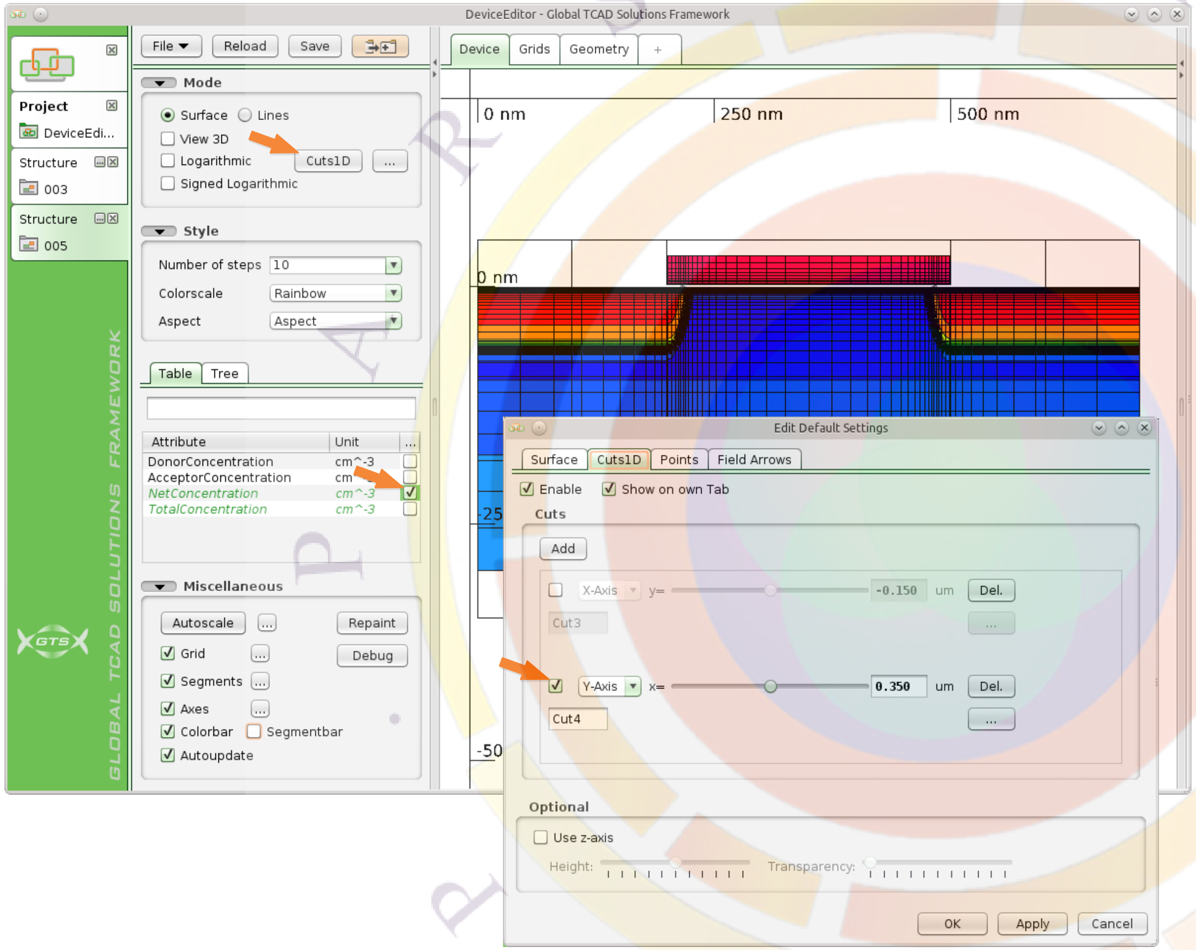
- Enter 10 in the field for the number of grid lines
- Set the minimum distance to 1 nm by entering `0.001` in the field “Min. Distance”
- Hit the enter key of your keyboard to update the settings



2.3.3. Make Simulation Grid

Apply the changes and re-grid the device:

- Select “simulationGrid” in the grid tree
- Check that all segments are selected (the default is to include new segments automatically)
- Press the “(Re-) Grid” button.



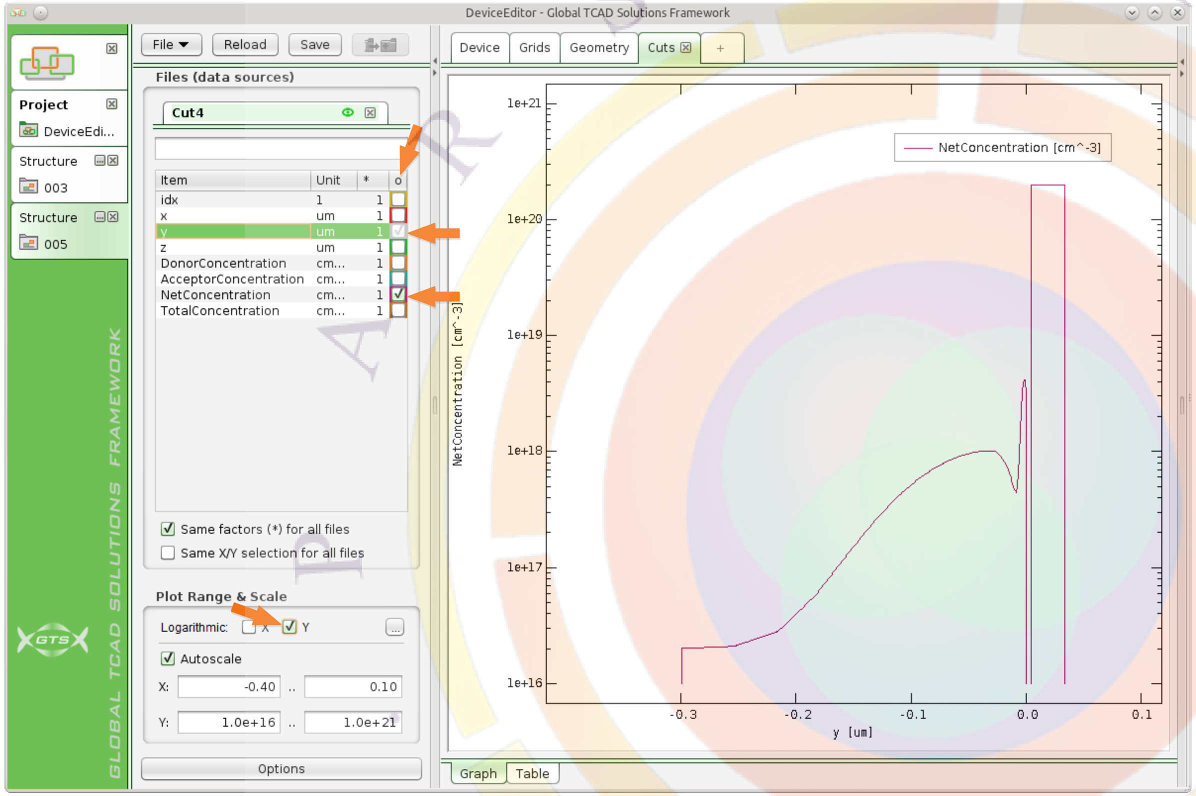
2.3.4. Net-Doping Profile, 1D Cut

After creating the device geometry and the mesh, we are going to check the doping profile:

- Switch to the "Device" page
- Switch to the "Table" view
- Check "NetConcentration"
- Check "Grid" to see the grid

To check the doping gradient, make a one-dimensional cut through the device:

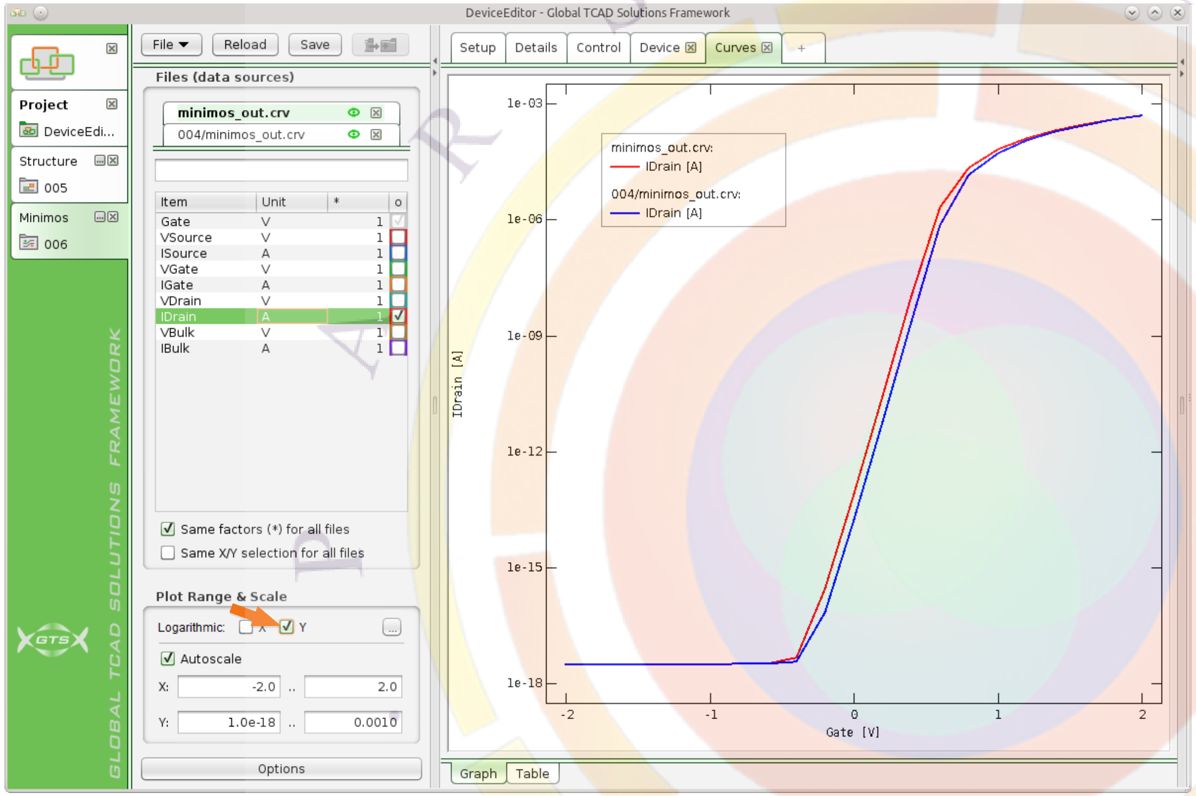
- Click on "Cut1D" in the Mode section
- Activate a cut parallel to the Y-axis
- Define the x-position of the cut, default is the center of the device, which fits perfectly here
- Press "OK" to generate the cut, this adds a "Cuts" page at the top of the work area



2.3.5. Display One-Dimensional Cut

To display the net doping concentration like in the figure, perform the following steps:

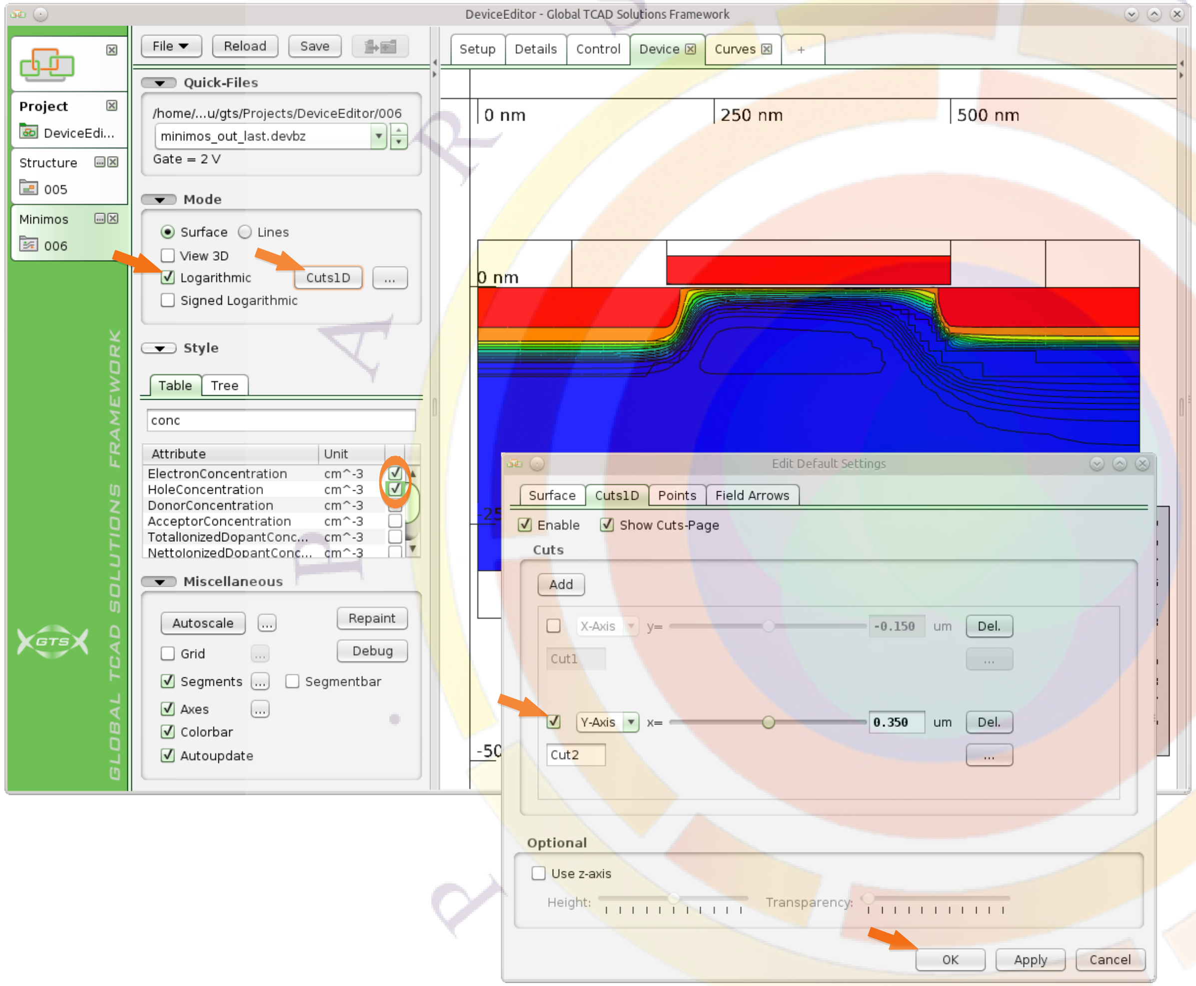
- Click on the table header above the rightmost column to deselect all quantities (if not done automatically)
- Select the quantity “y”; right-click to open the context menu and choose “Set as X-axis”to plot the device coordinate “y” on the abscissa (if not done automatically)
- Check the “NetConcentration” quantity
- Set the “Plot Range & Scale” to “Logarithmic Y”



2.3.6. Copy Settings, Compare Results

Now we want to compare the results of the simulation of the original nMOS and the modified nMOS with the PolySi Gate.

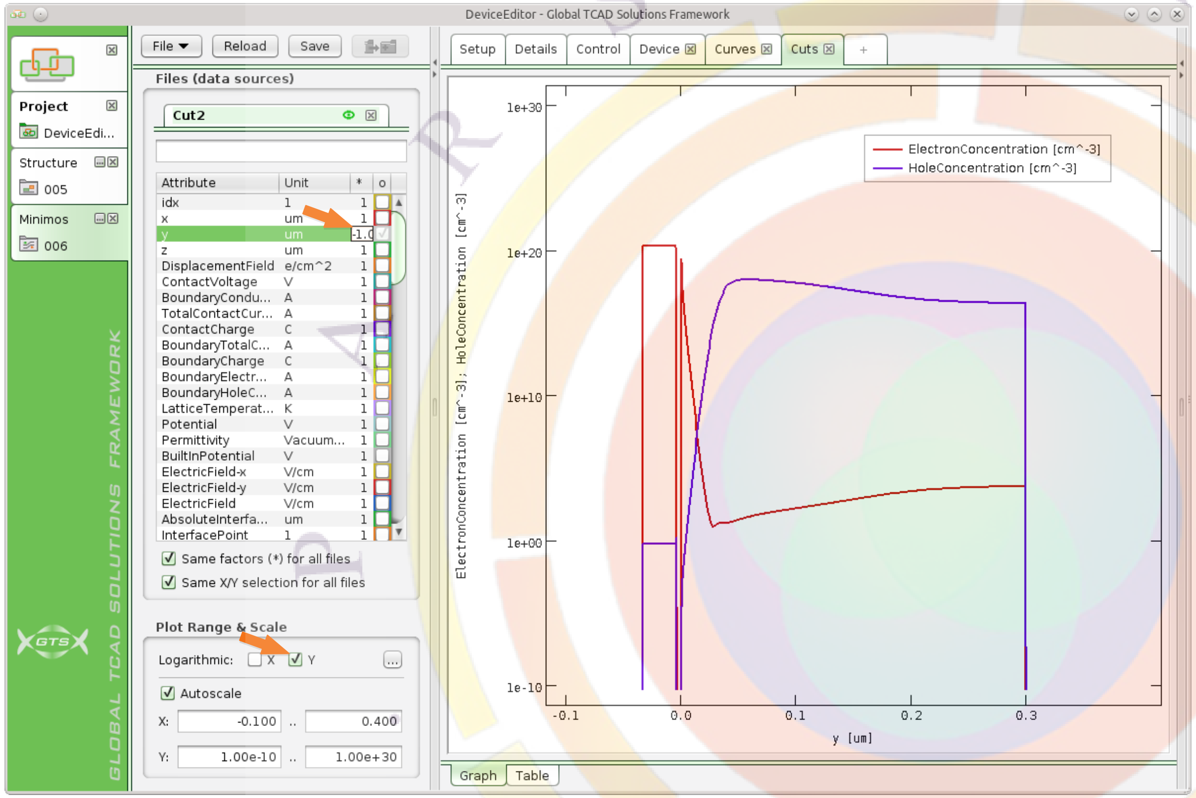
- Switch back to the "Device" page again
- Press the "Next tool" button, choose "Minimos-NT", select "Create new ToolFolder" and "Copy settings from other Minimos ToolFolder" (compare 1.2.10)
- Select one of the Minimos ToolFolder we have used to simulate the transfer characteristic and click "OK"
- The settings are the same and we can run the simulation in the "Control" page
- In the "Curves" page we can compare the transfer characteristics by selecting the desired values and opening the previous simulations (compare 1.2.11)



2.3.7. Carrier Density

Finally, we want to check the carrier depletion in the poly-gate layer in on-state.

- Switch to the "Device" page
- Check the quantities "ElectronConcentration" and "HoleConcentration" in the quantity table
- Select "Logarithmic" to obtain a logarithmic interpolation
- Click on "Cut1D" in the Mode section
- Activate a cut parallel to the Y-axis
- Define the x-position of the cut, default is the center of the device, which fits perfectly here
- Press "OK" to generate the cut, this adds a "Cuts" page at the top of the work area



2.3.8. Carrier Depletion

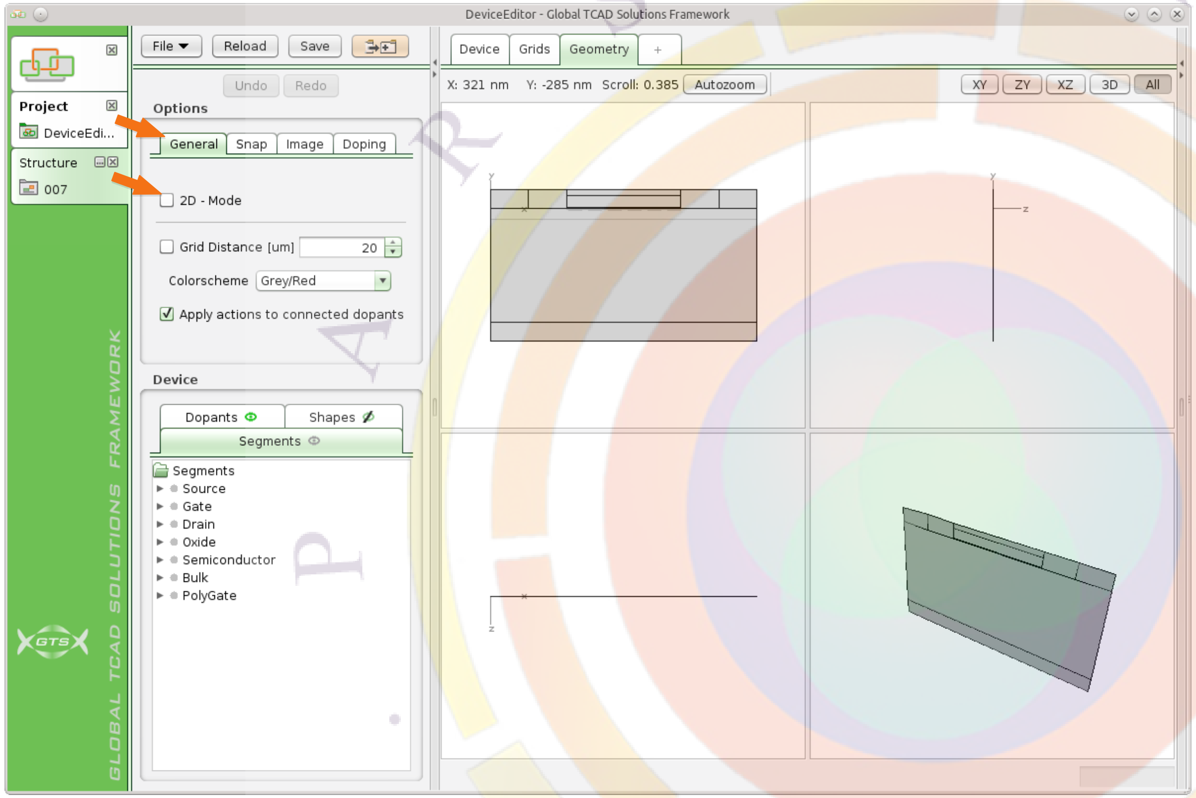
To plot the carrier concentrations as shown in the figure:

- Select the quantity “y”; right-click to open the context menu and choose “Set as X-axis” to plot the device coordinate “y” on the abscissa
- To have the surface at the left-hand side of the plot, change the multiplication factor “*” of the quantity “y” to `-1.0`
- Check both “ElectronConcentration” and “HoleConcentration”
- Set the “Plot Range & Scale” radio button to “Logarithmic”
- To zoom, click in one corner of the plot, press and hold down the left mouse button and move to the other corner of the plot area you want to zoom in

Part 3

Editing 3D Device Structures

In this example we will extrude the 2D device, created in the last section. It will be explained how the segments and the mesh can be extended. Simulation results will be compared to the formerly created 2D results. Based on this newly generated 3D nMOS we will get familiar with editing 3D devices. Step by step the simple nMOS is changed to a STI-MOS. The Results of both 3D structures will be compared.



3.1. Extrude Segments

3.1.1. Duplicate, Switch Mode

The 3D device will be based upon a copy of the 2D device with the PolyGate created in the last section of the tutorial.

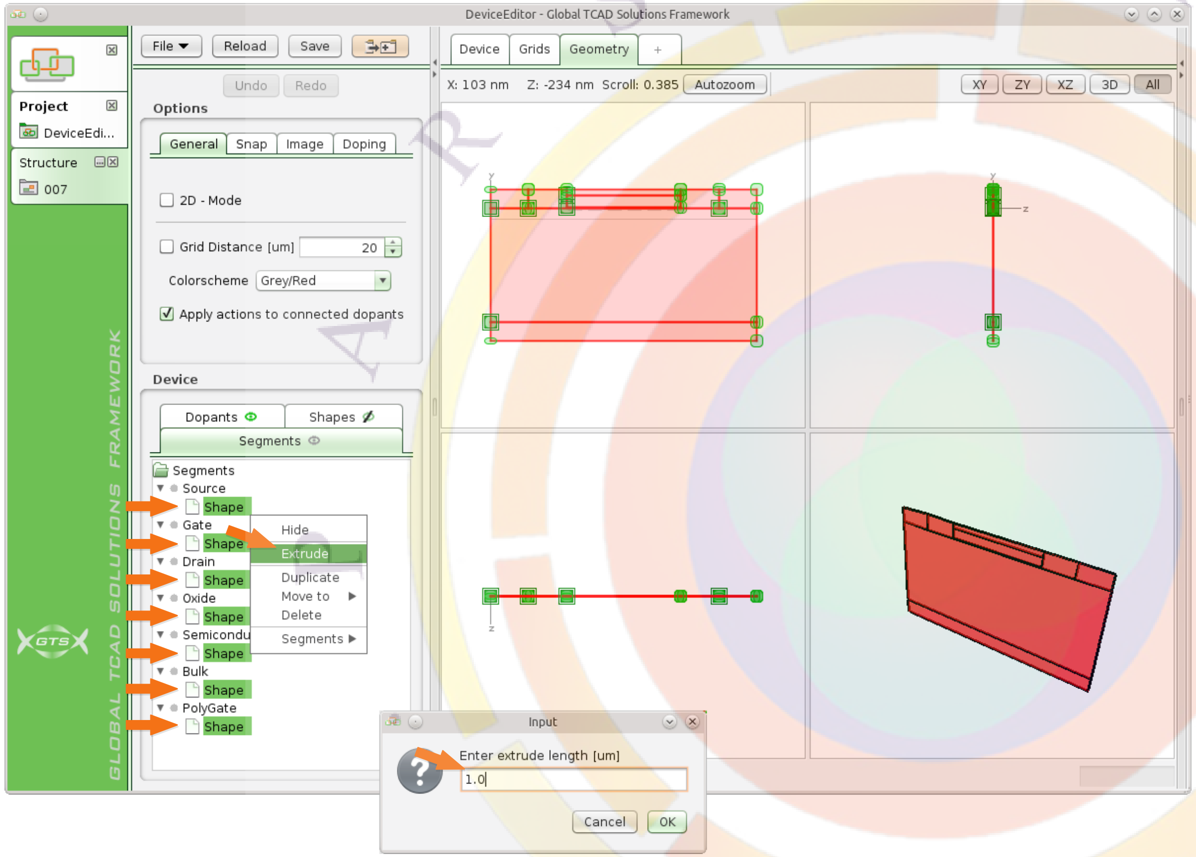
- Switch to the “Project” view and press the “Next tool” button of the latest 2D-structure
- Select “Duplicate this ToolFolder” and press “OK”

Switch from 2D to 3D mode:

- Switch to the “Geometry” page
- In the “Options” area switch to the tab “General”
- Uncheck “2D-Mode”

The device panel is split into 4 views: Top-, front-, side- and perspective 3D.

Use the mouse wheel to zoom in and out in the 2D and 3D views.

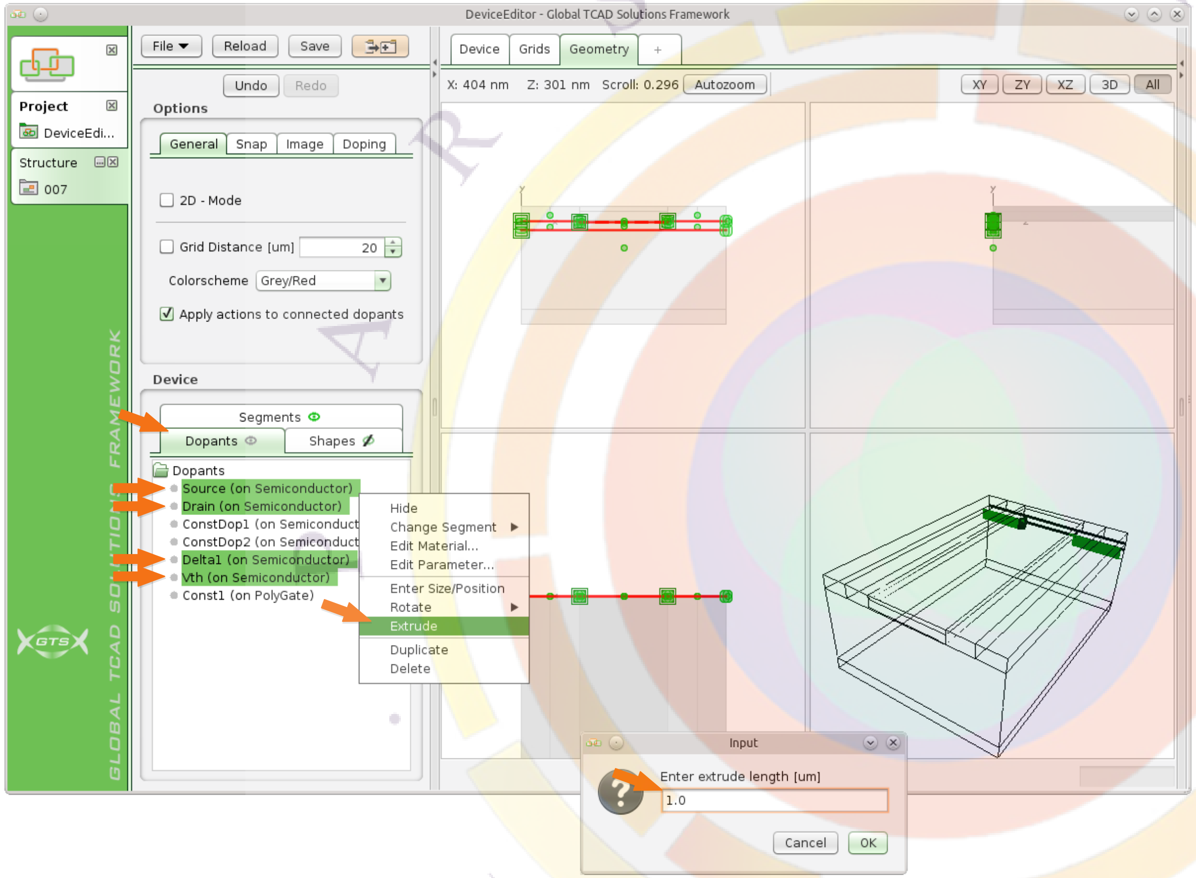


3.1.2. Extrude Segments

Extrude all segments:

- In the segments tree, open all segments by double clicking on them
- Select all shapes at the same time (see hint below)
- Use the right mouse button on one of the shapes and select "Extrude"
- In the dialog appearing enter the desired width of the device: $1\,\mu$m
- Press "OK"

To select more than one item, press and hold down the Ctrl key while clicking on additional items.

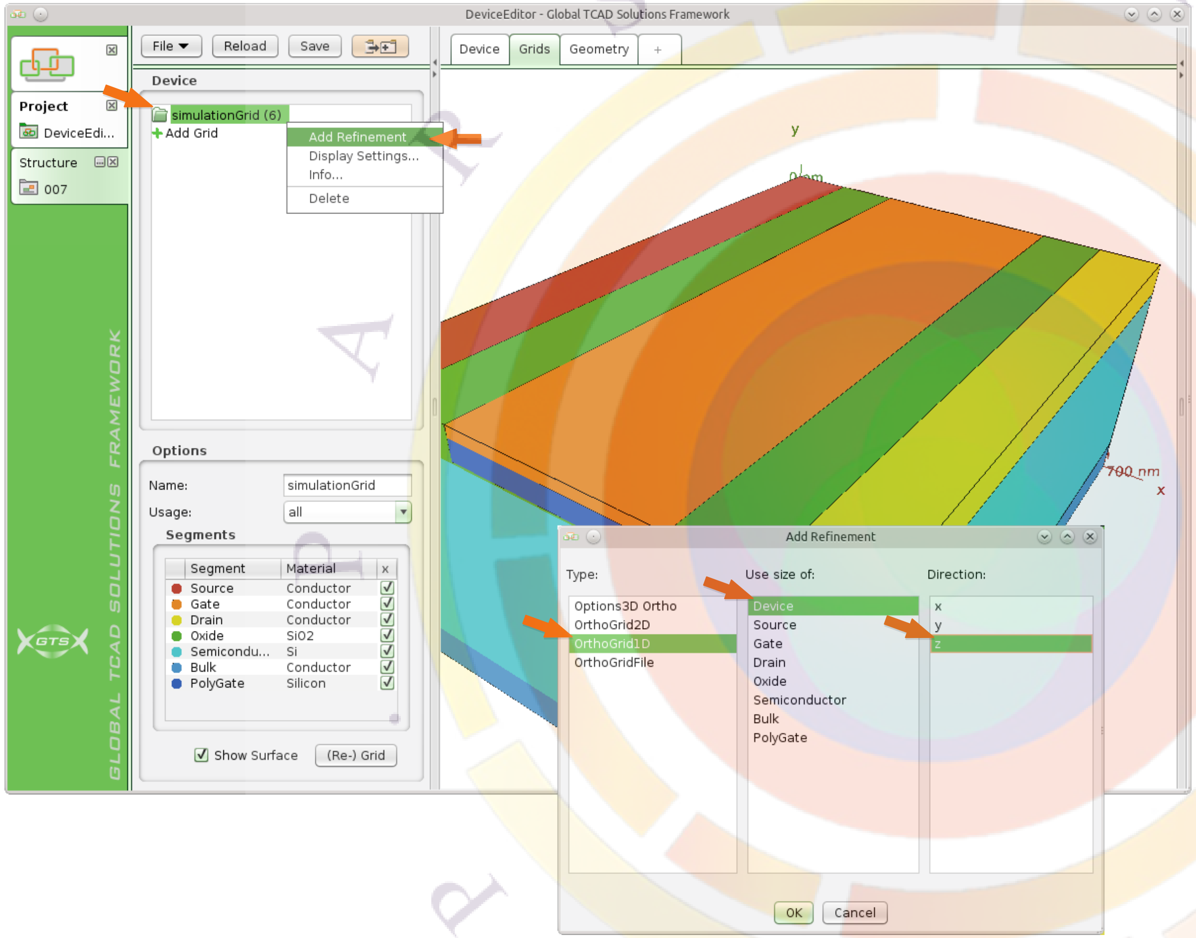


3.1.3. Extrude Doping

Repeat the last steps for the dopants:

- Switch to the "Dopants" tab
- Select the four non constant doping shapes as shown in the figure (one by one, using the Ctrl key)
- With this selection, right-click on one of the shapes and select "Extrude"
- In the dialog appearing, enter the desired width of the device: 1 μm
- Press "OK"

There are no areas to expand for constant dopings.

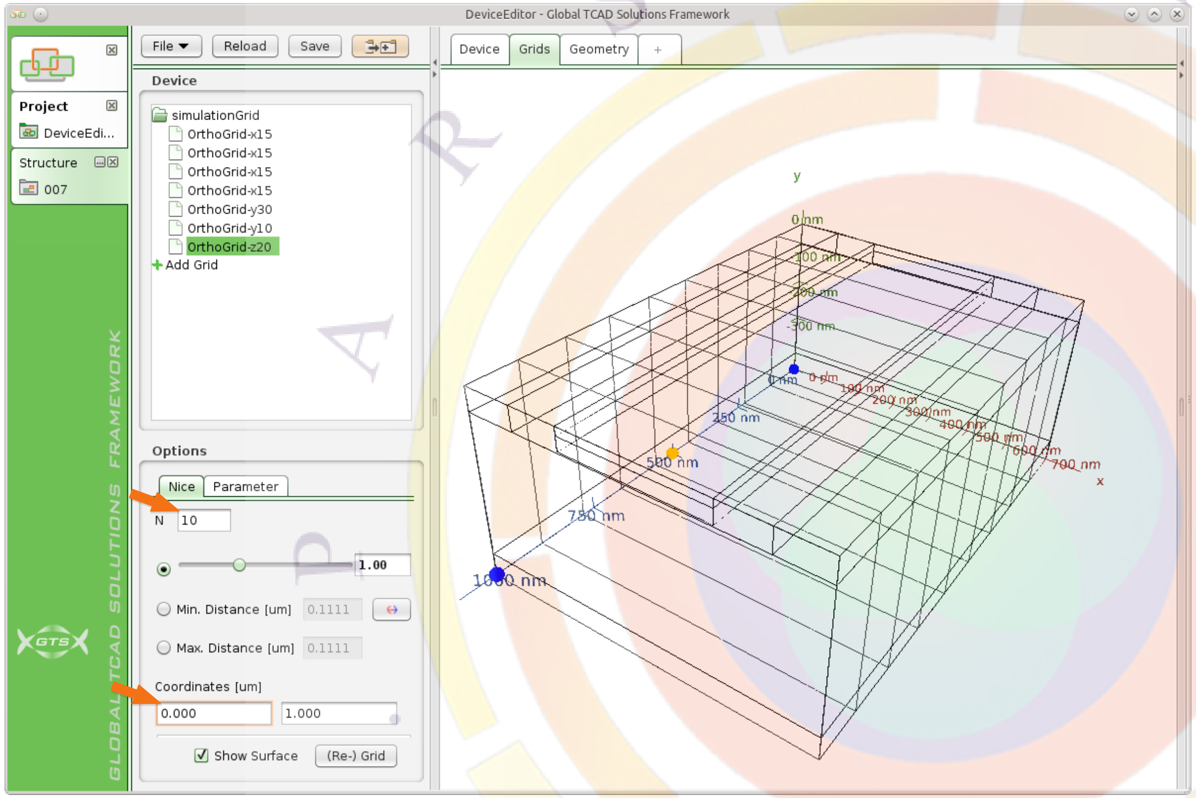


3.2. Edit Grid

3.2.1. Add Refinement in Third Dimension

Do the following to add a refinement in the third dimension:

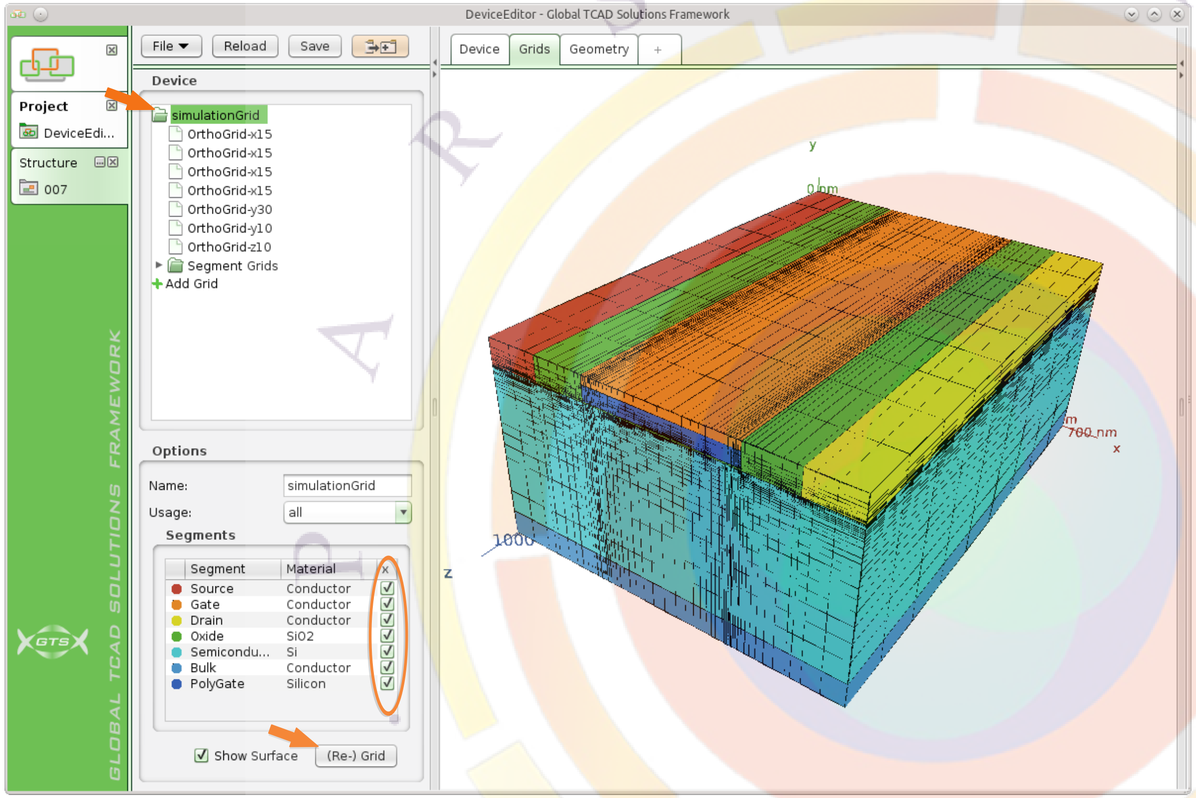
- Switch to the “Grids” page
- Right-click on “simulationGrid” and choose “Add Refinement”
- Choose “OrthoGrid1D”, with size of “Device”, and direction “z”
- Confirm by pressing “OK”



3.2.2. Adjust Refinement

To adjust the refinement, you can either use the blue and yellow handles at the device axis, or (preferably) you can directly access the parameters in the options panel below:

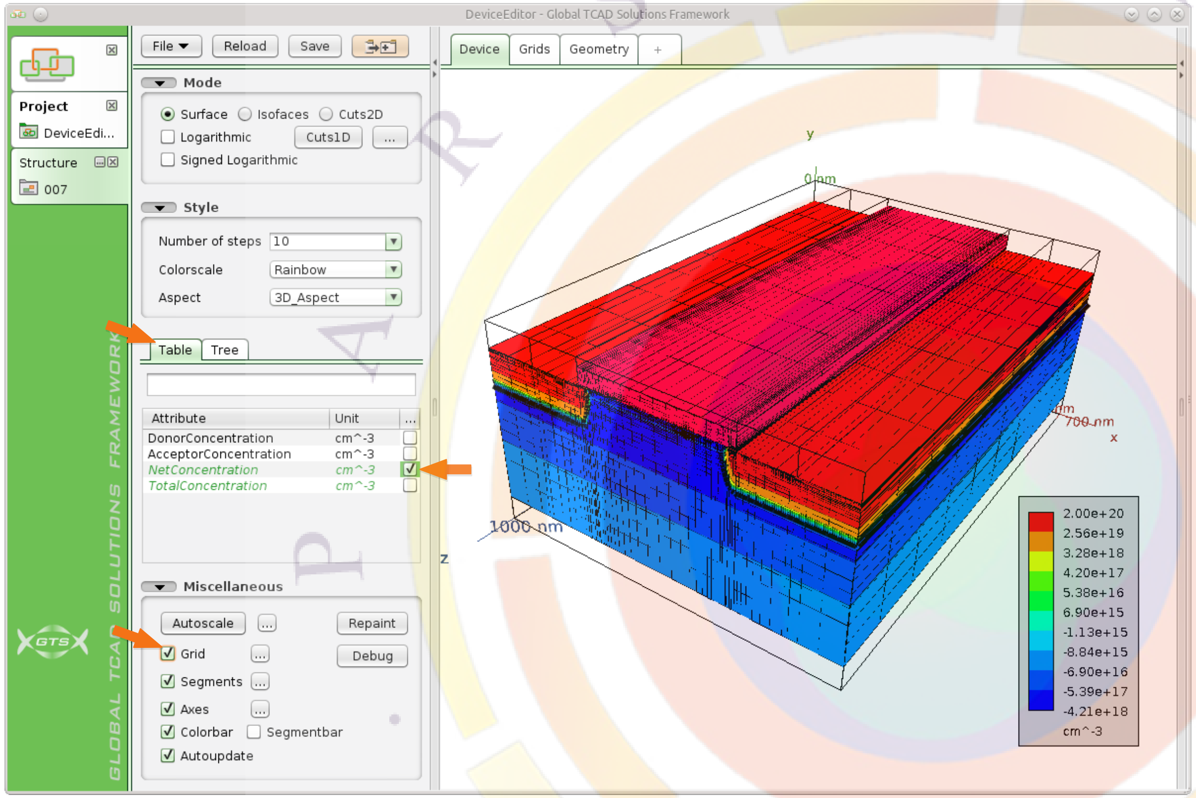
- Enter “10” in the field for the number of grid lines
- Press Enter to update the settings



3.2.3. Create Simulation Grid

Apply the changes and re-grid the device:

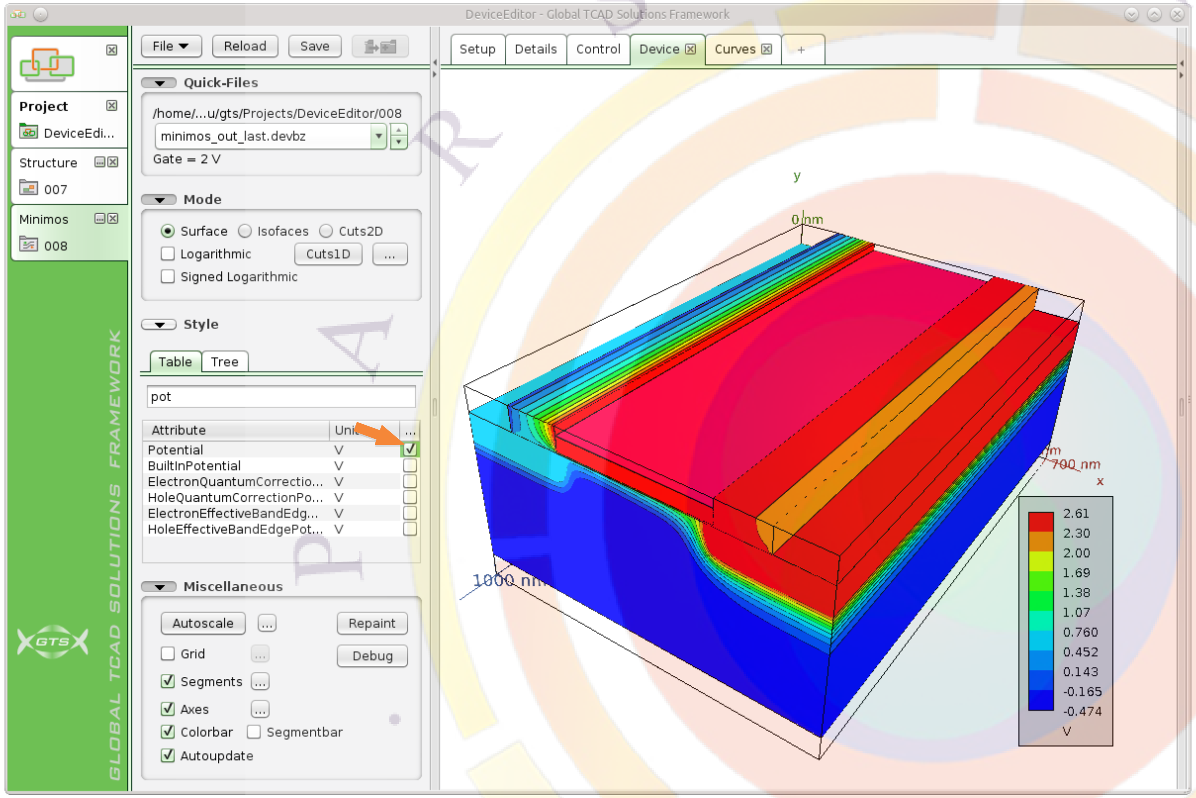
- Select “simulationGrid” in the grid tree
- Make sure that all segments are checked in the list
- Press the “(Re-) Grid” button



3.2.4. Net-Doping Profile

After creating the device geometry and the grid, we are going to check the doping profile:

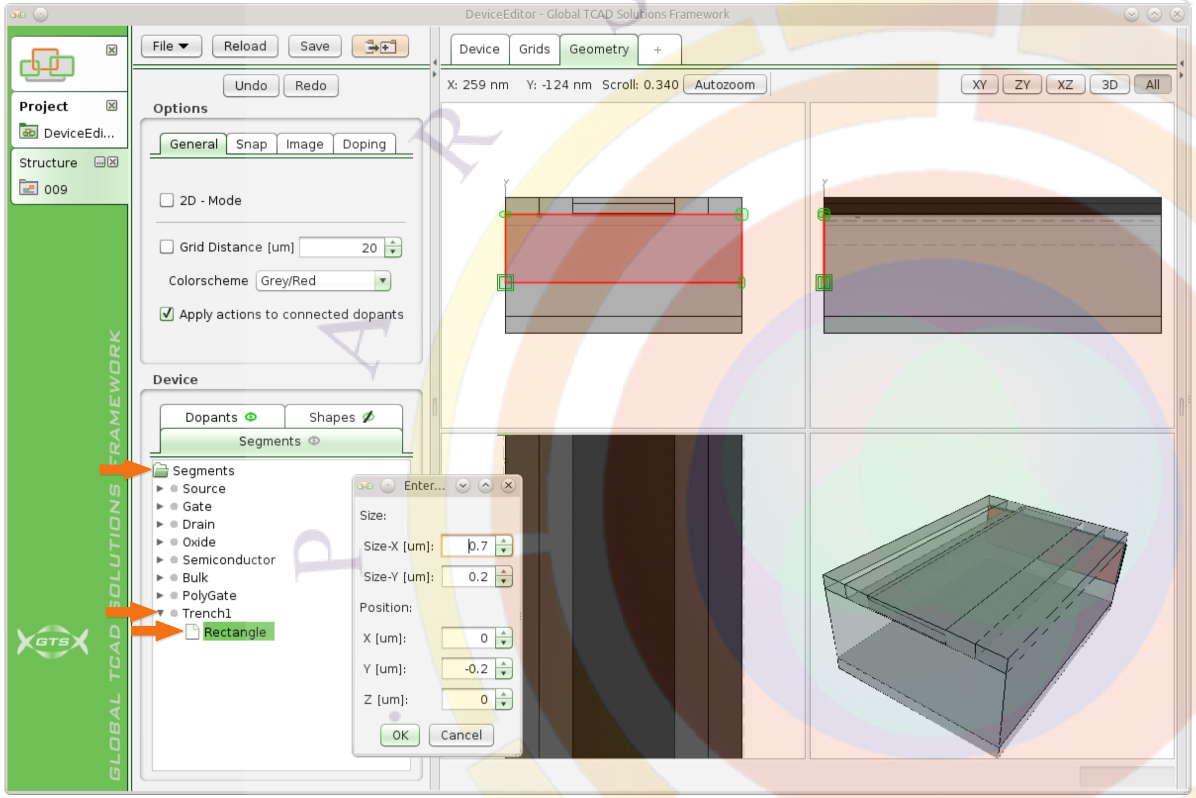
- Switch to the “Device” page
- Switch to the “Table” view
- Check “NetConcentration”
- Check “Grid” to see the grid



3.2.5. Device Simulation

We will now run a device simulation on the adapted device and copy the settings from the 2D device simulation ToolFolder:

- Select the structure ToolFolder “007” and press the “Next tool” button.
- Choose “Minimos-NT”, select “Create new ToolFolder” and “Copy settings from other Minimos ToolFolder” (compare 1.2.10)
- Select one of the Minimos ToolFolder we have used to simulate the transfer characteristic and click “OK”
- The settings are taken from the previous Minimos-NT run, so we can run the simulation in the “Control” page
- Switch to the “Device” page and select the check box of the quantity “Potential”

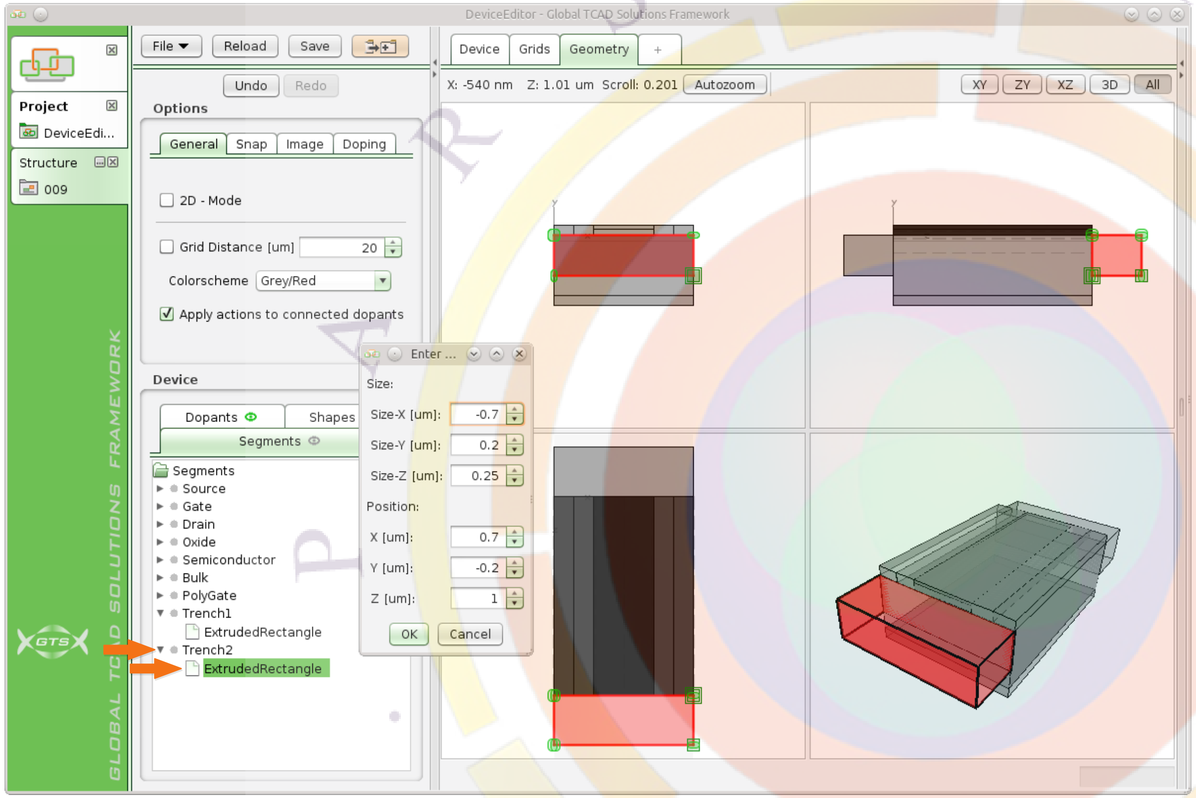


3.3. STI-MOS

3.3.1. Add Trench1

Now we want to add a SiO2 Trench to the device, we have created in the last section of this tutorial.

- Switch to the “Project” page and press the “Next tool” button of the 3D device, we have created in the last section of this tutorial
- Choose “Duplicate this ToolFolder” and press “OK” to create the new ToolFolder
- Open the new ToolFolder and switch to the “Geometry” page
- In the “Segment” section create a new segment and name it “Trench1”
- Add a rectangle to the segment “Trench1” via the context menu of the segment and “Add 2D Shape » Add Rectangle”
- Place the rectangle and adjust its size. The position and size is shown in the figure



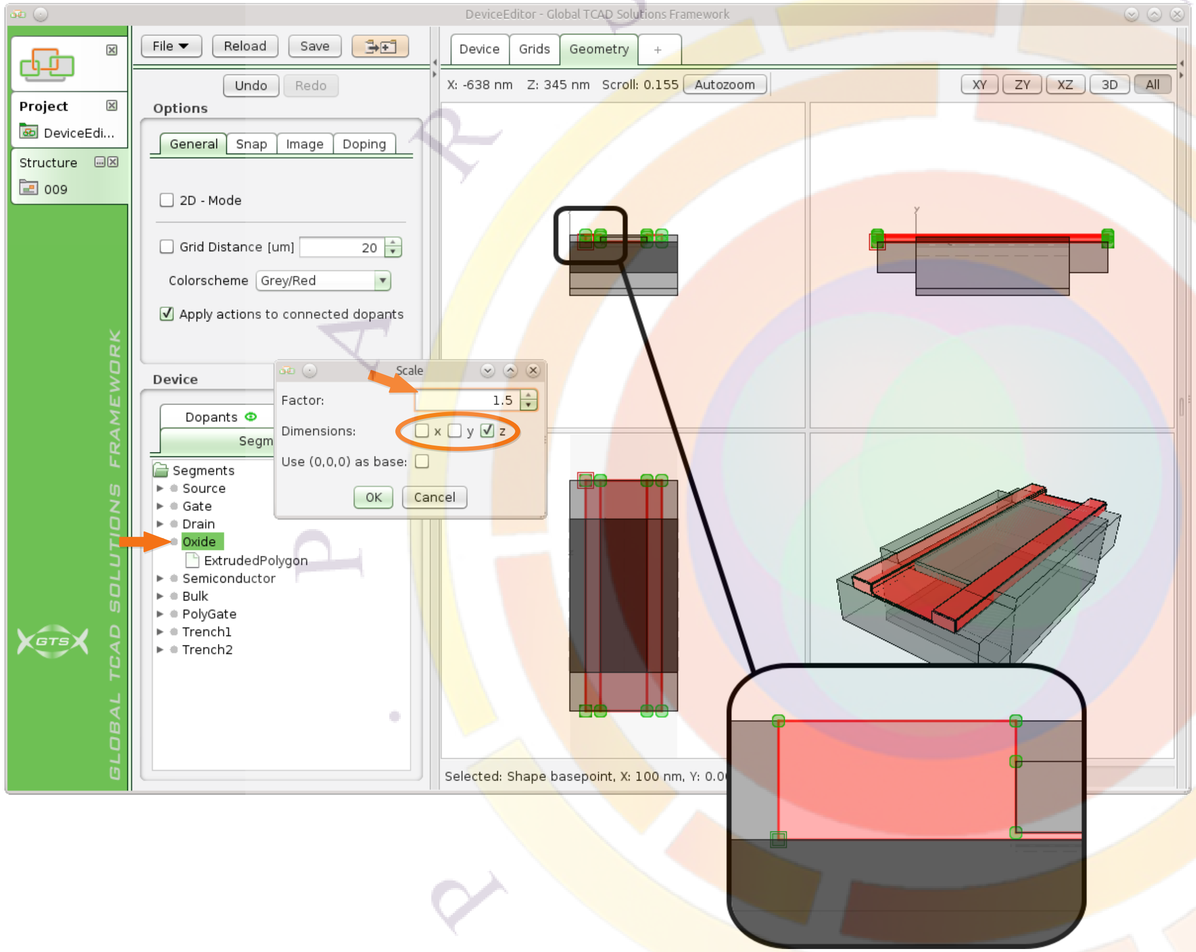
3.3.2. Finish Trench

- Select the newly generated rectangle and enter an extrude lenght of $-0.25\,\mu$m
- In order to duplicate the segment "Trench1", right-click the segment and choose "Duplicate" in the context menu
- Name the duplicated segment "Trench2"
- Place the segment "Trench2" on the opposite side of the device. Size and position of the new segment is shown in the figure

Now assign the desired material to the new segments:

- Select both segments, "Trench1" and "Trench2"
- Right click one of the selected segments and choose "Edit Material..." in the context menu
- Press "Change" and choose "Insulator » SiO2"

We use the sign of the extrude value to set the direction of the extruding.

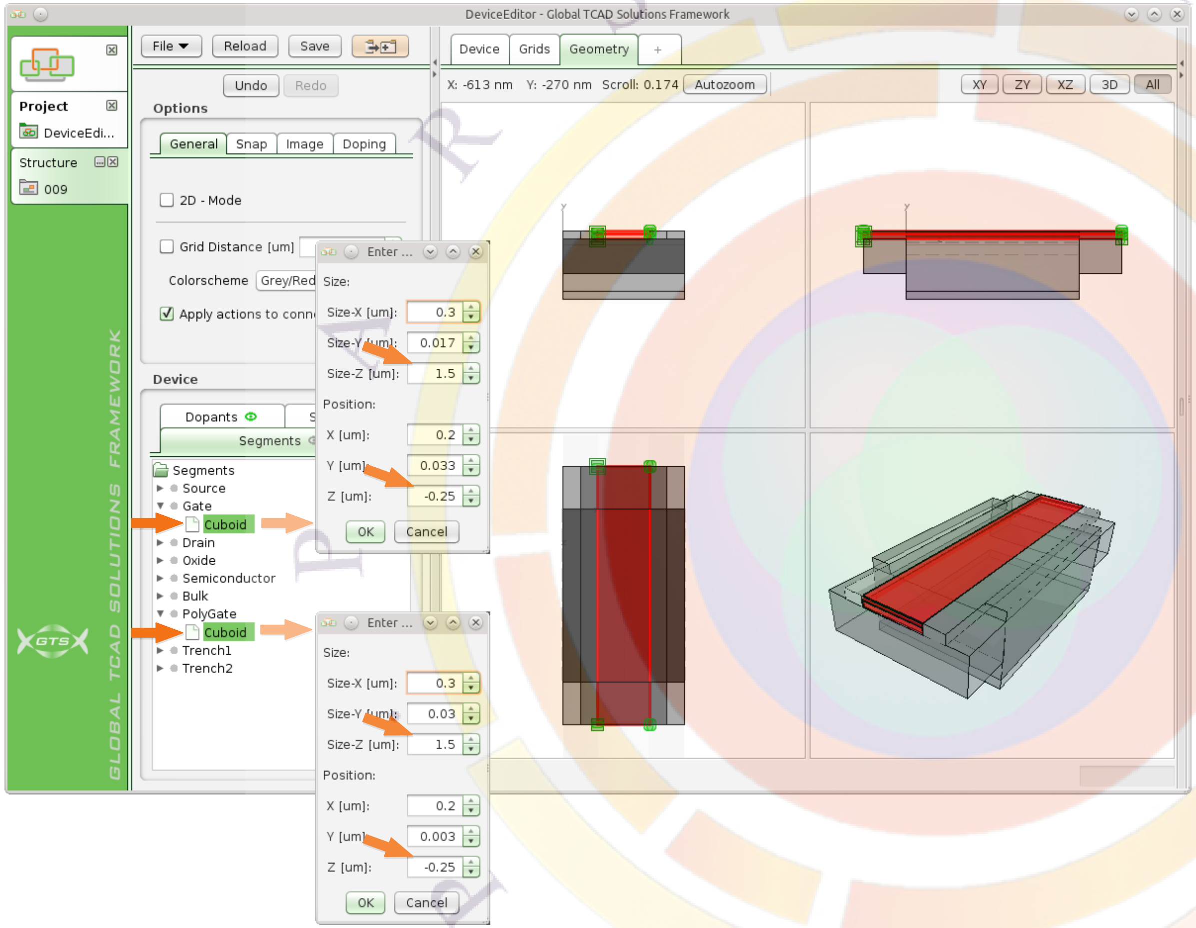


3.3.3. Extend Oxide

- In order to scale the segment "Oxide", right-click the segment and choose "Scale" in the context menu
- Enter a scale factor of 1.5, and choose only the z-direction
- Confirm the scale factor by pressing enter to see the result
- Press "OK" and move the Oxide to the position as shown in the figure. Zoom in to make sure the Oxide is placed correct

Use the mouse wheel to zoom and hold the right mouse button to shift the view.

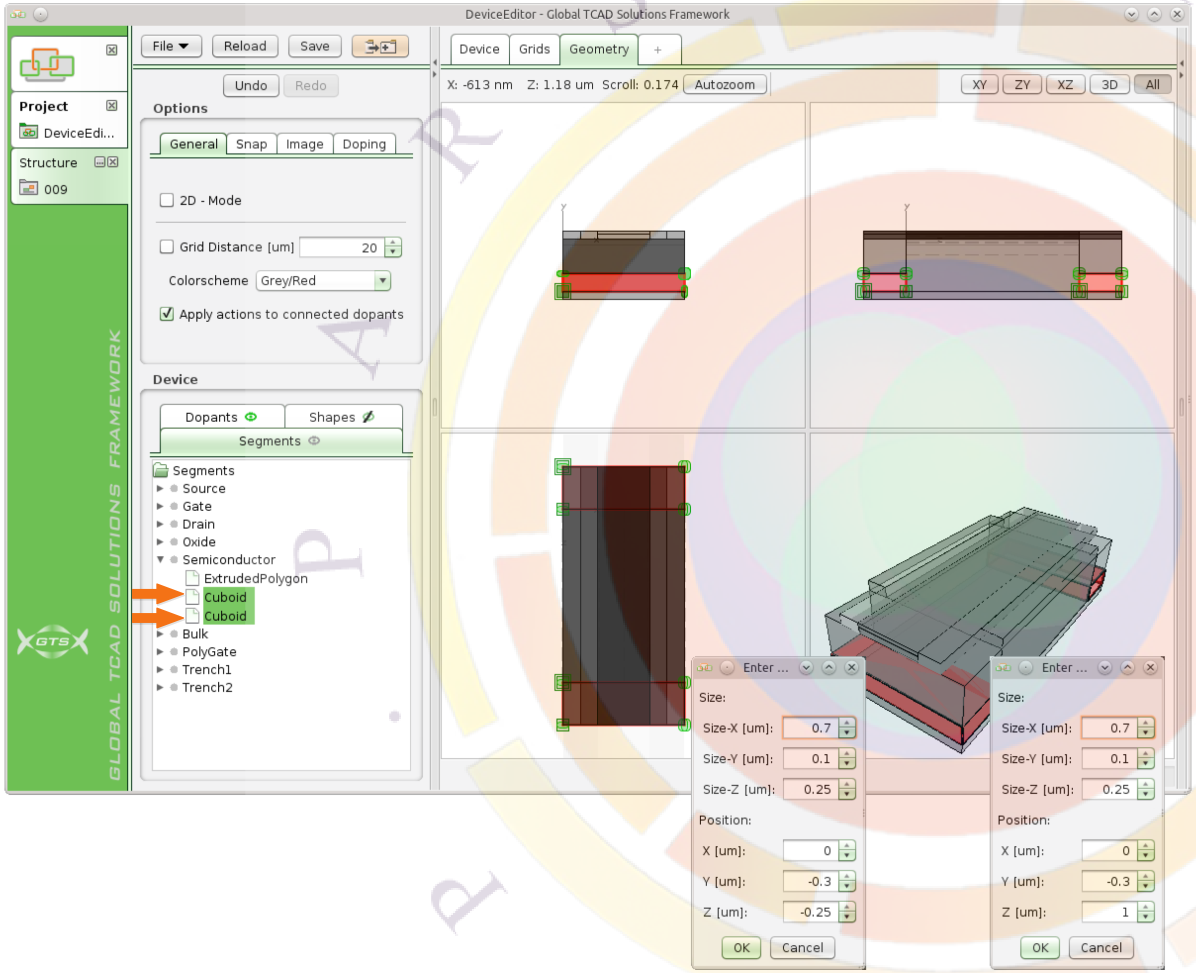
Use the "Undo" button to undo changes.



3.3.4. Extend Gate and PolyGate

- In order to extend the segment "Gate", right-click the corresponding cuboid and choose "Enter Size/Position" in the context menu
- Set "Size-Z" to $1.5\ \mu$m and the position "Z" to $-0.25\ \mu$m
- Press "OK" and extend the segment "PolyGate" the same way

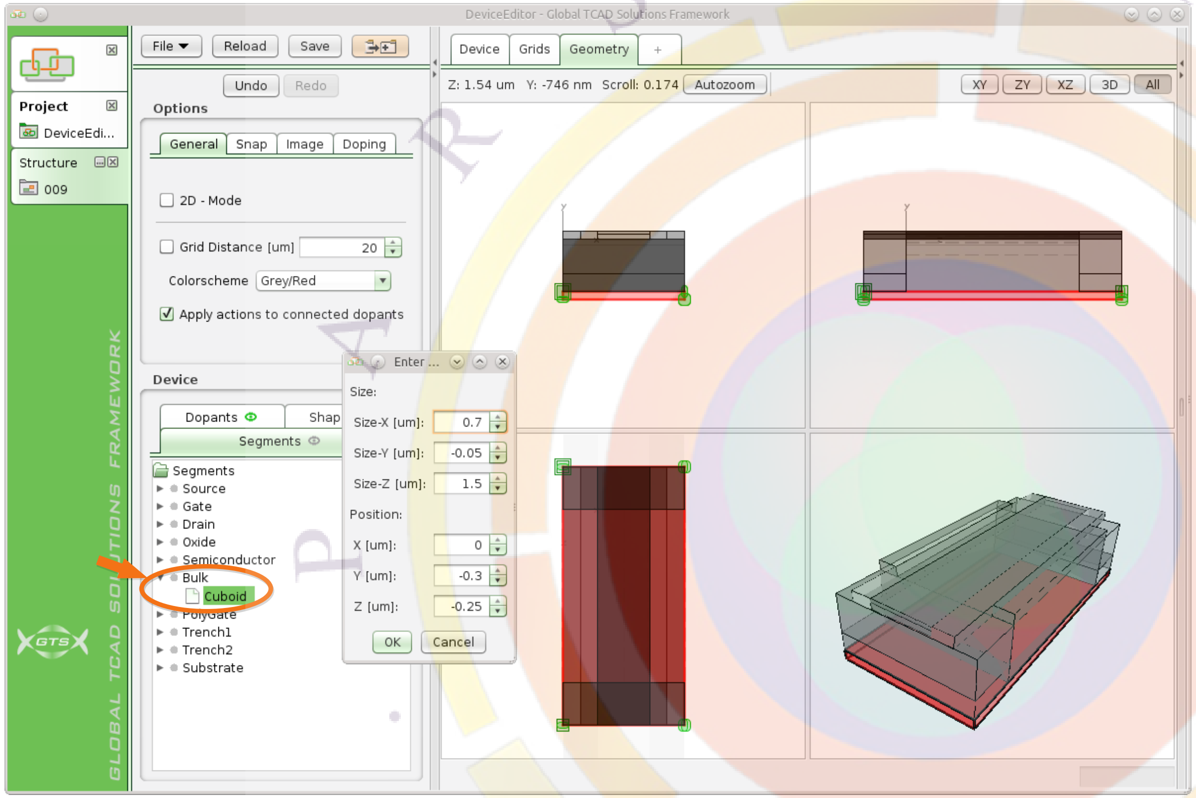
Gate and PolyGate have to be edited seperately, because otherwise their values for X- and Y-axis would become the same as well.



3.3.5. Add Substrate

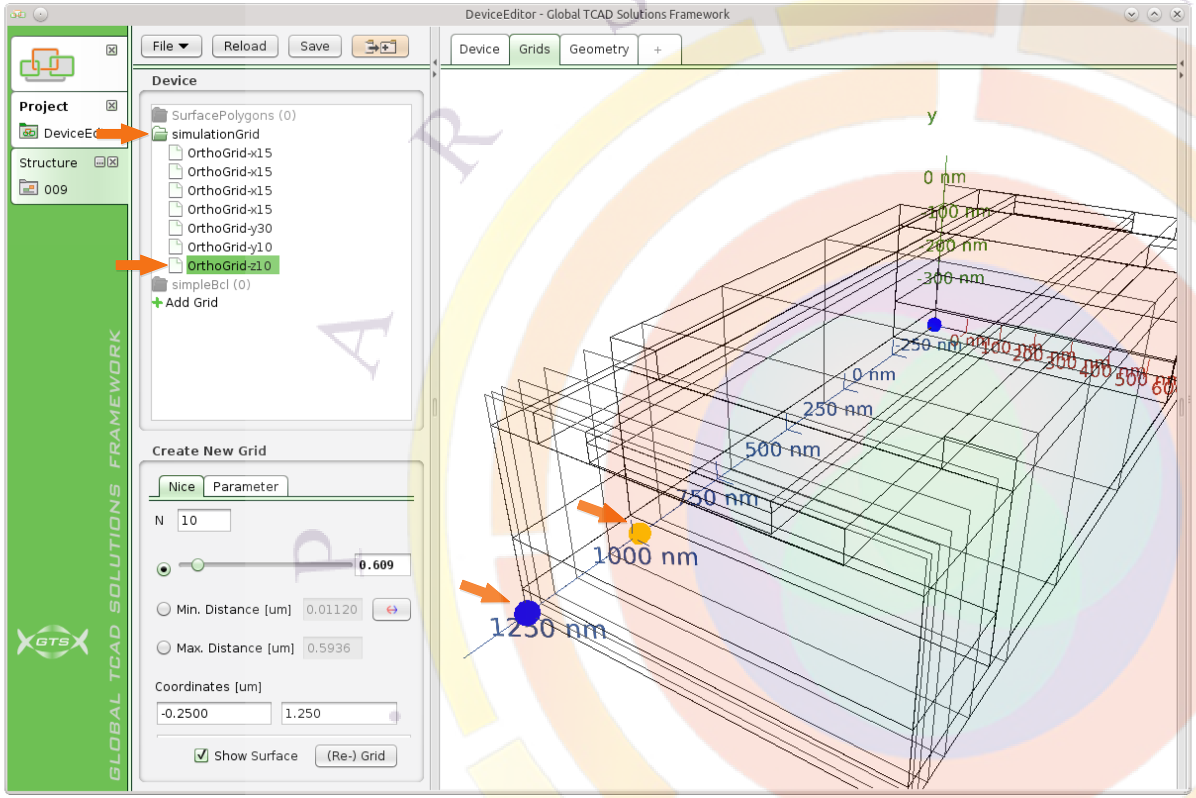
We want to add Si substrate below the trenches. This time we will generate a cube, instead of extruding an rectangle.

- In the “Segment” section right click the segment “Semiconductor” to open its context menu
- Choose “Add 3D Shape » Add Cube” to add two cubes to the segment “Semiconductor”
- Place the cubes and adjust their size. The position and size of the two new cubes are shown in the figure



3.3.6. Extend Bulk

- Select the segment “Bulk” in the “Segments” section
- Extend it to fit beneath the whole device. The size and position is shown in the figure



3.3.7. Edit Grid Using Drag Points

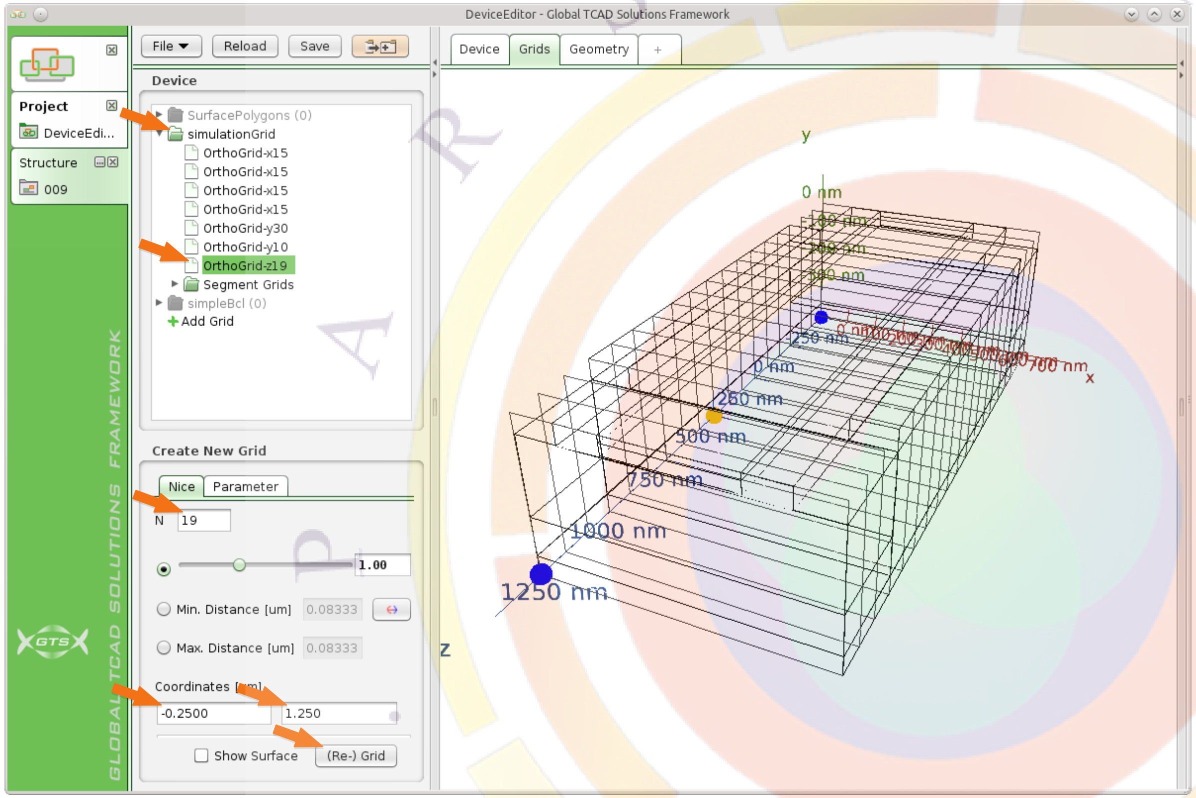
Finally we have to adapt the grid of the device. The grid can be configured with drag points.

- Switch to the "Grids" page
- Select "OrthoGrid-z10" and enter the values as shown in the figure

There are two different types of drag points to modify the grid:

- Drag the blue points to define the start and the end of the grid
- Drag the yellow point to change the grid interval

Move the drag points to get familiar with them. In the next step we will edit the values in the control panel.

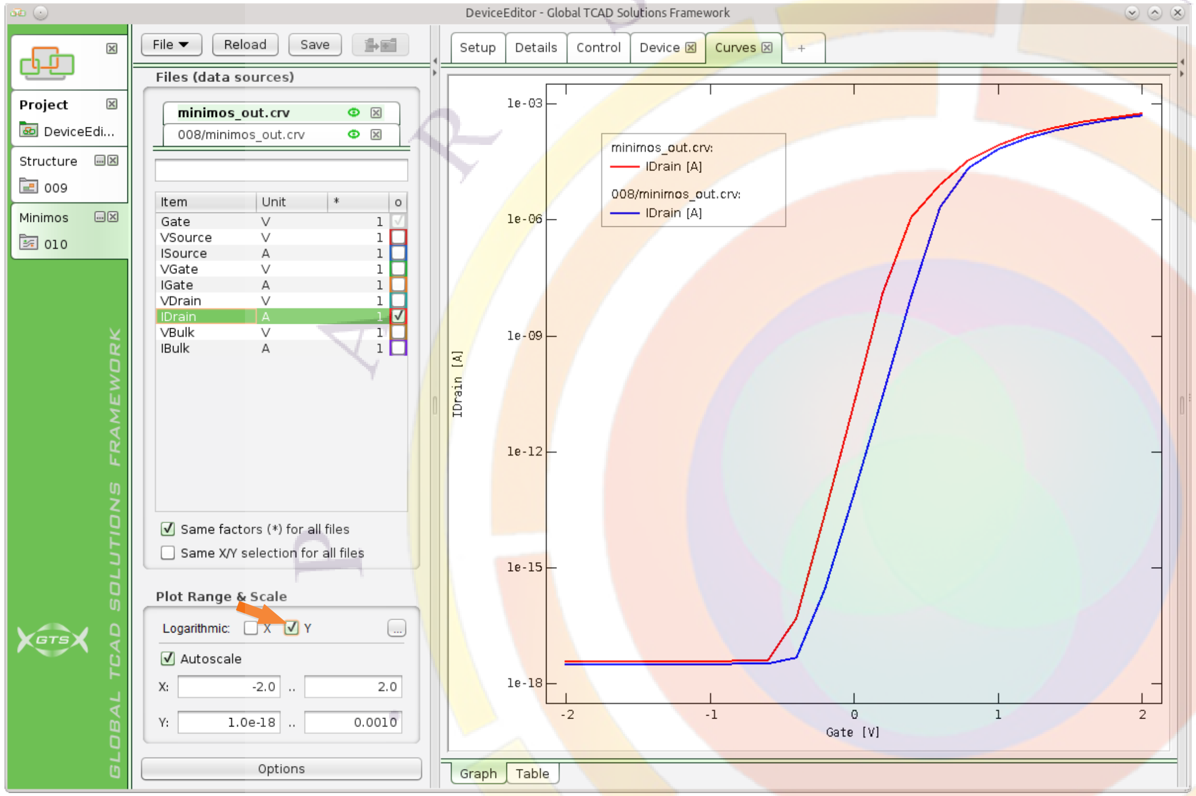


3.3.8. Edit Grid

Now enter the desired values in the control panel and re-grid:

- Enter the values as shown in the figure
- Select the all segments in the "Segments" section and press the "(Re-) Grid" button (compare 2.3.3)

Always confirm with the enter key, after editing a value.



3.3.9. Compare 3D Results

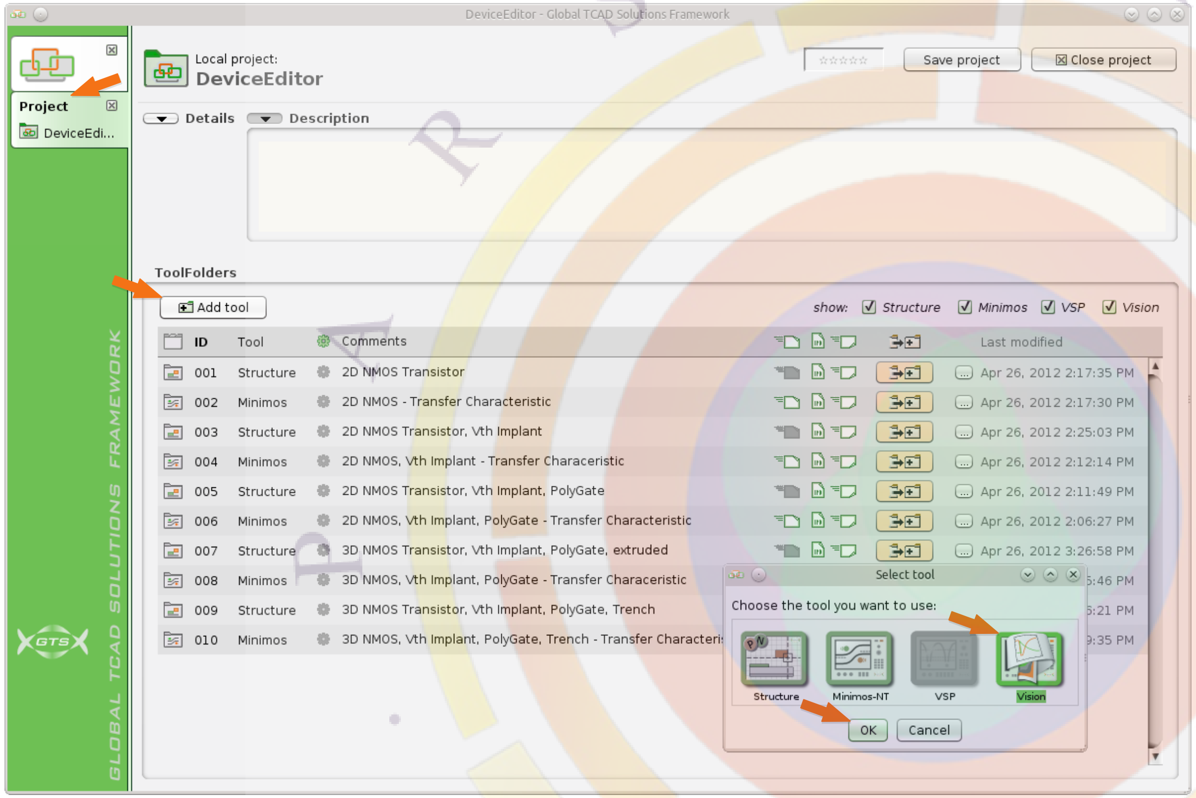
We will now run a device simulation on the adapted device and copy the settings from the 3D device simulation ToolFolder:

- Press the “Next tool” button, choose “Minimos-NT”, select “Create new ToolFolder” and “Copy settings from other Minimos ToolFolder” (compare 1.2.10)
- Select the Minimos ToolFolder we have used to simulate the transfer characteristic of the 3D device without a trench and click “OK”
- The settings are taken from the previous Minimos-NT run, so we can run the simulation in the “Control” page
- Switch to the “Curves” page and display the transfer characteristic. Add the output of the simulation of the 3D nMOS without the SiO2 trench (compare 1.2.11)

Part 4

Comparing Results

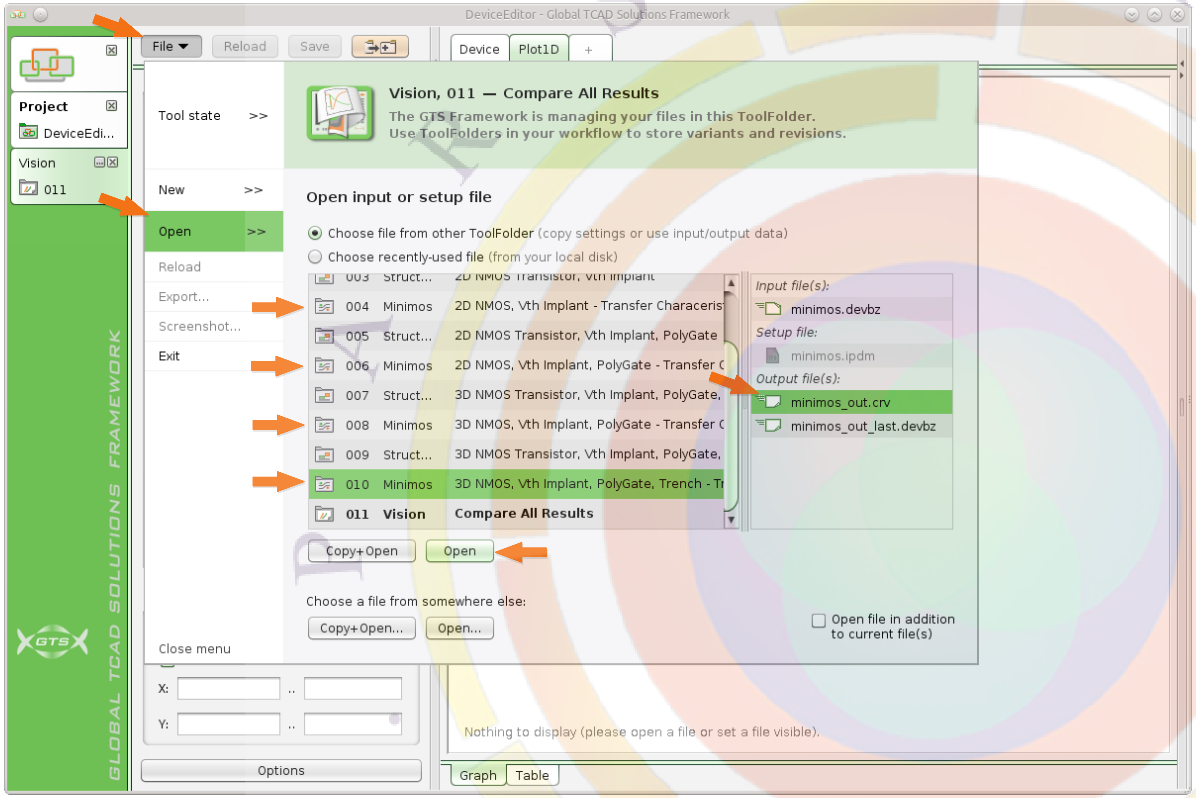
The preferable way of comparing results from various simulations is to add a new GTS Vision tool. In this chapter, we will use a GTS Vision tool to compare the results of all previous simulations.



4.0.1. Add GTS Vision Tool

- Switch to the "Project" page
- Press the "Add tool" button
- Select the tool "Vision"
- Press "OK" to create the new ToolFolder

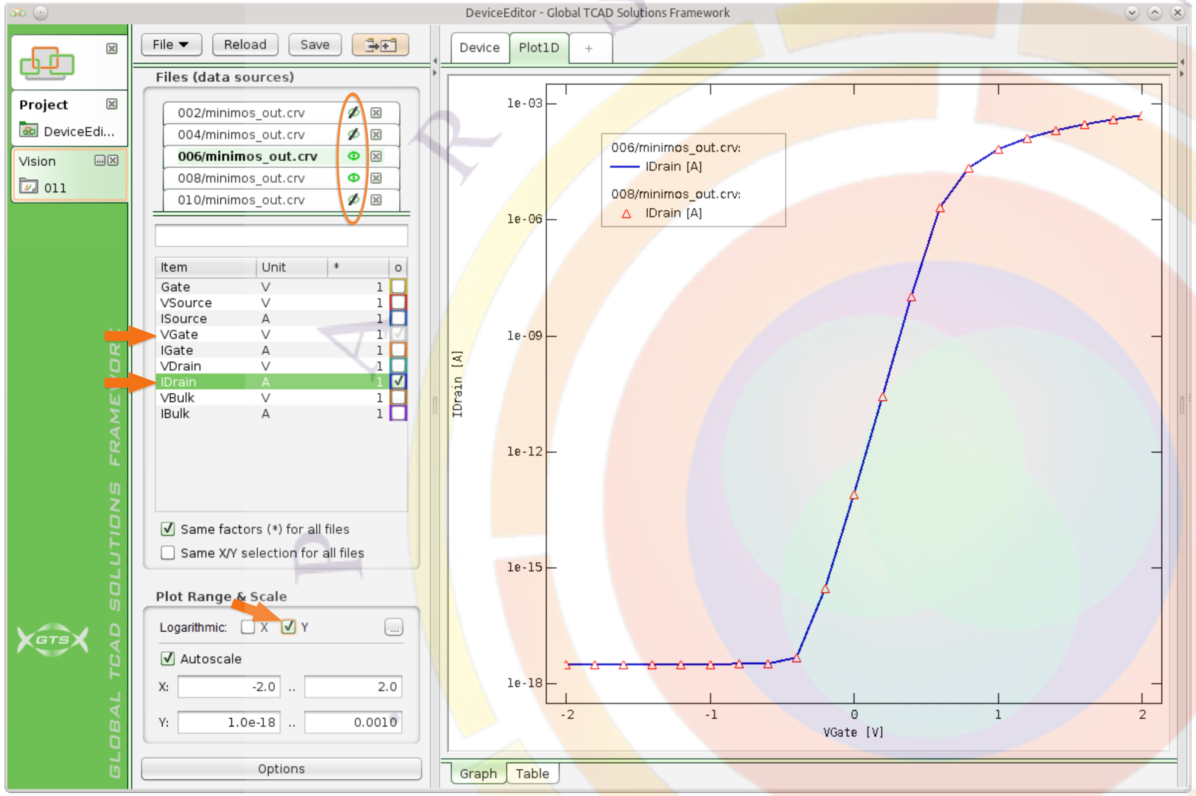
The GTS Vision ToolFolder does not automatically store any files or settings. Make sure to export your visualizations before closing.



4.0.2. Open Files to Compare

In GTS Vision, you can easily open and compare files from different ToolFolders:

- Switch to the “Curves” page
- Select the menu “File » Open”
- Select “Choose File from other ToolFolder”
- In the ToolFolder list choose one of the Minimos folders
- On the right-hand side, select the file `minimos_out.crv`
- Make sure “Open Files in addition to any current files” is checked
- Press “Open”
- Repeat the steps to open all output files from the 2D and 3D simulation

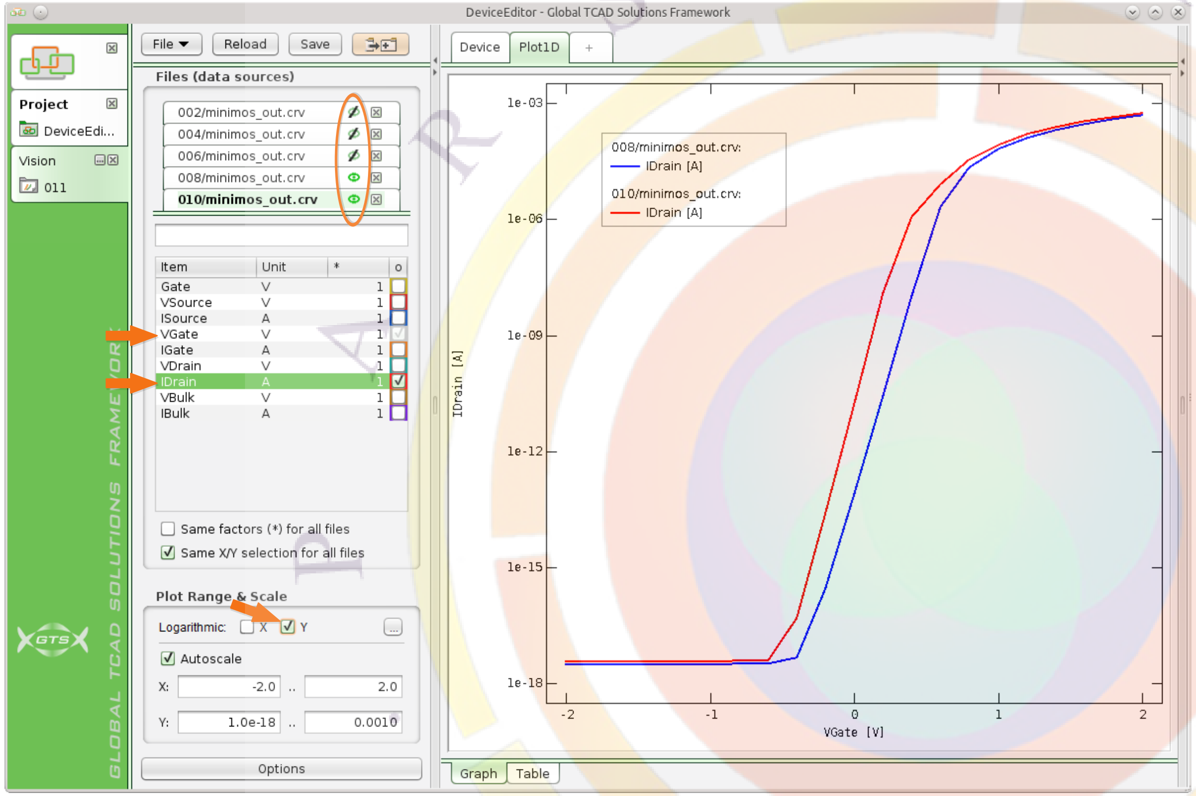


4.0.3. Compare Output Curves: 2D, 3D

All output files are now loaded. To display the transfer characteristic configure the output as follows:

- Click on the table header above the rightmost column to deselect all quantities
- Select the quantity "VGate"; right-click to open the context menu and choose "Set as X axis"
- Check the box beside "IDrain" to show the drain current over the gate voltage
- Set the "Plot Range & Scale" to "Logarithmic Y"
- Use the "Visibility" button, next to the file names to select which curves you want to display

In the figure we compare the results of the 2D device with the 3D device, both with Poly Gate and Vth implant. They yield the same result.



4.0.4. Compare Output Curves: 3D Trench

Finally we compare the results of the 3D device, with and without the SiO2 Trench. Few changes to the settings have do be done, to obtain the graph, shown in the figure.

Choose "File » Screenshot. . . " to save the plot as an image file.

Conclusion

In this tutorial, the 2D and 3D device editing capabilities of GTS Structure have been introduced. First, an nMOS transistor, created from a template, has been modified. A threshold voltage doping has been added and the Gate contact has been changed to a PolySi gate.
In a further step, the 2D device was extruded to a 3D device. This new 3D device was modified further to simulate an STI-MOS transistor. Finally, the results of the various 2D and 3D devices have been compared.

Further Reading

We welcome you to have a look at further *GTS tutorials* and *examples*, which you can open in ***GTS Framework***. Next to the basic ones included with the release, you can download more sophisticated tutorials and examples from MyGTS at `https://globaltcad.com/mygts`. Extracting the archives to your projects folder makes the tutorials visible in the projects list (highlighted yellow). Previews are provided at `http://www.globaltcad.com/en/solutions.html`.

For additional information, please refer to `http://globaltcad.com/` or feel free to contact us at `info@globaltcad.com`.

Appendix A

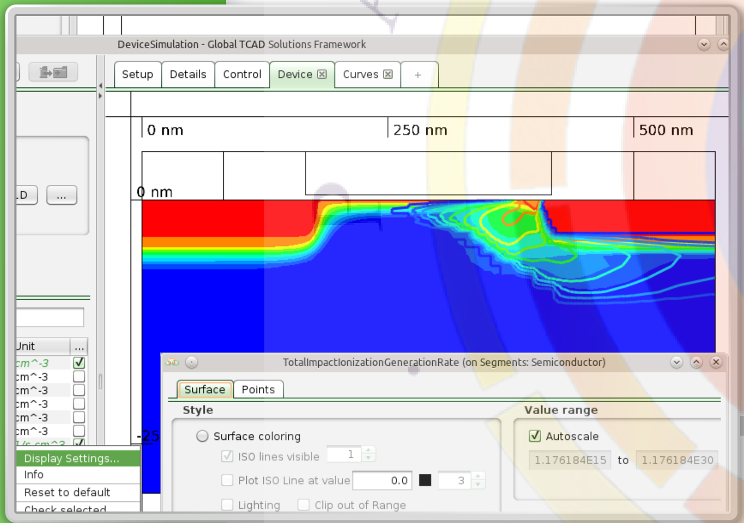
ToolFolder List

The project **DeviceEditor** contains the following ToolFolders (TF):

TF	Tool	Description
T01	Structure	2D NMOS Transistor
T02	Minimos	2D NMOS - Transfer Characteristic
T03	Structure	2D NMOS Transistor, Vth Implant
T04	Minimos	2D NMOS, Vth Implant - Transfer Characeristic
T05	Structure	2D NMOS Transistor, Vth Implant, PolyGate
T06	Minimos	2D NMOS, Vth Implant, PolyGate - Transfer Characteristic
T07	Structure	3D NMOS Transistor, Vth Implant, PolyGate, extruded
T08	Minimos	3D NMOS, Vth Implant, PolyGate - Transfer Characeristic
T09	Structure	3D NMOS Transistor, Vth Implant, PolyGate, Trench
T10	Minimos	3D NMOS Vth Implant, PolyGate, Trench - Transfer Characteristic
T11	Vision	Compare All Results

GLOBAL TCAD SOLUTIONS — GTS FRAMEWORK — TUTORIAL

GTS FRAMEWORK



Device Simulation

Tutorial

global TCAD solutions®

Device Simulation – Tutorial, GTS Framework Release 2016.09
Revision of March 25, 2017

Global TCAD Solutions GmbH
Bösendorferstraße 1/12
1010 Wien
Austria / Europe

Phone: +43 1 9255049
Email: info@globaltcad.com

Additional information is provided at: http://www.globaltcad.com/

Cover & layout: prausedesign

Contents

Introduction

Using This Tutorial

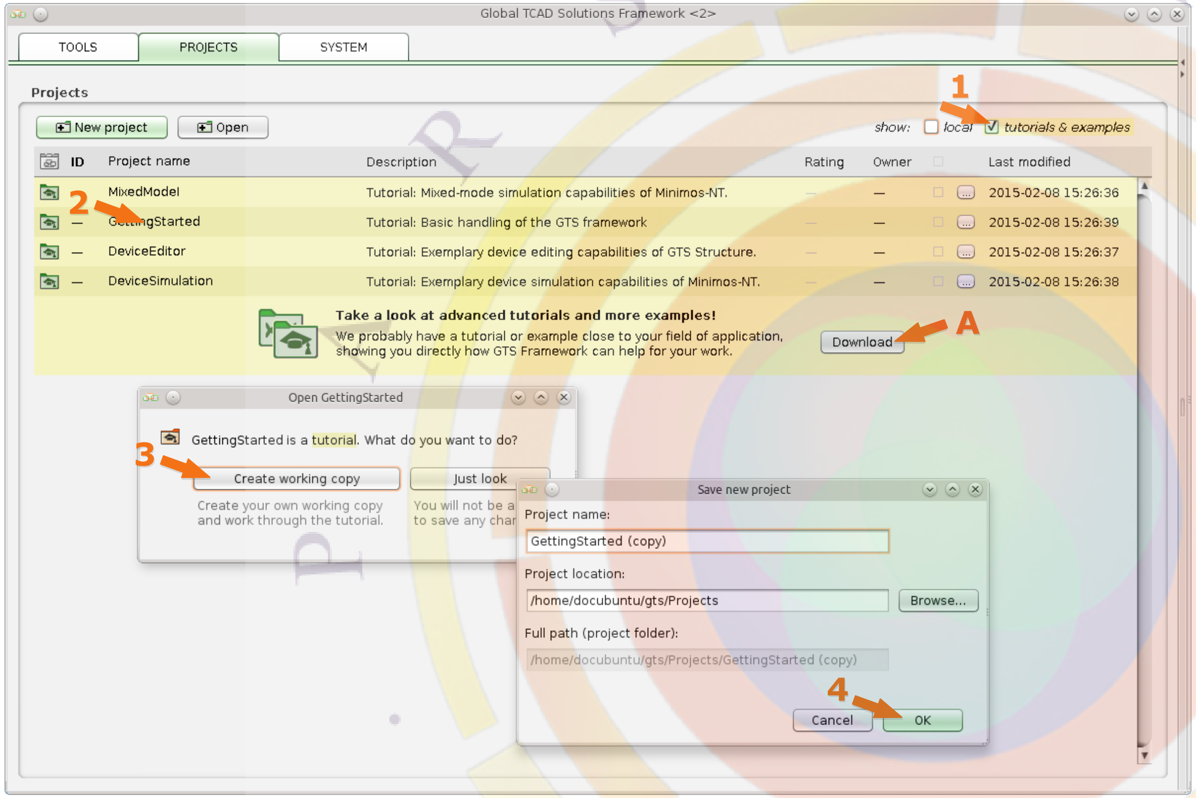
GTS *tutorials* generally consist of a written description (this text) and a GTS *project* (simulation data) which you can open in ***GTS Framework*** (yellow items in project list). As shown below, we recommend to create a working copy from the project, and proceed step by step, guided by this text. If not yet familiar, please refer to the *GettingStarted* tutorial.

Working in a copy of a tutorial project, you can open the yellow Txx ToolFolders at any time. They contain the data of the described simulations, which you can use for reference or as starting points for your own simulations.

Only a few tutorials are included with the release; the others are available via *MyGTS* at `https://globaltcad.com/mygts`. When logged in, please download the respective file and extract it to the gts projects folder on your PC. After restarting GTS Framework, the tutorial project will appear in the *Projects* list.

This is an introductory (level 1) tutorial.

This tutorial was created using GTS Framework Release 2016.09. Other releases might need adjustments or have slightly different user interfaces.



Project / Working Copy

If you have not already done so, go to the *Framework Home* and locate the project associated with this tutorial in the *Projects* list, and create your working copy from it:

1. Check “tutorials & examples”
2. Click on the respective tutorial project: ***Device Simulation***
3. Choose “Create working copy”
4. Check the project name, click “OK”

The project is created and opened, so that you see the *Project Home*. — *Ready to start!*

A. If you miss the respective project, please download it via *MyGTS* (click “Download”, see previous page).

Alternatively, you can start the tutorial with an empty project – just create a new project. (The advantage of the tutorial project is that you have the results at hand any time.)

Part 1

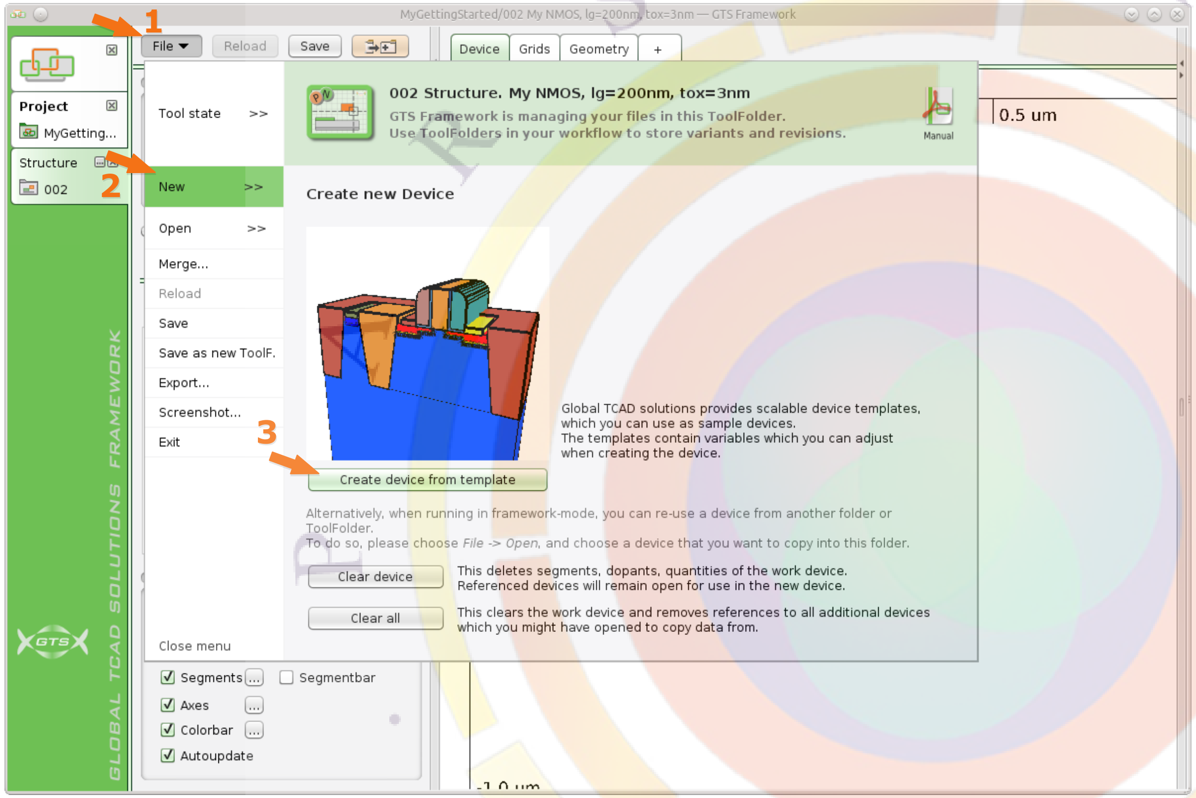
Steady State Simulation

In this example, we will set up steady state simulations. In order to gain convergence performance, we will show how results from one simulation can be used to initialize another one. The imported values will be used as initial guess for the Newton solver allowing fast and reliable convergence.

This feature is very convenient for large devices with a high number of mesh points, when simulations start at or near already calculated operating points. It is also very useful when encountering poor convergency behavior.

Furthermore we will implement quantum tunneling through the oxide and take a look at the resulting Gate currents for different Drain voltages.

It is assumed that you are already familiar with the main concepts of GTS Framework and the idea of projects and ToolFolders, as explained in the "Getting Started" tutorial.

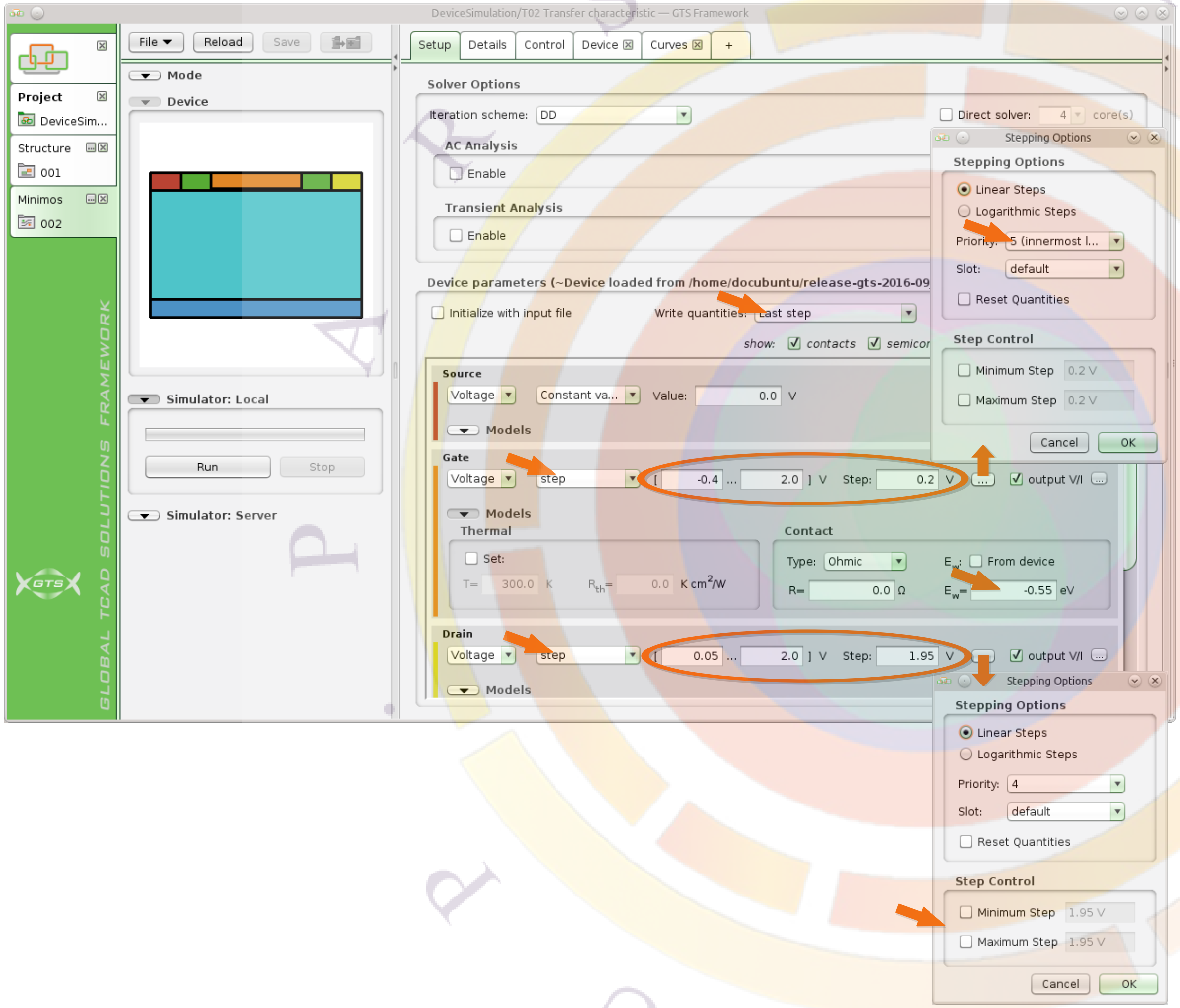


1.1. Transfer Characteristics

1.1.1. Device

- In your working copy of the *Device Simulation* tutorial project, add a “Structure” ToolFolder
- Click *File, New, Create Device from Template*
- Choose the nMOS device template `nmos_0250um_2d_ortho.ipd` and create the device
- Click the NextTool button, and Yes to save
- In the NextTool dialog, choose *Minimos*

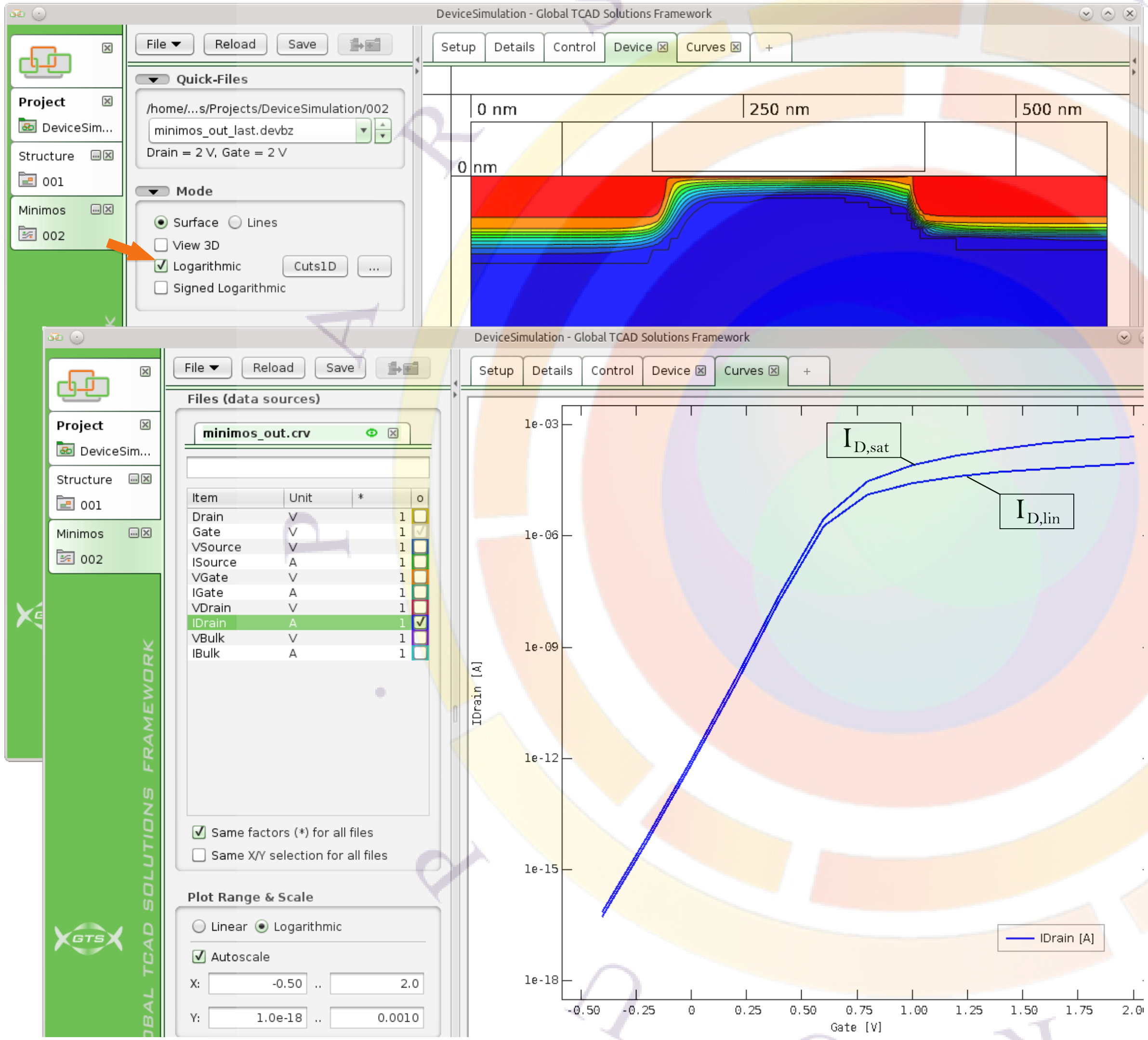
For details, please see the *Getting Started* tutorial.



1.1.2. Setup Idlin, Idsat

- Set up the Minimos-NT tool to simulate transfer characteristics (see figure for settings and voltage ranges)
- Make sure to set “Write quantities” to “Last step”. This last step at maximum Gate and Drain voltage will be used to initialize the next simulation.

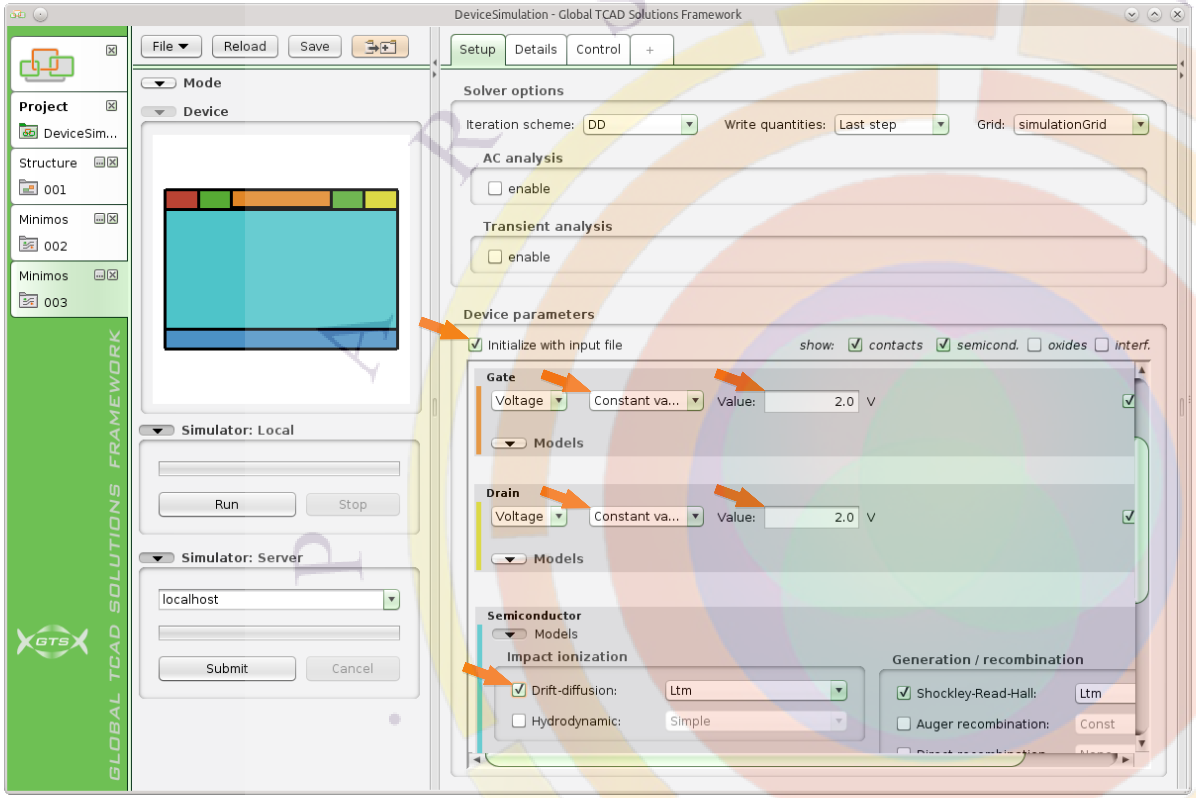
The Drain voltage stepping is set up to run at 0.05 V to simulate in linear region and at 2 V to simulate in saturation region.



1.1.3. Results

- Switch to the “Control” page and run the simulation
- Check the distributed quantities of the last simulation step in the “Device” page
- The upper part of the figure shows the distribution of the electron concentration in a logarithmic scale
- The lower part of the figure shows the settings to display the transfer characteristic in the “Curves” page

Right click the legend to hide it, or to move it to a desired location. If no legend is displayed one can access it by right clicking in the area of the graph.

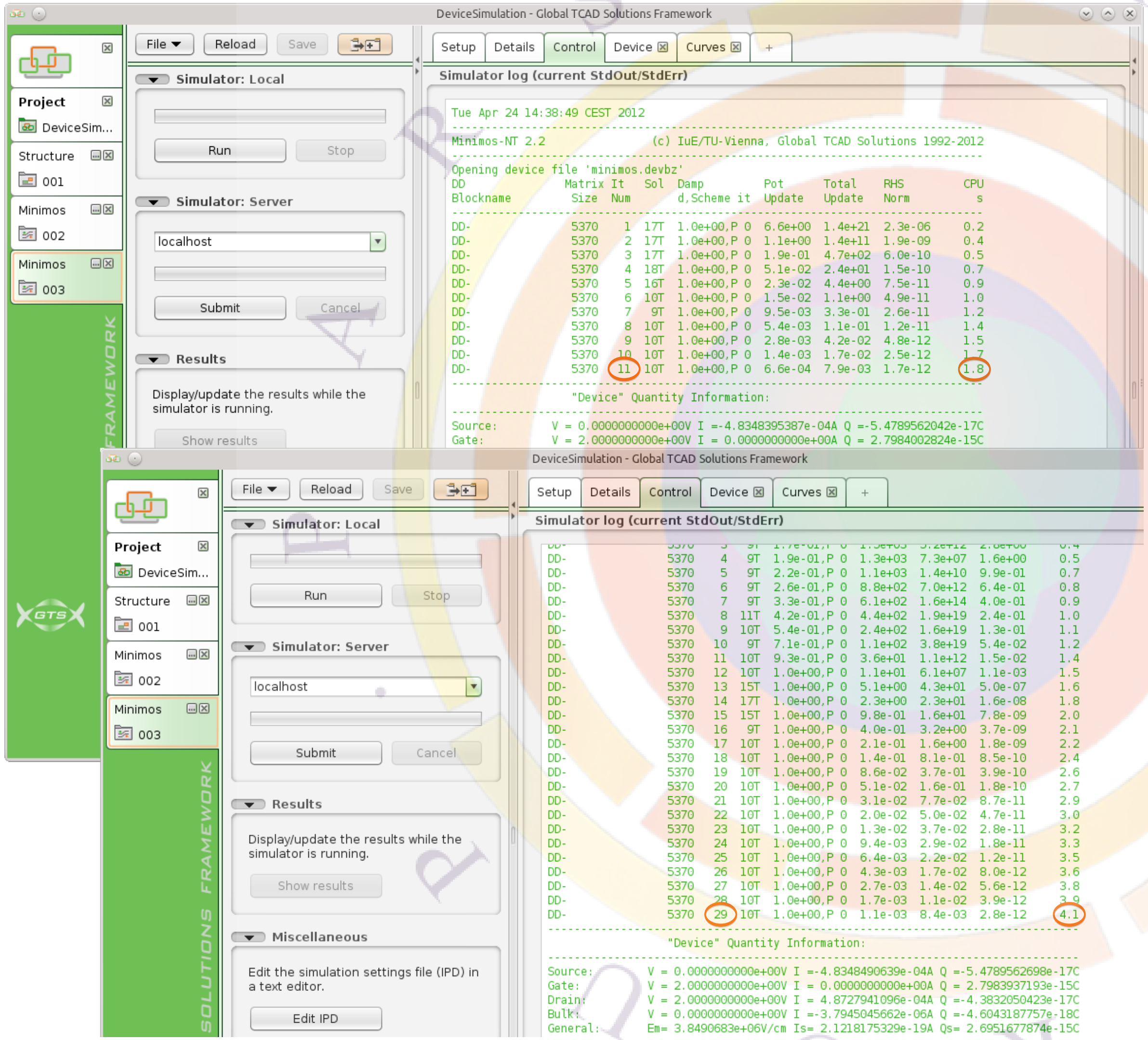


1.2. Impact Ionization

1.2.1. Initialize With Input File

For analyzing the changes in the device if the impact-ionization model is included, a new Minimos-NT ToolFolder will be added. The simulation will be performed at the Drain and Gate voltages from the final step in the previous simulation. To optimize simulation time and convergency, the results of the previous simulation will be used to initialize the new simulation:

- Create a new Minimos-NT ToolFolder using the “Next tool” button of the previous simulation (and not of the GTS Structure tool)
- Select “Create new ToolFolder”, “Copy settings from ...”, and select the previous Minimos-NT simulation run, which is ToolFolder “002” in this tutorial, as copy source
- Activate the “Initialize with input file”
- Change the rest according to the figure

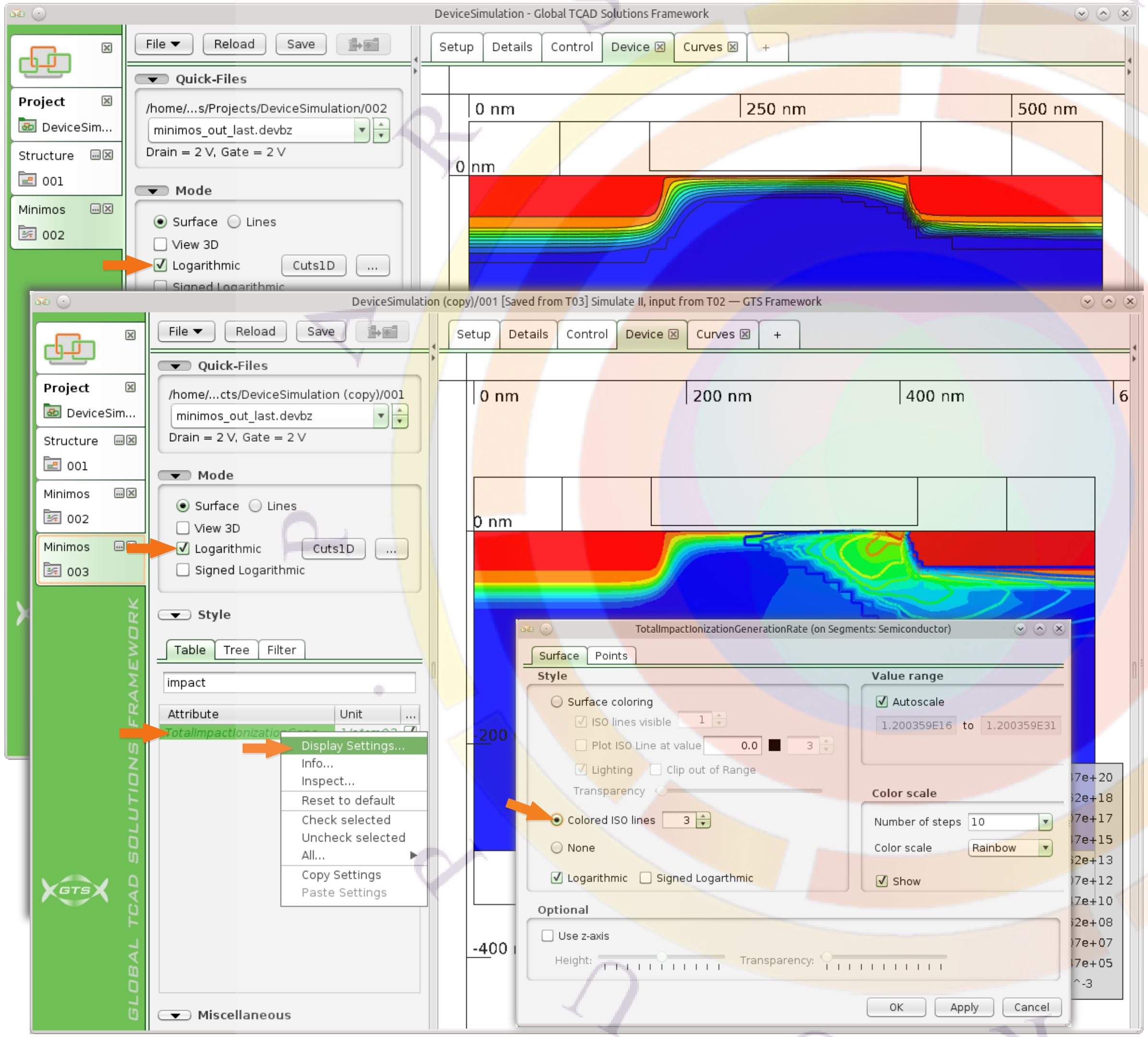


1.2.2. Analyze Performance Gain

Starting the simulation with the given options leads to the output in the upper half of the figure. For a comparison, rerun the simulation without the check-box “Initialize with input file” and the simulator output will be similar to the lower half of this figure.

The absolute performance gain in the example is low, but it becomes substantial in complex structures.

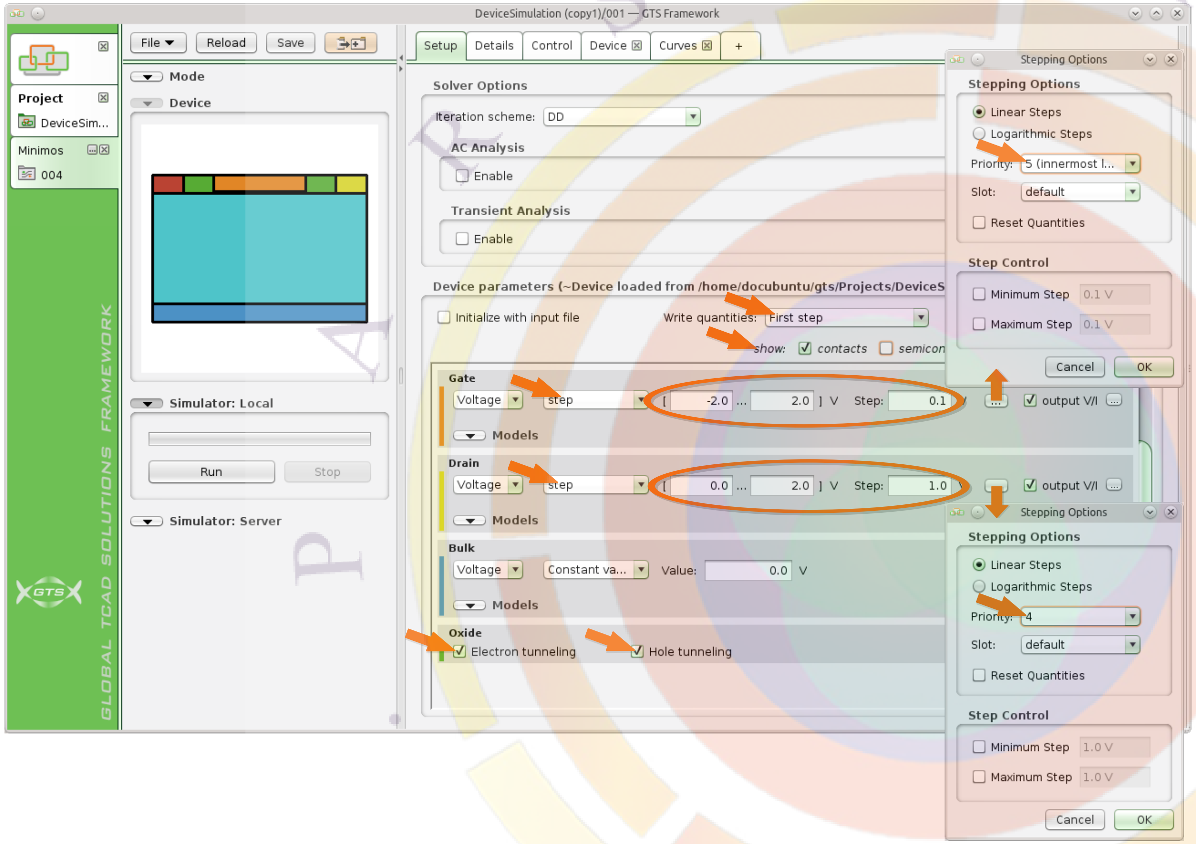
In complex structures it might be hard to directly simulate at boundary conditions and contact potentials far away from equilibrium. Preceding steppings and the use of initialization procedures are required to improve convergency.



1.2.3. Results

Compare the simulation results with and without impact-ionization. The Drain current increases by 10% as can be seen in the table of the output quantities located on the "Curves" page.

The figure shows two screenshots, the upper one without and the lower one with impact-ionization. In both, the surface color shows the electron concentration. In the lower one, additionally the iso-lines of the impact-ionization generation rate are shown. The settings for this can be found using the "Display Settings" command from the context menu of the respective quantity. There, you can also switch off the ISO lines, as was done for the electron concentration. All scales are logarithmic.

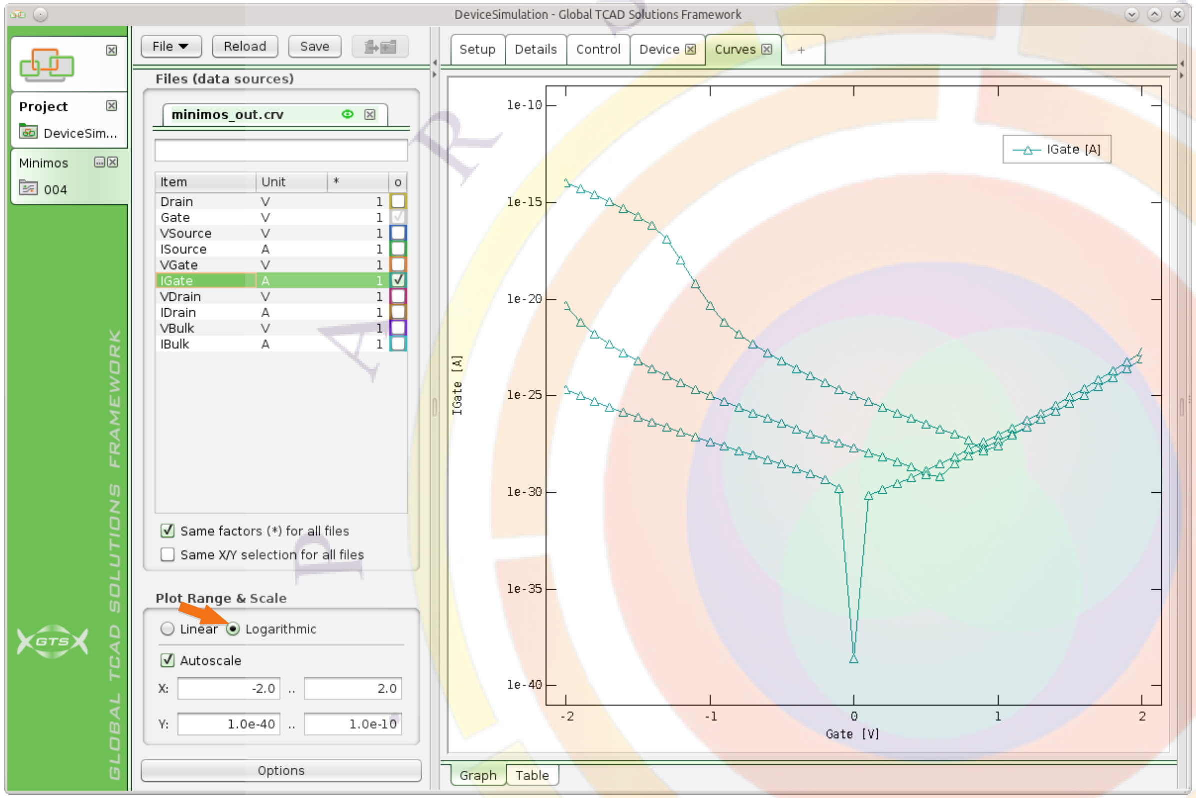


1.3. Oxide Tunneling

1.3.1. Setup Quantum Tunneling

Set the voltage ranges and activate tunneling in a new Minimos-NT ToolFolder:

- Create a new Minimos-NT ToolFolder by using the "Next Tool" button of the original structure, created in this tutorial
- In the "Device Parameters" panel, check the "oxides" filter to display the Oxide segment. For the Oxide segment, activate tunneling for both electrons and holes
- Set Gate and Drain voltages as shown in the figure
- Assign -0.55 eV to the work function difference E_w of the Gate contact (relative to the substrate)
- Finally make sure the stepping priority for the Gate is higher than for the Drain. To do so, check the priority in the respective Stepping Options via the "..." button of Gate and Drain
- Run the simulation



1.3.2. Results

Switch to the “Curves” page and inspect the Gate current, depending on the Gate voltage. Three curves are plotted, corresponding to the different Drain voltages 0 V, 1 V and 2 V.

To check the exact values for different Drain voltages one can switch to the “Table” section (instead of “Graph”) and add the Drain voltage to be plotted.

A higher tunneling current, especially for $V_\mathrm{d} = 2$ V in the region of negative Gate voltage, occurs due to accumulation of electrons close to the Drain area.

Part 2

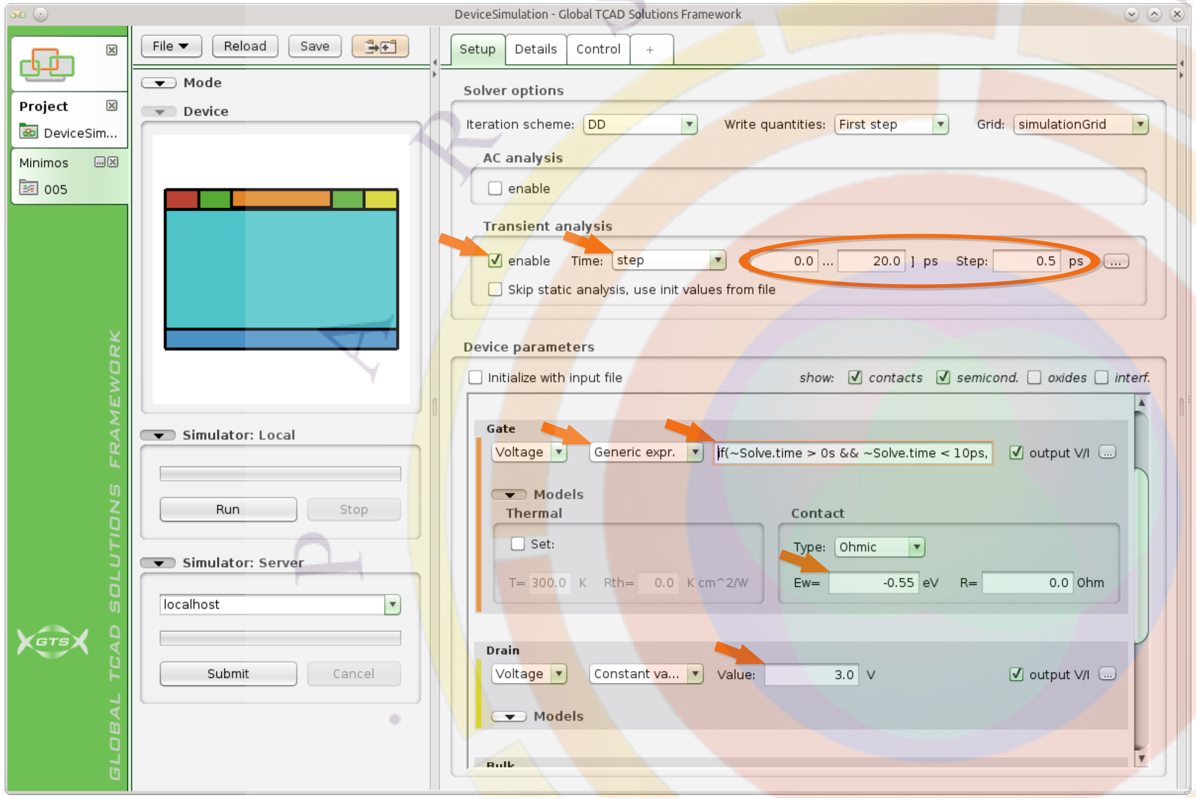
Transient Simulation Mode

In the transient simulation mode, the semiconductor equations are solved including the time derivatives. In the following some switching examples will highlight how this mode can be applied.

In the transient mode, the input values at time zero are solved as a common static solution, for times greater than zero, the time dependency becomes relevant. For entering time dependent boundary conditions, the variable `~Solve.time` holding the current time can be used.

Using self-heating in the device simulation includes temperature increase in the lattice due to ohmic losses leading to a non-constant temperature distribution within the device. To account for these effects, the heat-flow equations will be solved self consistently with the semiconductor equations. Many model parameters like the mobility, for example, are influenced by the local temperature.

Using self-heating requires the additional definition of thermal boundary conditions.



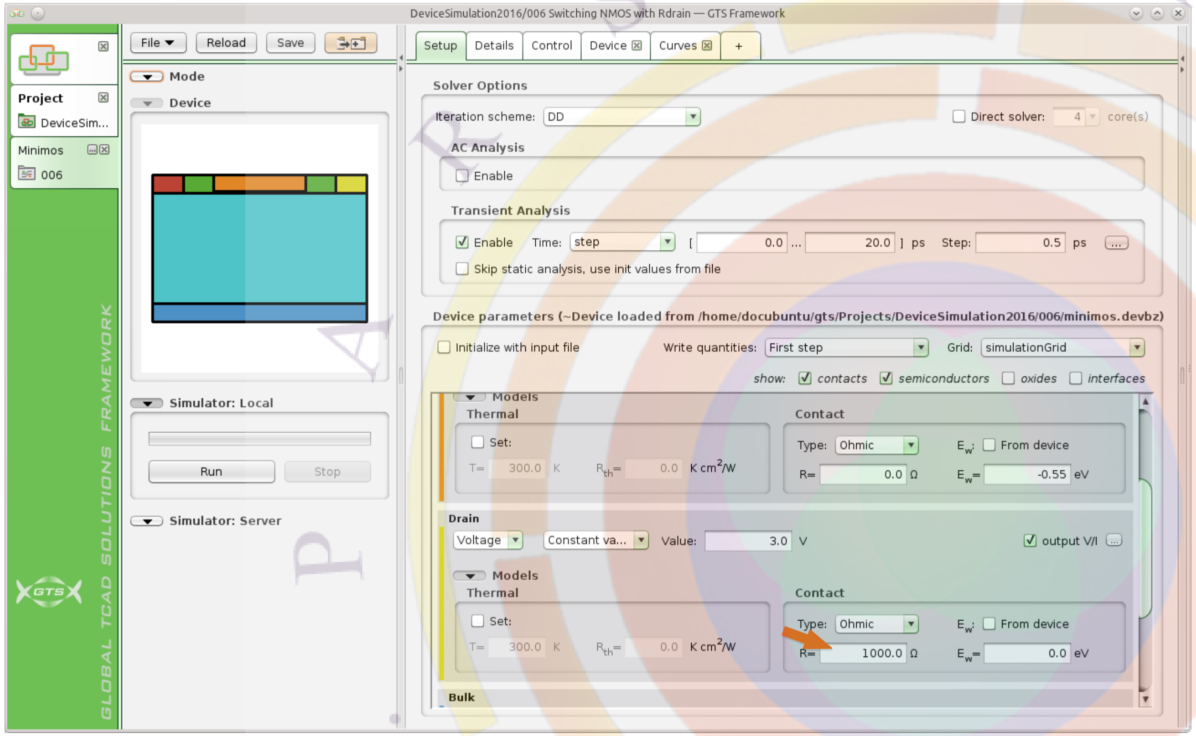
2.1. Switching With Contact Resistance

2.1.1. Setup Transient Simulation

Use the original structure generated in this tutorial and create a new Minimos-NT tool folder. The time dependent transient simulation can be activated using the "enable" check-box in the "Transient analysis" section. Change the other settings as shown in the figure. The expression `if(~Solve.time > 0ps && ~Solve.time < 10ps, 3V, 0V)` generates a single rectangular switching cycle raising from 0 to 3 V at time zero and back to 0 V at time 10 ps.

Note that the reduced expression `if(~Solve.time < 10ps, 3V, 0V)` would use 3 V for the initial static simulation and thus result in the lack of a rising edge at time 0 ps.

Click "Run" to start the simulation.



2.1.2. Add Contact Resistance

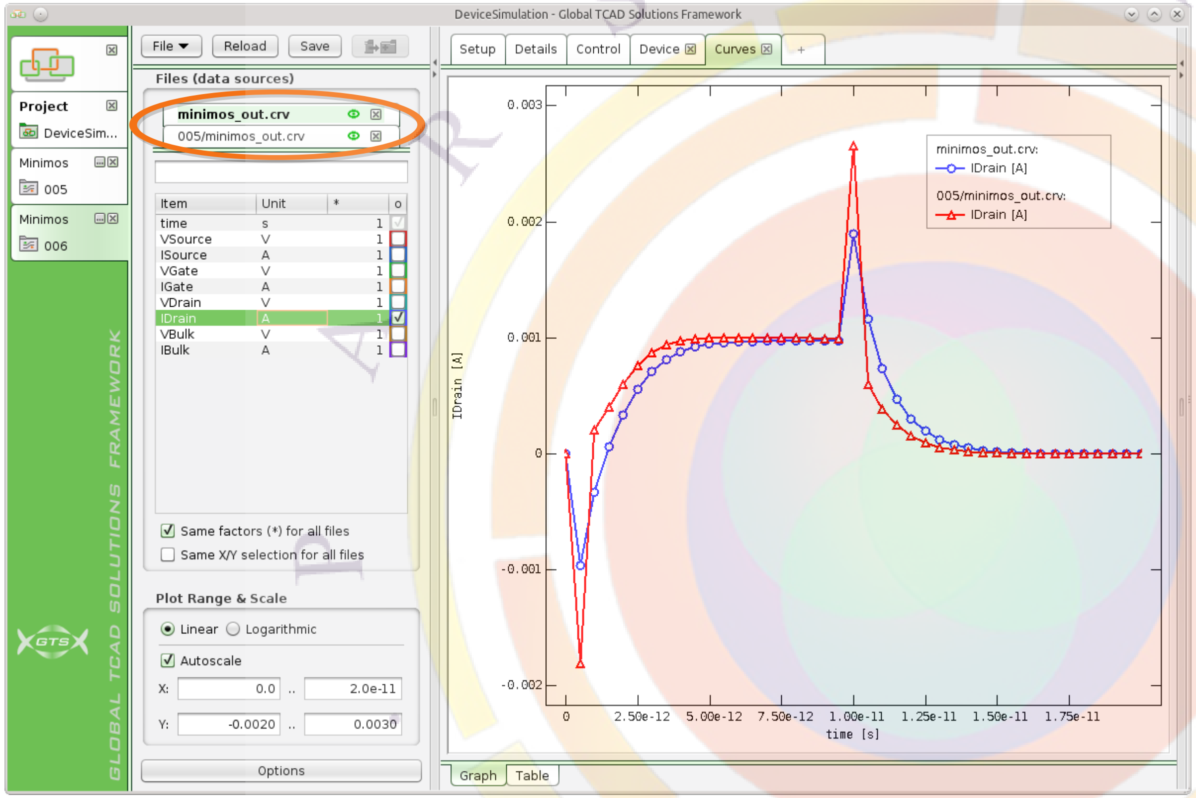
Next, we want to survey the influence of a resistance in the Drain contact which also influences the current peaks during the input voltage shoulder.

To preserve the existing results, we will continue in a new ToolFolder:

1. Go back to the Setup page
2. Click the orange "Next Tool" button
3. Select "Duplicate this ToolFolder"
4. Click OK

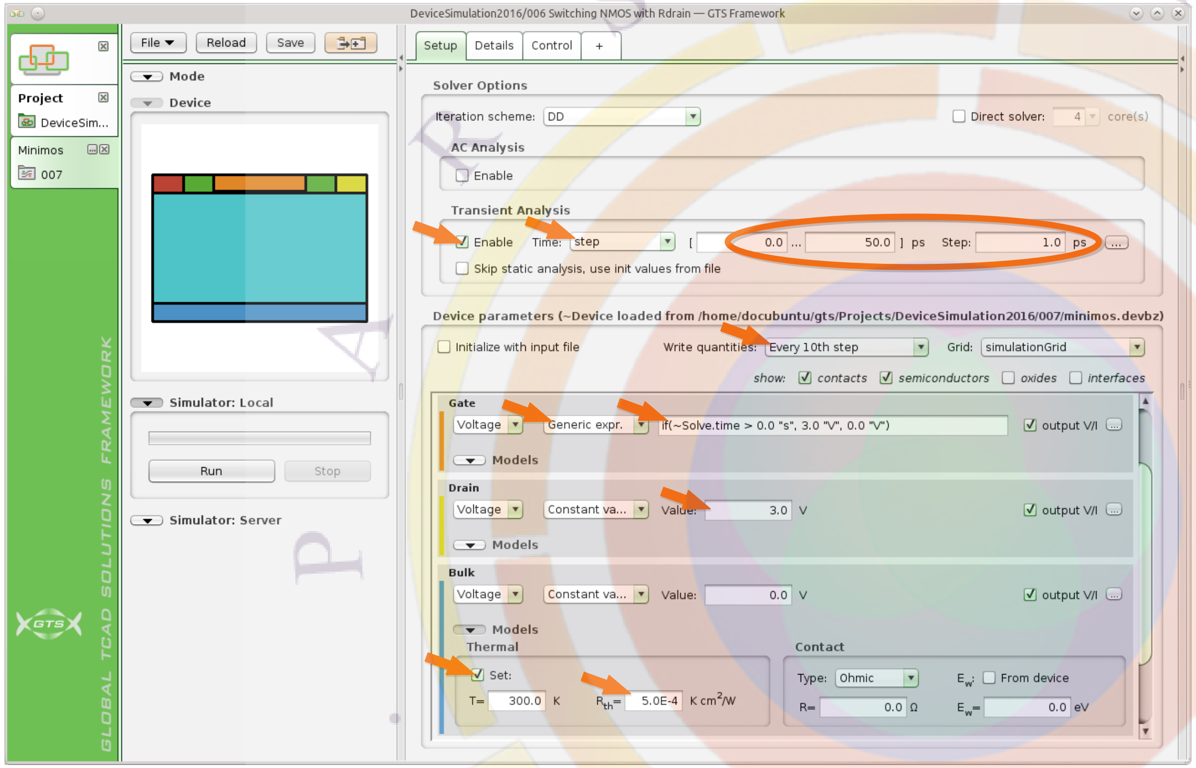
A new ToolFolder, containing a copy of your previous data, is created and opened. In the Setup page, the said resistance can be introduced:

1. Enter the resistance as marked in the figure.
2. Run the simulation



2.1.3. Results

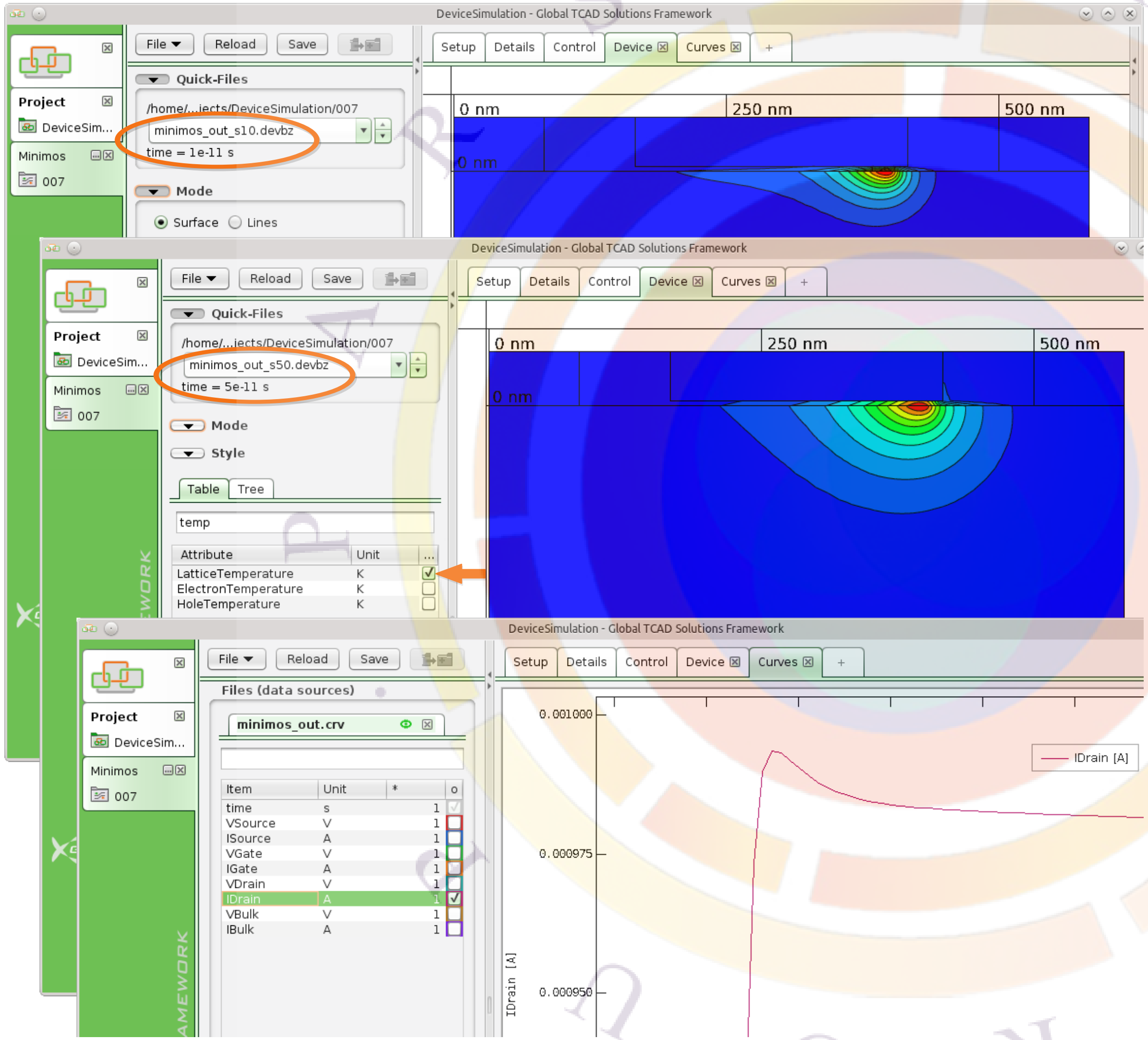
The switching behavior can be plotted using the "Curves" page. The output of the simulation with and without the additional resistor can be combined using the "File » Open" dialog in the "Curves" page.



2.2. Self-Heating

2.2.1. Setup

Use the original structure generated in this tutorial and create a new Minimos-NT ToolFolder. Setup the transient simulation settings and the thermal contact details as shown in the figure. The expression `if(~Solve.time > 0s, 3V, 0V)` generates a single edge at the beginning of the simulation. Assign -0.55 eV to the work function difference E_{w} of the Gate contact, relative to the substrate. Before starting the simulation note the detailed settings in the next step.



2.2.2. Results

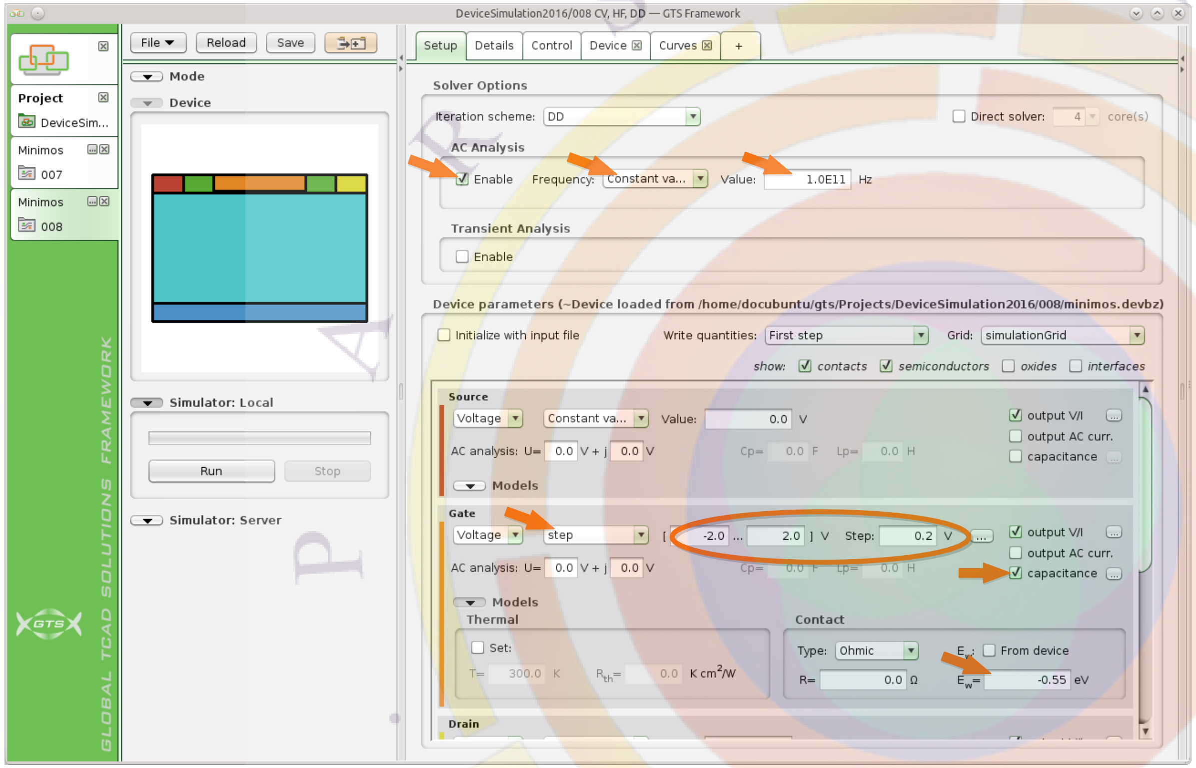
There are now more than one device output files, one every 10 steps. The names include a suffix _sxx were xx represent the step number. The desired file can be chosen in the "Quick-Files" area of the "Device" page. In the figure the temperature distribution at steps 10 and 60 is shown, corresponding to the time steps 10 ps and 60 ps, respectively. The figure also shows a plot of the Drain current decreasing over time due to the overall negative temperature coefficient found in this simple MOS structure.

In the lower part of the figure, the graph is zoomed in to the relevant area of the drain current.

Part 3

Small-Signal Mode

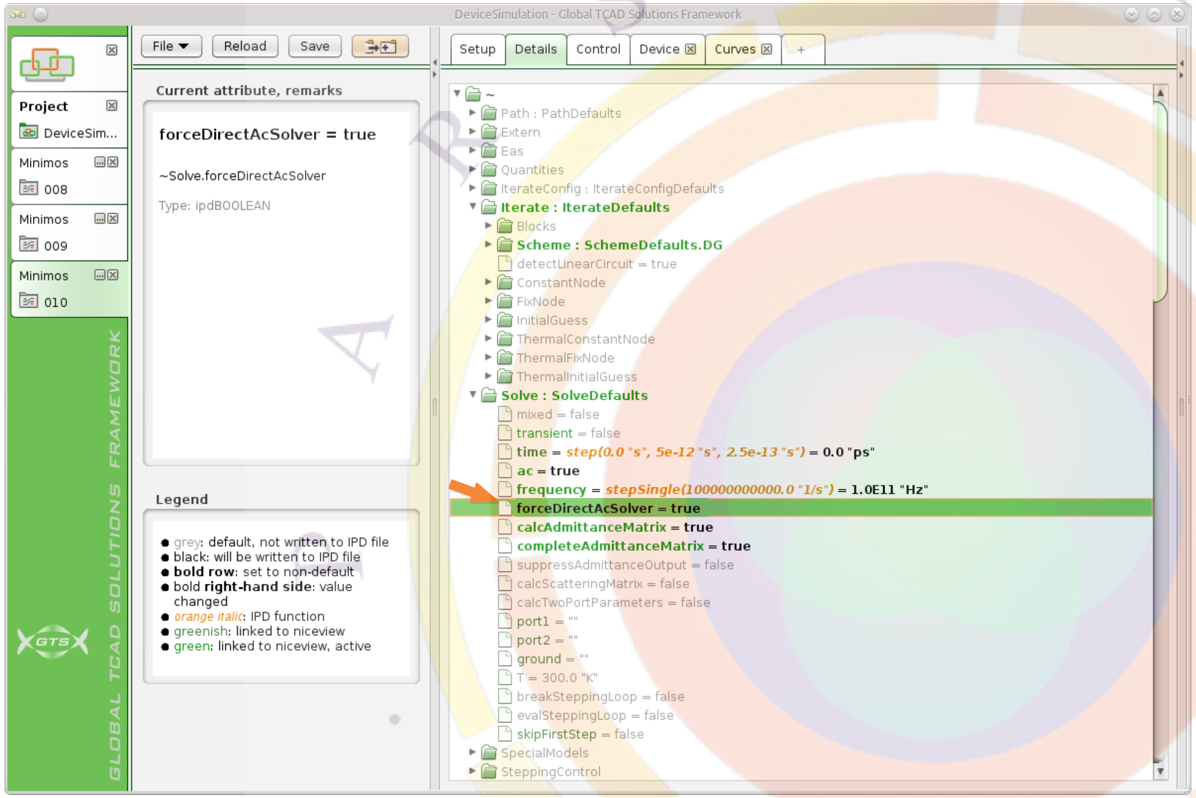
In the small-signal mode (in the context of Minimos-NT often referred to as AC mode), the equation system is linearized at a given DC operating point and solved using complex AC boundary conditions. Additionally, Y- and S-matrices as well as quadrupole parameters can be evaluated. To show some features a CV-Curve for high and low frequency will be simulated, the density gradient model will be used and finally interface traps will be taken in account by introducing Shockley-Read-Hall recombination.



3.1. CV-Curve

3.1.1. Setup AC Analysis

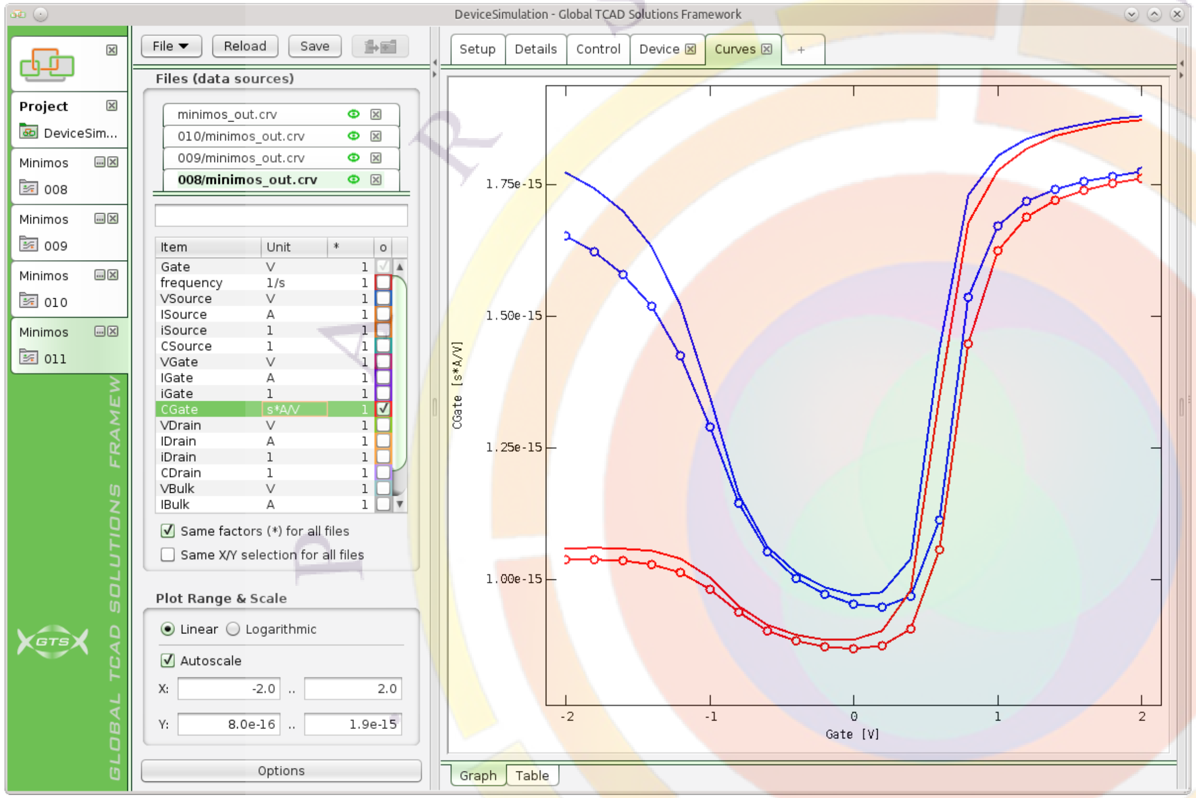
In this example, the CV-curve of the nMOS created in step one will be calculated. For this, create a new Minimos-NT ToolFolder using the "Next tool" button of the structure generated in the first step and set the parameters according to the figure. Start the simulation. With 100 GHz this simulation will generate the high-frequency curve. Copy this tool folder and change the frequency to 1 MHz to calculate a low-frequency curve. Run this simulation as well.



3.1.2. Density Gradient, Detailed Settings

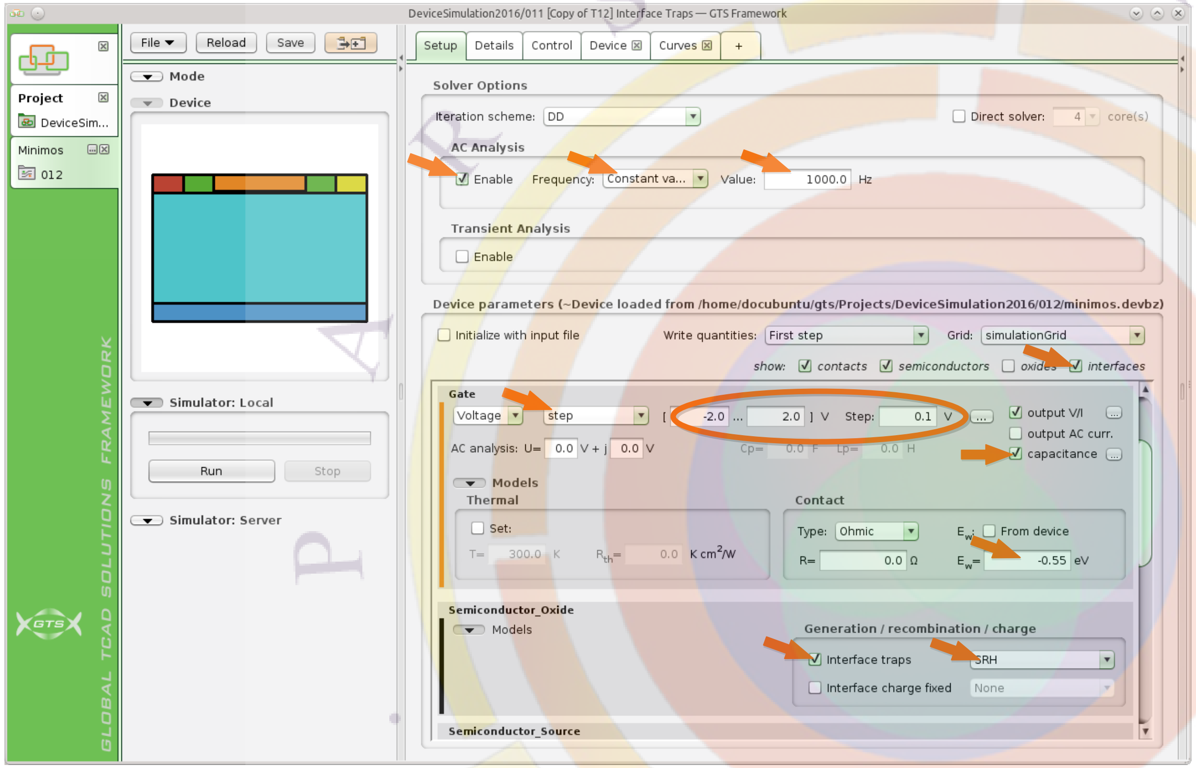
As we want to compare the results of the simulation with and without the density gradient model, copy both of the previously generated tool folders. Open the new tool folders and change the iteration scheme in the “Setup” page from `"DD"` to `"DG"`. So we end up with four tool folders to compare the results of high and low frequency CV curve with and without using the density gradient model.

To achieve better convergence for the DG simulation in the AC domain, choose a direct solver by setting `~Solve.forceDirectAcSolver = true` as shown in the figure.



3.1.3. Results

The CV curves can be plotted using the "Curves" page. The output of the simulation for high and low frequency, with and without density gradient model can be combined using the "File » Open" dialog in the "Curves" page.

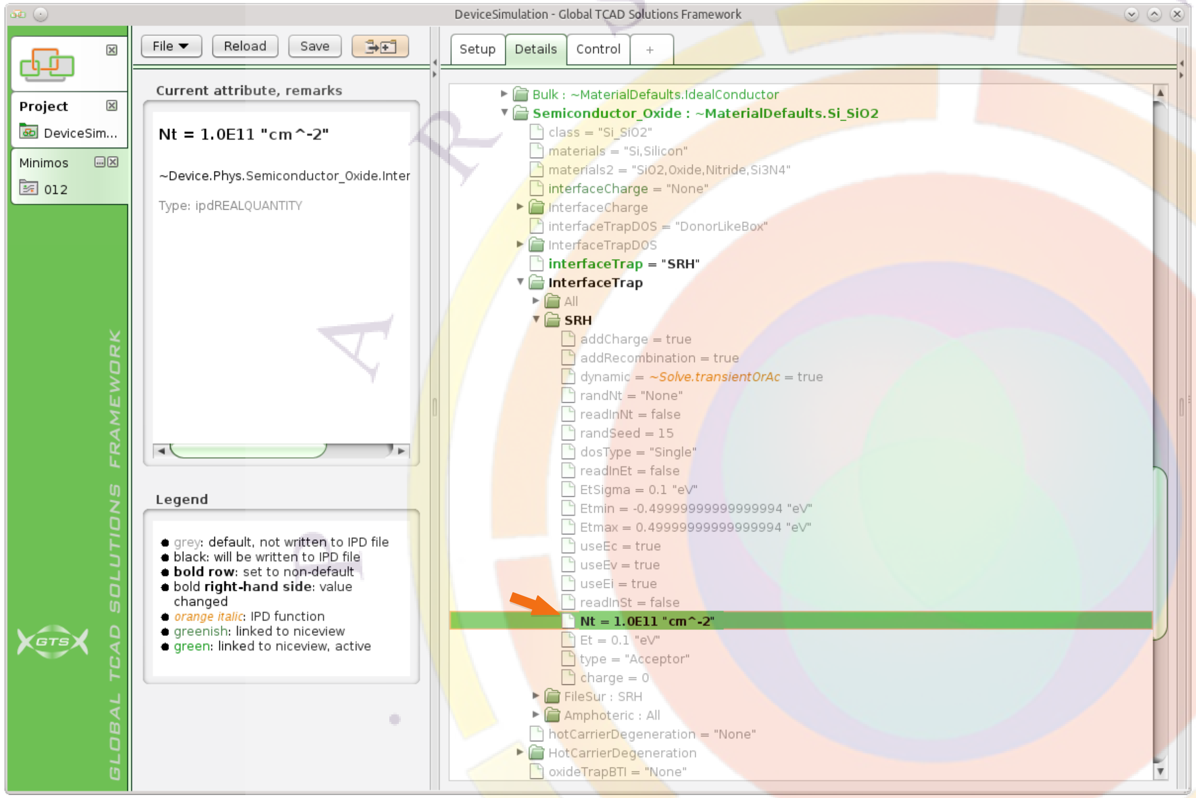


3.2. Interface Traps

3.2.1. Setup SRH Recombination

Set the frequency and voltage values and introduce SRH interface traps in a new Minimos-NT ToolFolder:

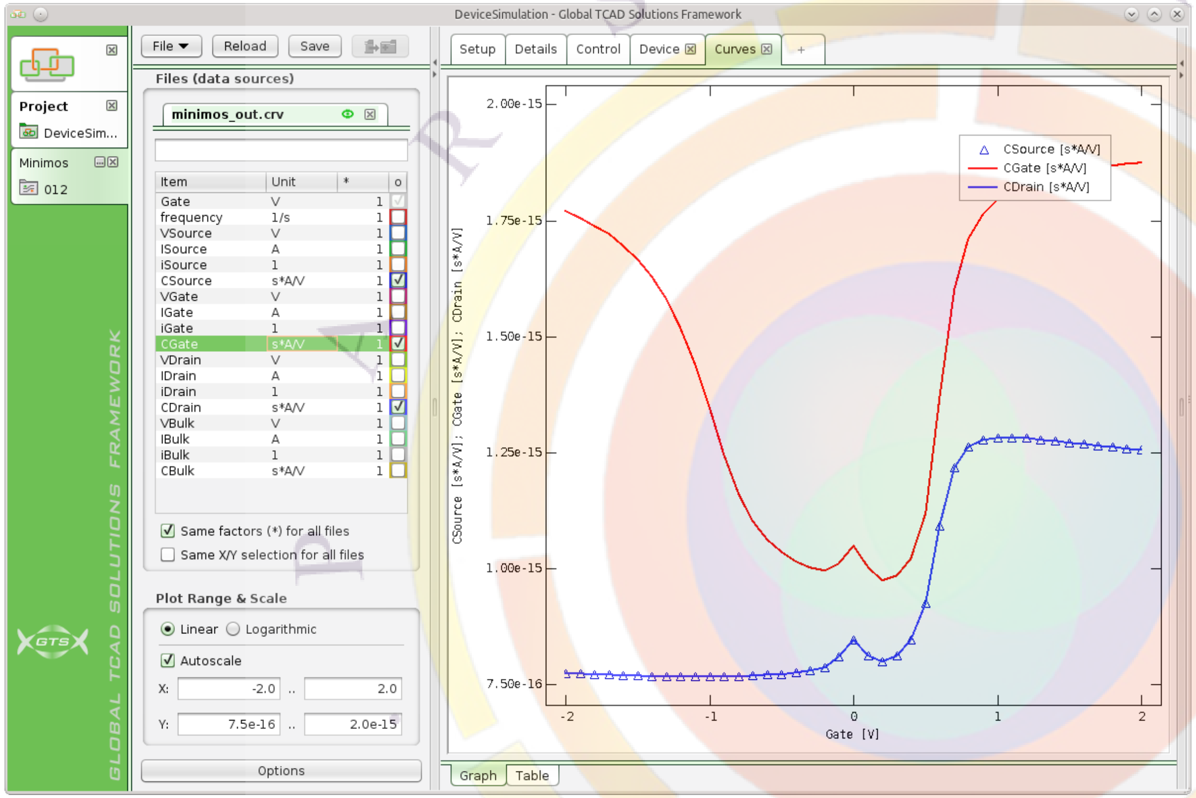
- Create a new Minimos-NT ToolFolder by using the “Next Tool” button of the original structure, created in this tutorial
- Make the interface menu visible in the “Device parameters” section to activate interface traps. Choose the SRH model
- Enable AC analysis with a constant frequency of 1 kHz
- Activate capacitance output for all contacts (Gate, Source, Drain, Bulk)
- Change the rest according to the figure



3.2.2. Trap Concentration

- Change the trap concentration N_t by editing the corresponding variable to "`1.0E11`" cm^{-2}
- Run the simulation in the "Control" page

Take note of the other editable attributes, for example the trap energy level E_t.



3.2.3. Results

Change to the “Curves” page and inspect the different capacitances, depending on the Gate voltage.

As a symmetric nMOS is simulated, the Drain and Source capacitances are the same.

Compared to a simulation without interface trap modeling, there is a significant increase of the capacitance for Gate voltages around 0 V. In this region the Fermi energy is in the range of the interface trap energy, and therefore those states can contribute to the charge.

Conclusion

In this tutorial, various device simulation capabilities of Minimos-NT have been presented. By example of steady state analysis, the possibility of initialyzing simulations with previously gained results and the usage of quantum tunneling through the oxide has been illustrated. Furthermore, transient simulation studies, consideration of self heating effects, as well as small signal analysis have been performed. The extraction of CV-curves has been explained, including the application of the Density Gradient and Shockley-Read-Hall models.

Further Reading

We welcome you to have a look at further *GTS tutorials* and *examples*, which you can open in ***GTS Framework***. Next to the basic ones included with the release, you can download more sophisticated tutorials and examples from MyGTS at `https://globaltcad.com/mygts`. Extracting the archives to your projects folder makes the tutorials visible in the projects list (highlighted yellow). Previews are provided at `http://www.globaltcad.com/en/solutions.html`.

For additional information, please refer to `http://globaltcad.com/` or feel free to contact us at `info@globaltcad.com`.

Appendix A

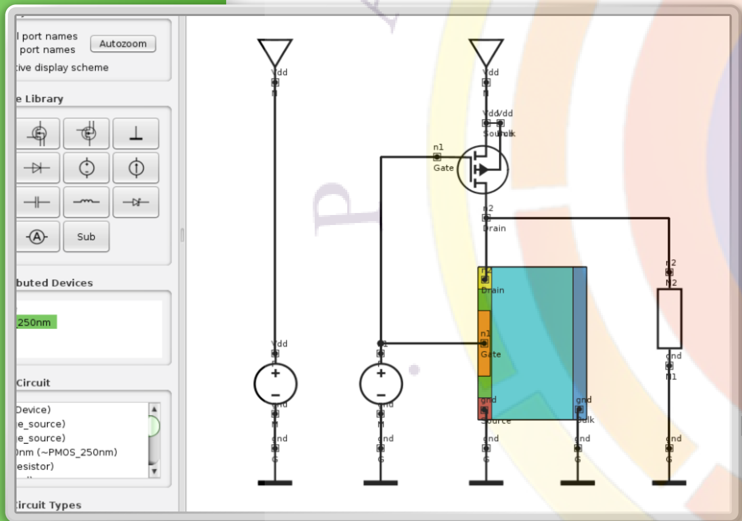
ToolFolder List

The project **DeviceSimulation** contains the following ToolFolders (TF):

TF	Tool	Description
T01	Structure	Simple 2D NMOS Transistor
T02	Minimos	Transfer characteristic
T03	Minimos	Simulate II, input from T02
T04	Minimos	Oxide tunneling
T05	Minimos	Switching NMOS
T06	Minimos	Switching NMOS with Rdrain
T07	Minimos	Self-Heating
T08	Minimos	CV, HF, DD
T09	Minimos	CV, LF, DD
T10	Minimos	CV, HF, DG
T11	Minimos	CV, LF, DG
T12	Minimos	Interface Traps

GLOBAL TCAD SOLUTIONS — GTS FRAMEWORK — TUTORIAL

GTS FRAMEWORK



Mixed-Mode Part I

Tutorial

global TCAD solutions

Mixed-Mode Part I – Tutorial, GTS Framework Release 2016.09
Revision of March 25, 2017

Global TCAD Solutions GmbH
Bösendorferstraße 1/12
1010 Wien
Austria / Europe

Phone: +43 1 9255049
Email: info@globaltcad.com

Additional information is provided at: http://www.globaltcad.com/

Cover & layout: prausedesign

Contents

Introduction

Using This Tutorial

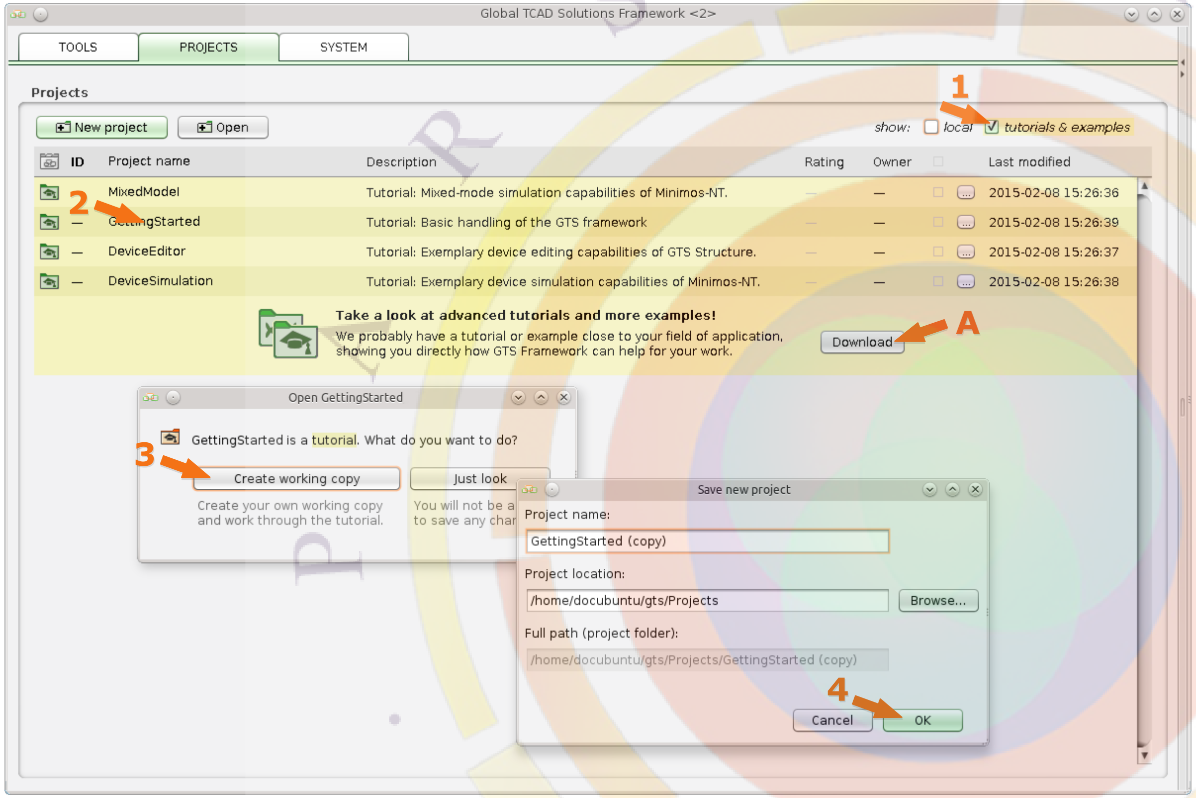
GTS *tutorials* generally consist of a written description (this text) and a GTS *project* (simulation data) which you can open in ***GTS Framework*** (yellow items in project list). As shown below, we recommend to create a working copy from the project, and proceed step by step, guided by this text. If not yet familiar, please refer to the *GettingStarted* tutorial.

Only a few tutorials are included with the release; the others are available via *MyGTS* at `https://globaltcad.com/mygts`. When logged in, please download the respective file and extract it to the gts projects folder on your PC. After restarting GTS Framework, the tutorial project will appear in the *Projects* list.

This is an intermediate (level 2) tutorial.

This tutorial was created using GTS Framework Release 2016.09. Other releases might need adjustments or have slightly different user interfaces.

Working in a copy of a tutorial project, you can open the yellow Txx ToolFolders at any time. They contain the data of the described simulations, which you can use for reference or as starting points for your own simulations.



Project / Working Copy

If you have not already done so, go to the *Framework Home* and locate the project associated with this tutorial in the *Projects* list, and create your working copy from it:

1. Check "tutorials & examples"
2. Click on the respective tutorial project: ***Mixed-Mode Part I***
3. Choose "Create working copy"
4. Check the project name, click "OK"

The project is created and opened, so that you see the *Project Home.* — *Ready to start!*

A. If you miss the respective project, please download it via *MyGTS* (click "Download", see previous page).

Alternatively, you can start the tutorial with an empty project – just create a new project. (The advantage of the tutorial project is that you have the results at hand any time.)

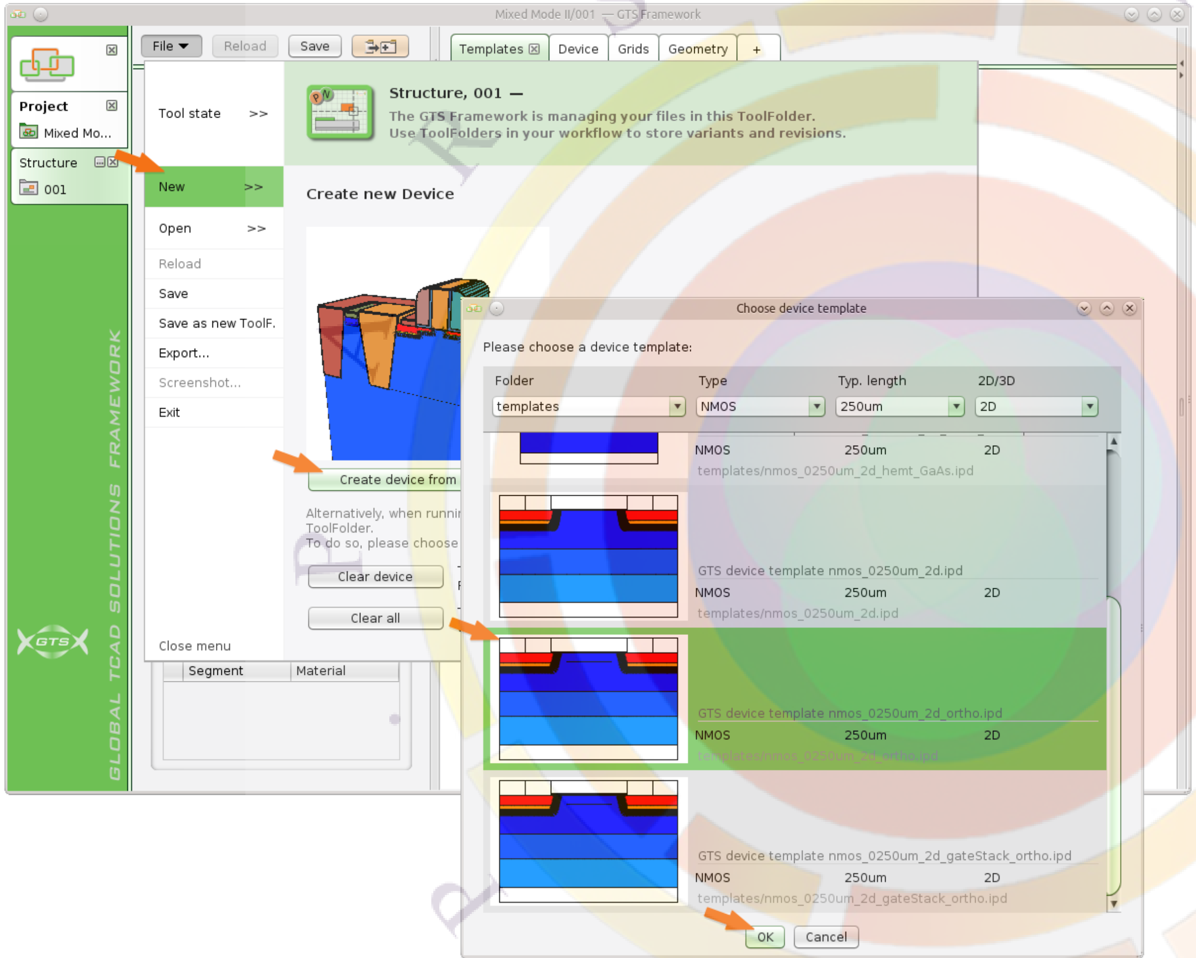
Part 1

Single Transistor Inverter

In this example, we will set up a mixed-mode simulation. In this mode the transfer characteristics of a distributed transistor and the output voltage of a single transistor inverter will be simulated. Afterwards the circuit will be extended with another distributed transistor to simulate the output voltage of a CMOS inverter. Finally one of the distributed transistors will be replaced by a ***Spice*** MOS Level 3 model to investigate the output voltage again.

It is assumed that you are already familiar with the main concepts of ***GTS Framework*** and the idea of projects and ToolFolders, as explained in the Getting Started tutorial.

The mixed-mode simulation model might not be included in your license. Please contact your distributor or Global TCAD Solutions directly for more information.



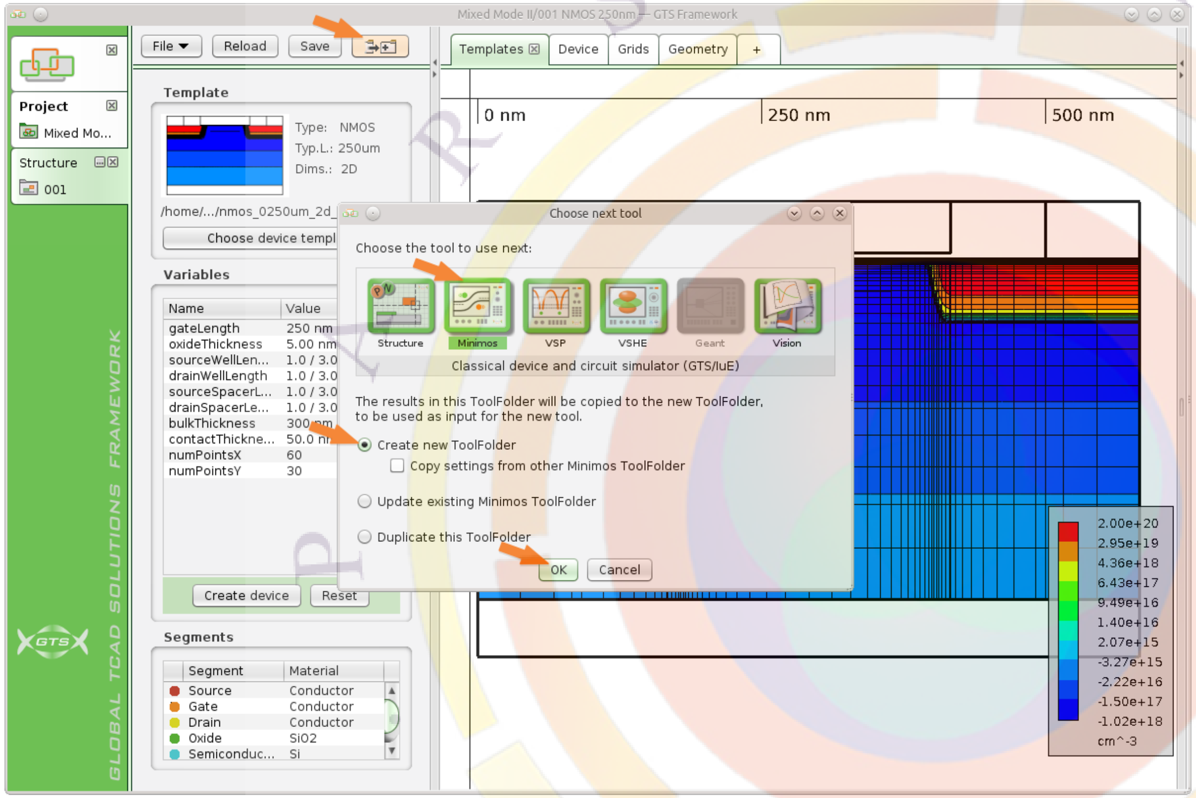
1.1. Set Up Mixed-Mode

1.1.1. Structure

- In your working copy of the *MixedMode_I* tutorial project, click "Add tool" to add a ***Structure*** ToolFolder

1.1.2. Create Distributed Device

- Create a 2D NMOS device with 0.250 μm channel length using ortho grid (`nmos_0250um_2d_ortho.ipd`)
- Use the button "Create device" to generate the device

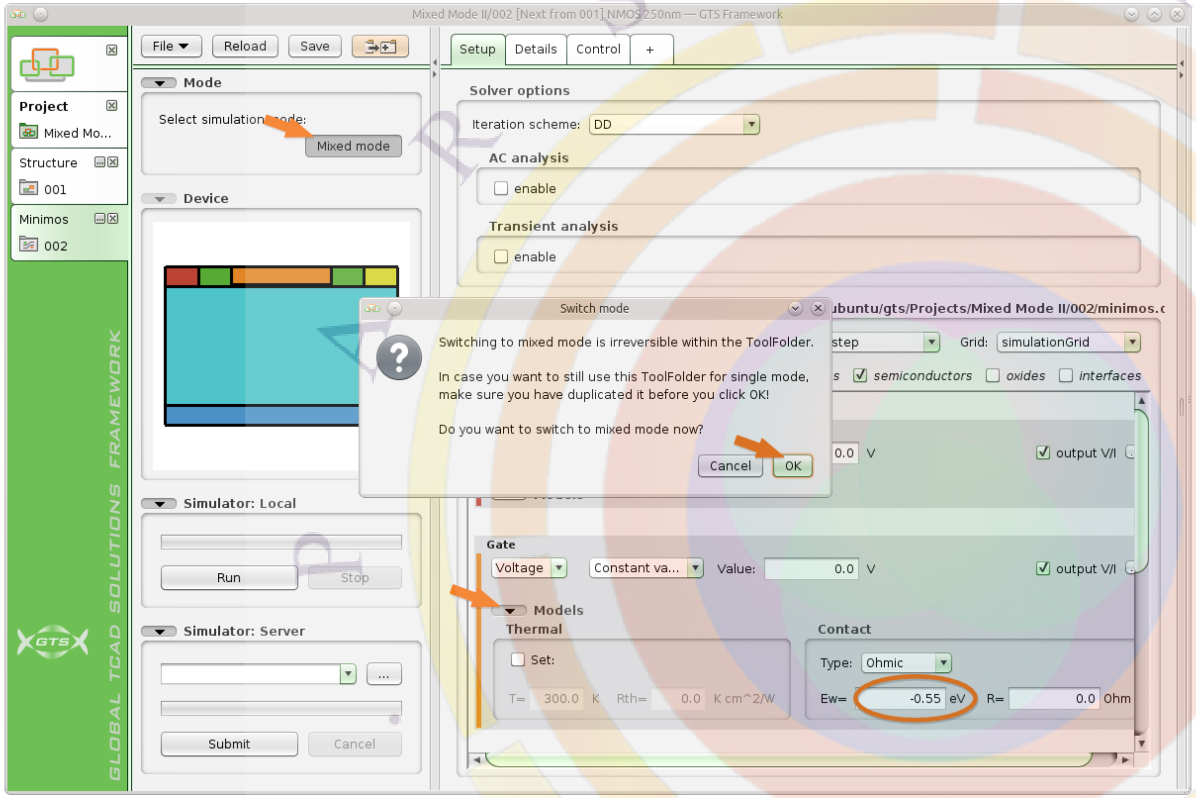


1.1.3. Create Minimos-NT ToolFolder

- Use the "Next tool" button to create a new ***Minimos-NT*** ToolFolder

As the projects will get larger, always use meaningful names for the ToolFolders.

Always confirm save requests with "Yes" in this tutorial.

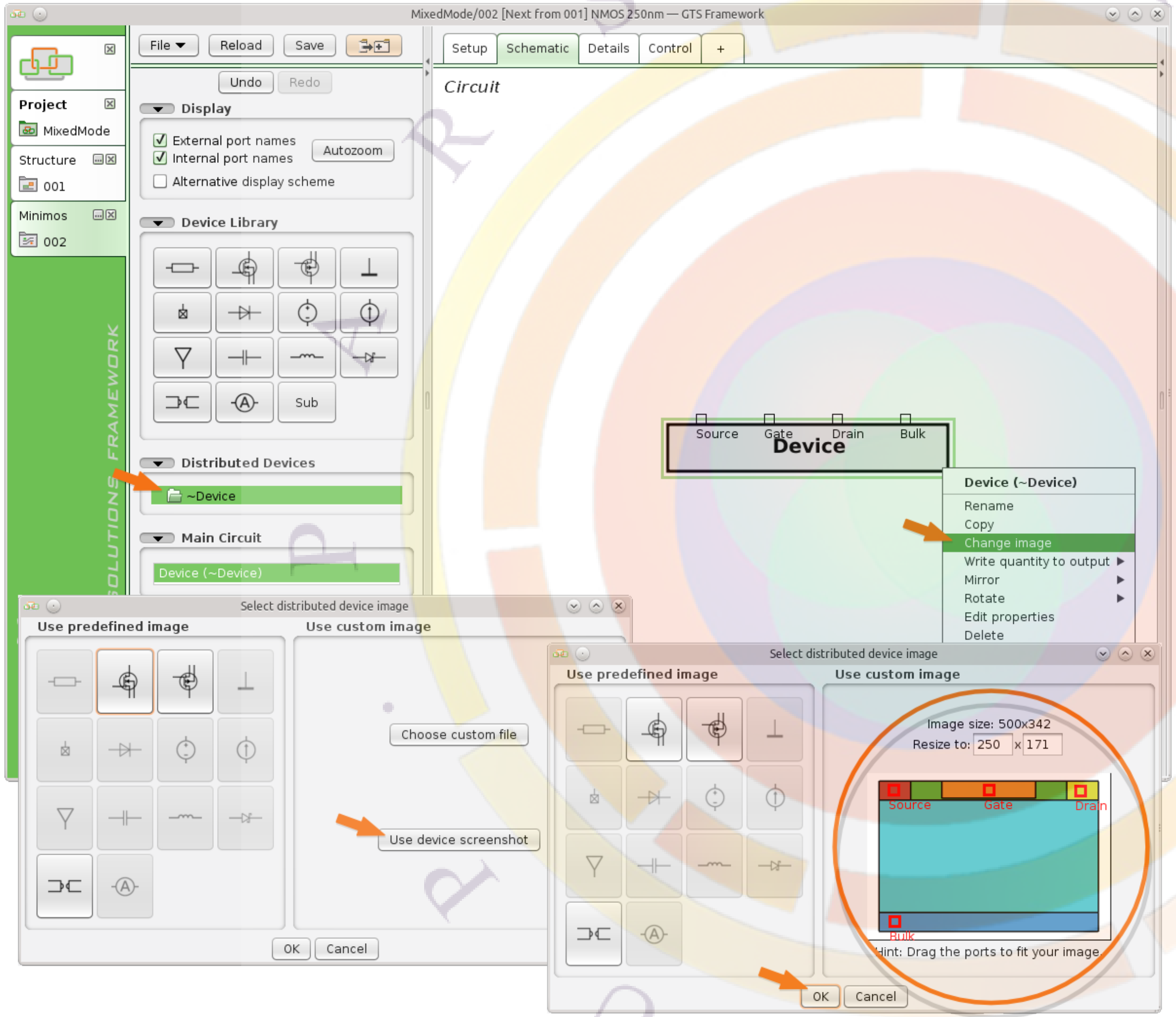


1.1.4. Switch to Mixed-Mode

- Set the work function difference of the Gate contact to the commonly used $E_w = -0.55\,\mathrm{eV}$
- Press the button "Mixed mode" placed in the group "Mode" in the control panel of the "Setup" page

This changes the simulation setup to mixed-mode. Additionally the simulation scheme will be changed to "MixedDD" automatically.

> The change into a mixed-mode simulation cannot be reverted in the graphical user interface. In case of doubt duplicate the tool folder before switching to mixed-mode.

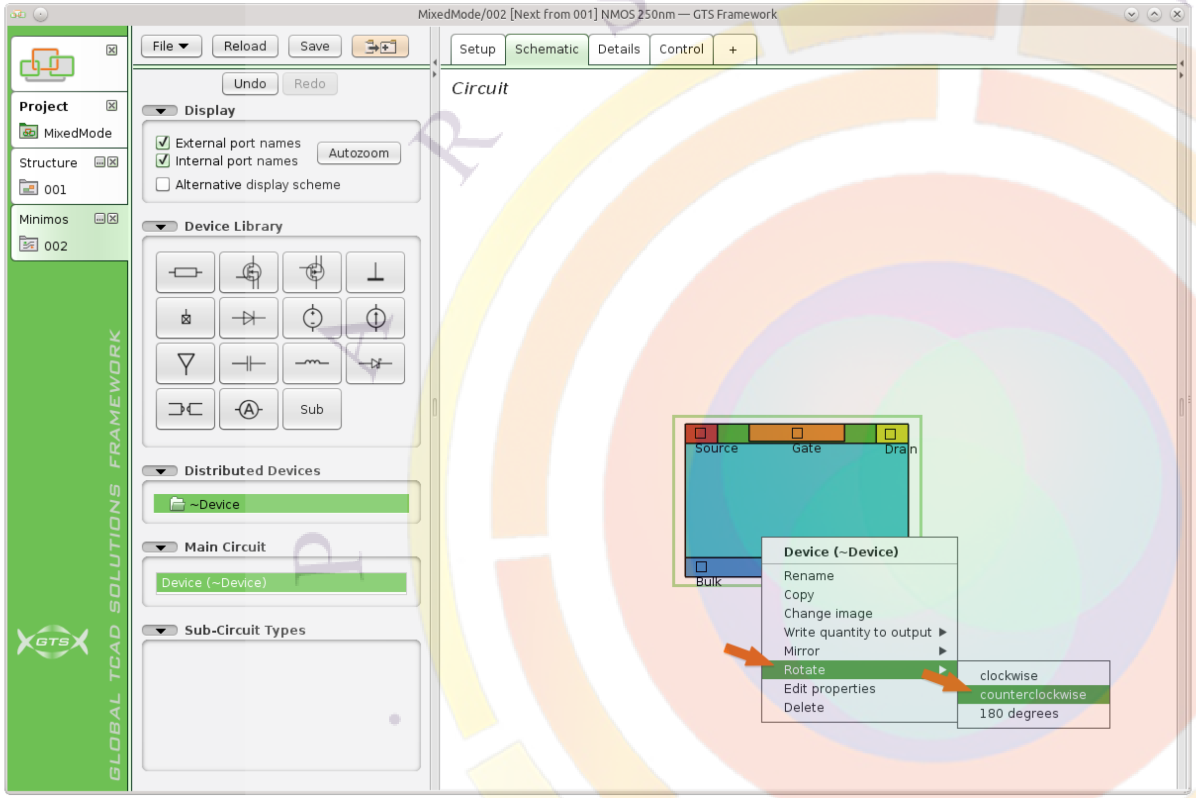


1.2. Create Circuit

1.2.1. Add Distributed Device

After switching to mixed-mode the "Schematic" page opens. First we add the distributed device and change its image:

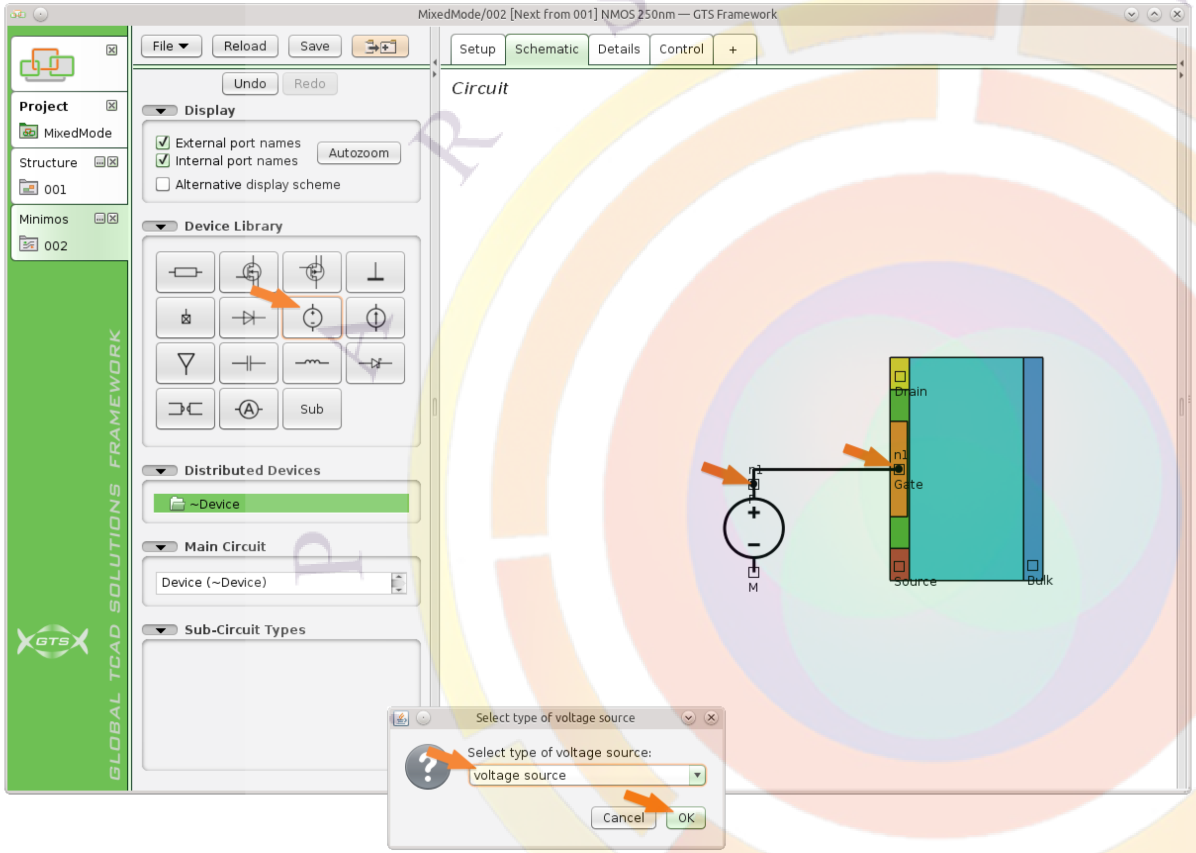
- Left-click on "~Device" in the "Distributed Devices" section of the control panel
- Place it in the work area with a left-click as shown in the figure
- Right-click on the device
- Choose "Change image" in the drop down menu
- Press "Use device screenshot"
- Make sure the ports are placed correctly and confirm with "OK"



1.2.2. Rotate Device

- Right-click on the Device
- Choose "Rotate" in the drop down menu and select "counterclockwise"

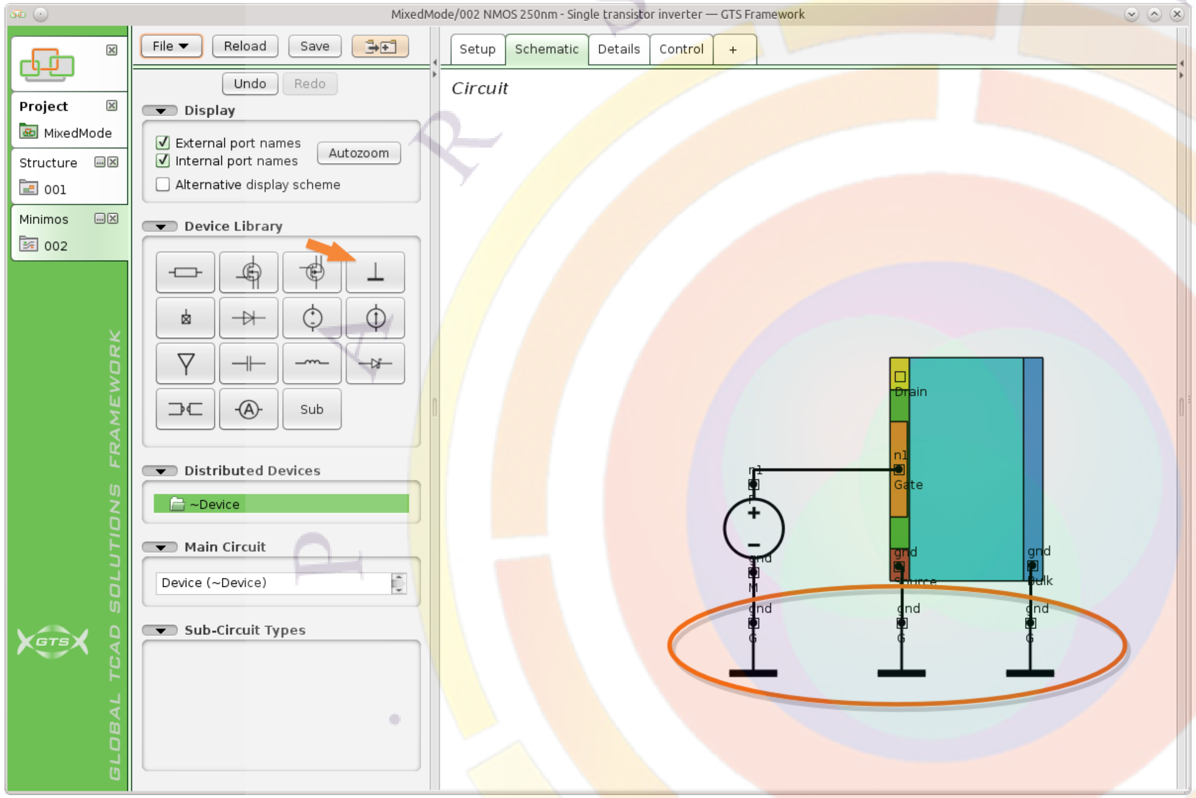
Components can be rotated with a right-click before placing them with a left-click as well.



1.2.3. Add Gate Voltage Source

Now we have to add a voltage source for the Gate voltage:

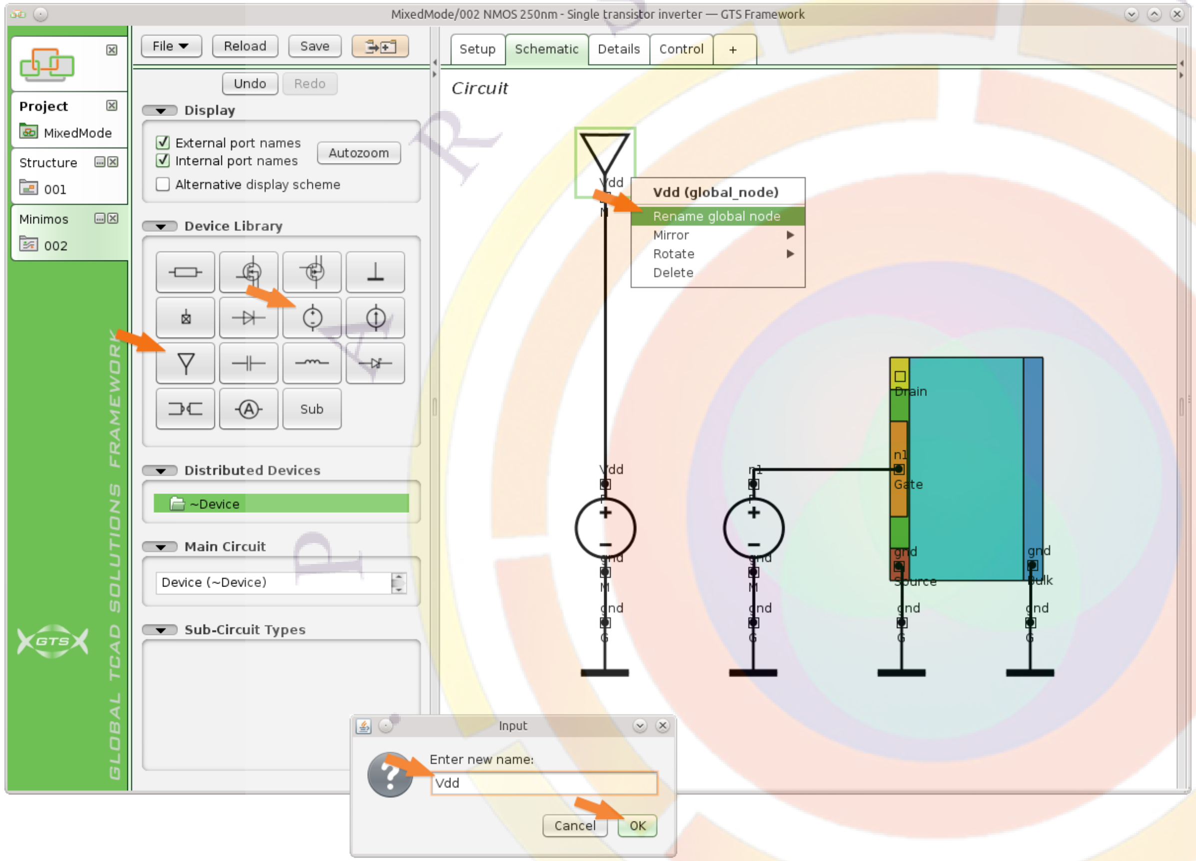
- Left-click on the symbol for a voltage source in the control panel
- Select "voltage source" and press "OK"
- Place the voltage source as shown in the figure
- Connect the voltage source with the Gate by left-clicking on the corresponding ports



1.2.4. Add Nodes And Wires

- Left-click on the GND symbol in the control panel
- Place the GND nodes as shown in the figure
- Connect the circuit as shown in the figure

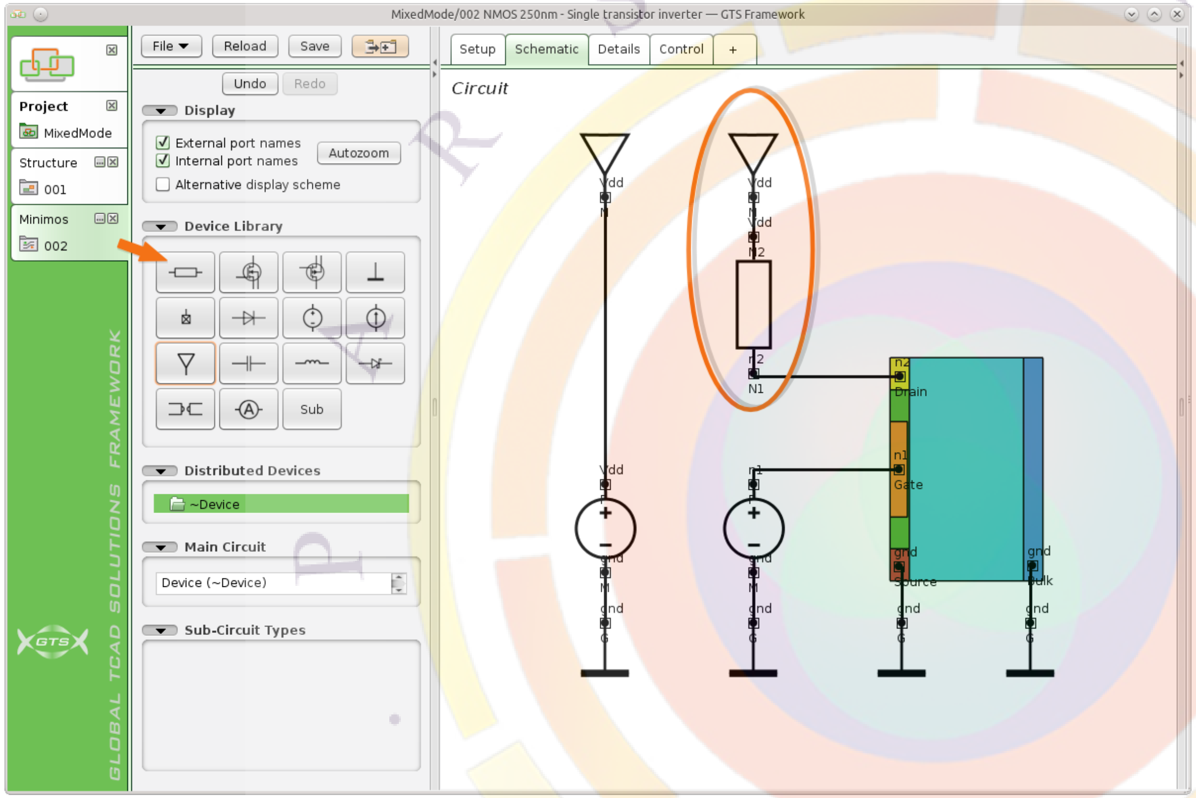
Press "space" if you want to place another instance of a previously placed component.



1.2.5. Add Drain Voltage Source

Now we have to add a voltage source for the Drain voltage:

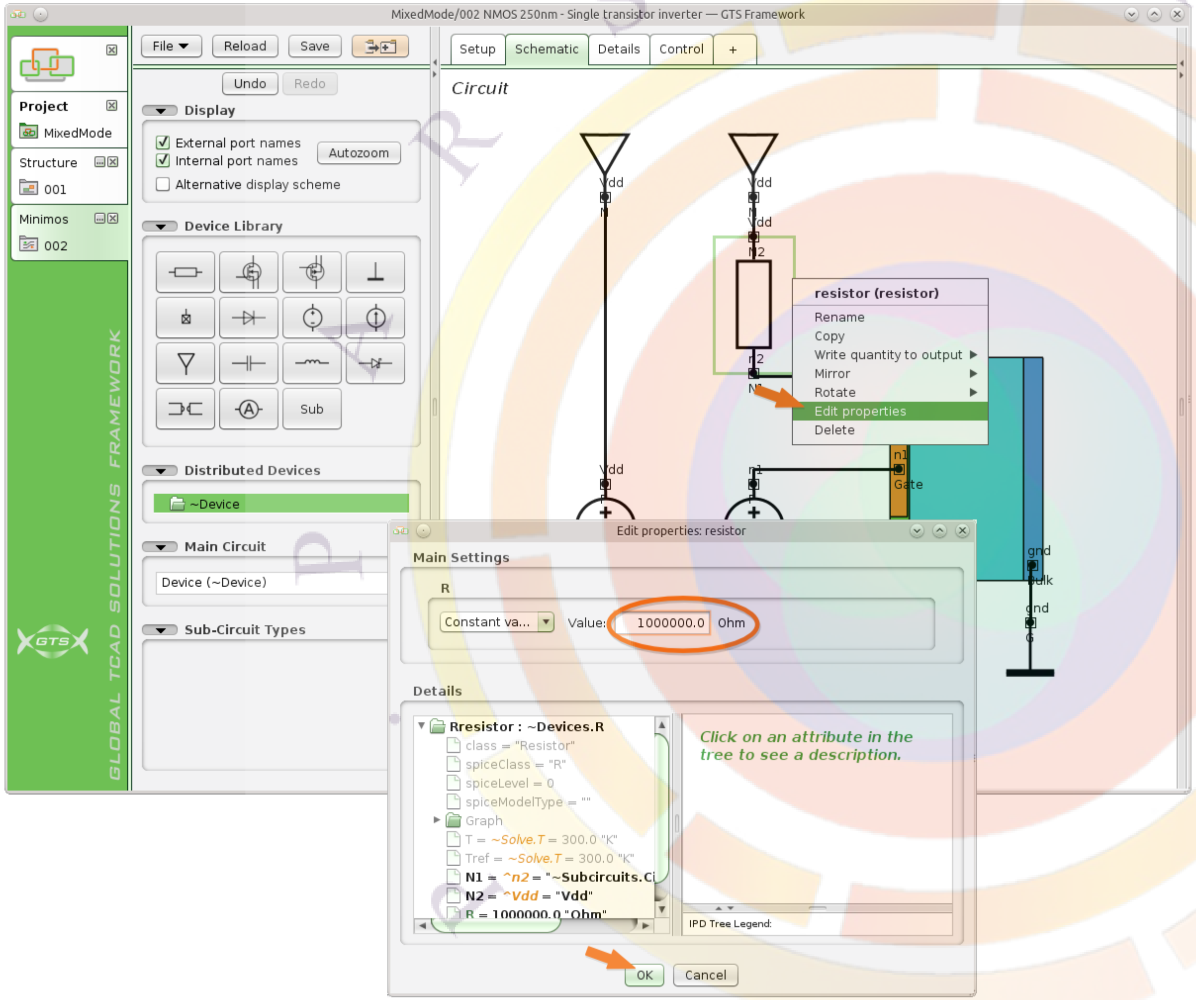
- Place another voltage source and
- Add a global node
- Connect them as shown in the figure
- Rename the new global node “Vdd” via the corresponding option of its drop down menu



1.2.6. Complete The Circuit

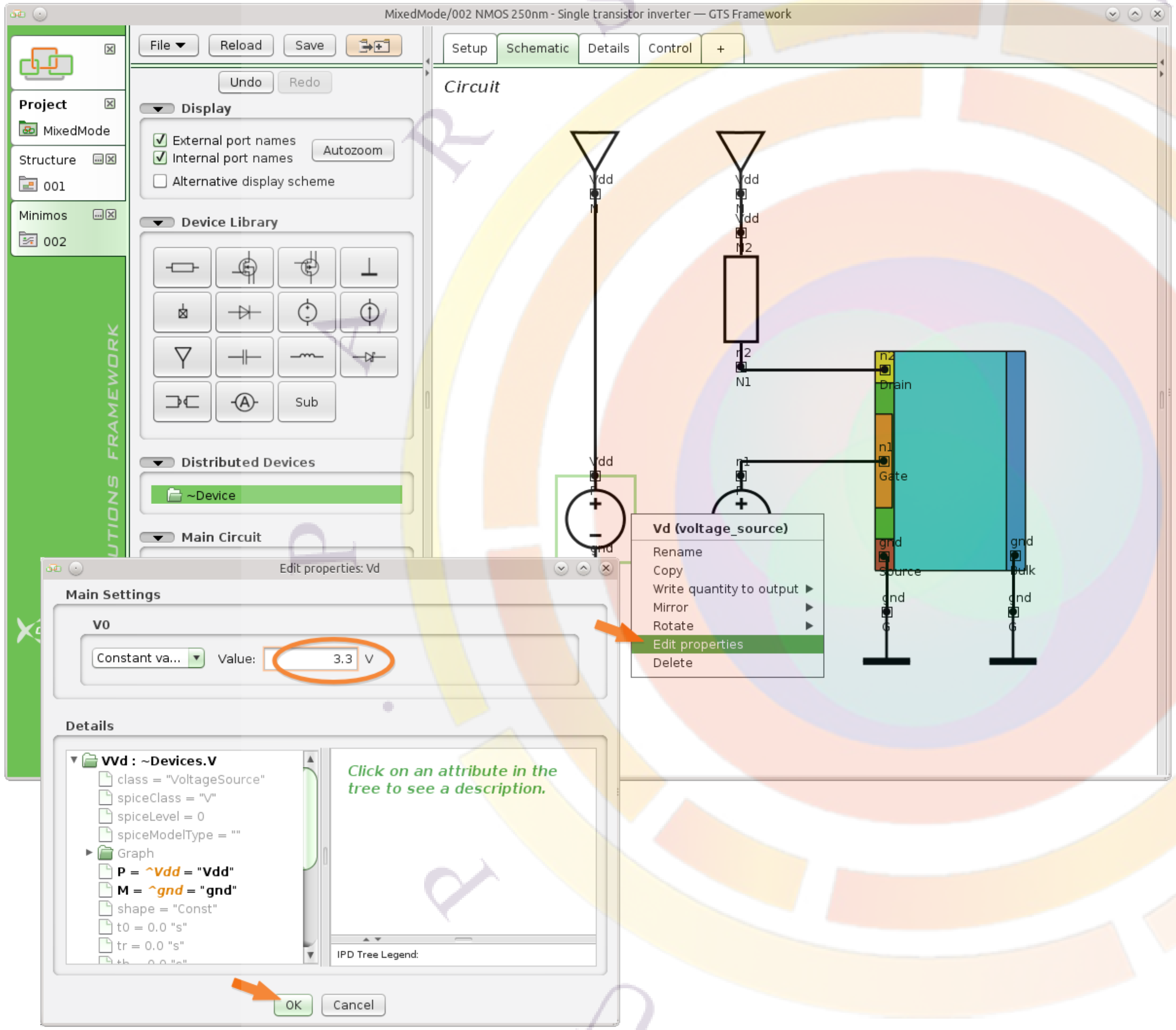
A resistor and the voltage supply for the NMOS has to be done to finish the first circuit:

- Place a resistor, rotate it with a right-click and place it as shown in the firgure
- Add a global node and name it “Vdd”
- Connect the components as shown in the figure



1.2.7. Edit Resistor Properties

- Right-click on the resistor and choose "Edit properties" in the drop down menu
- Set its value to `1.0E6` Ohm and confirm with "OK"

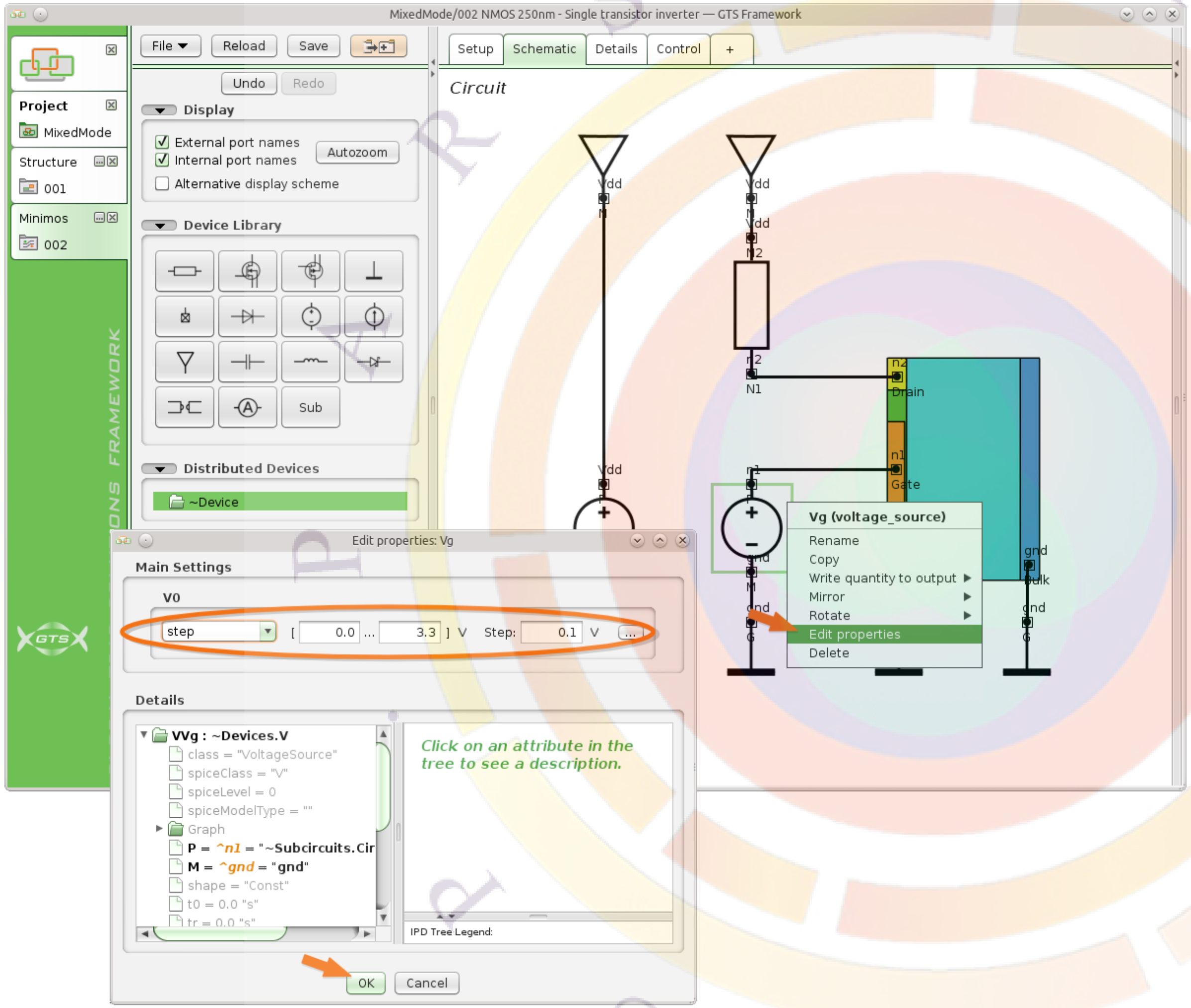


1.3. Simulate Transfer Characteristics

1.3.1. Edit Drain Voltage Source

Now we want to set up the Drain voltage source properties:

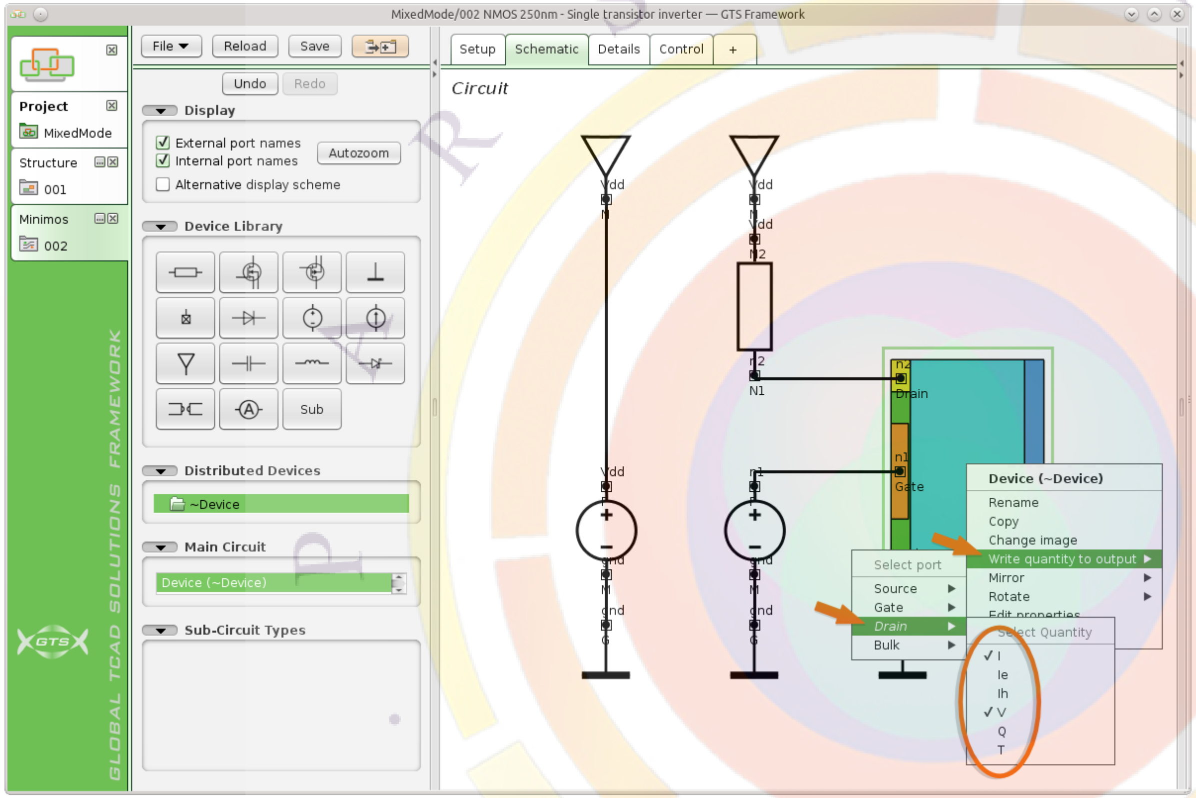
- Right-click on the left voltage source and choose "Rename" in the drop down menu
- Rename the voltage source "Vd" and confirm with "OK"
- Again right-click on the new voltage source and choose "Edit properties" in the drop down menu
- Enter a constant voltage value of 3.3 V and confirm with "OK"



1.3.2. Edit Gate Voltage Source

Now we want to set up the Gate voltage source properties:

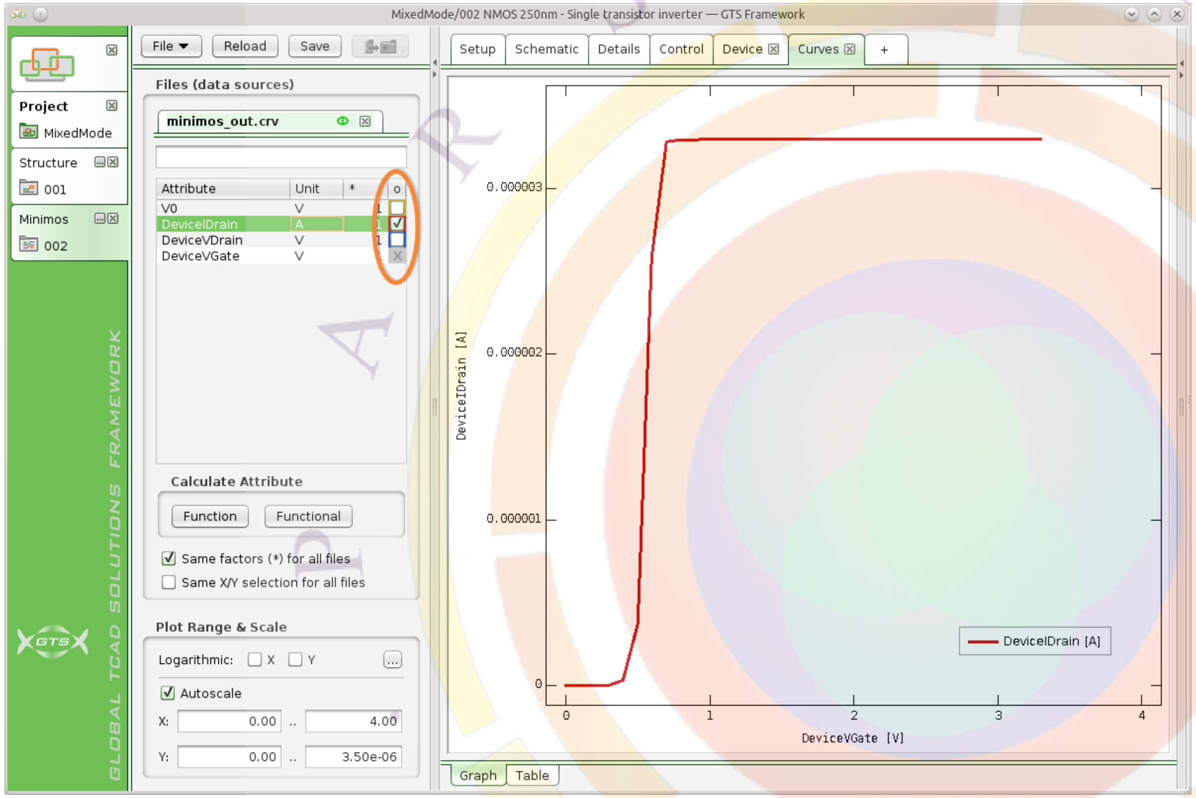
- Rename the Gate voltage source "Vg" and open its properties menu in the same way as for the Drain voltage source
- Select "step" instead of "Constant value" in the "Main Settings"
- Enter the values for the stepping as shown in the figure and confirm with "OK"



1.3.3. Set Up Output Quantities

The circuit is done but before running the simulation we should define the output quantities of the simulation:

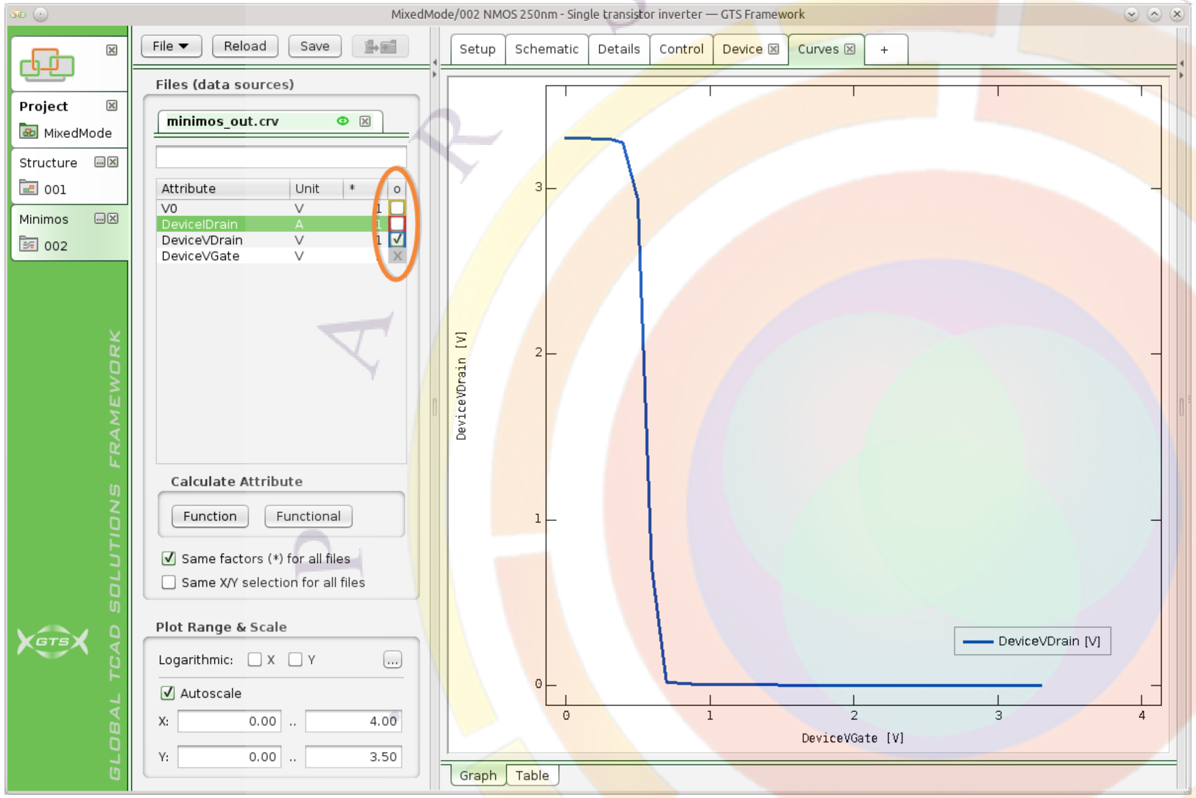
- Right-click on the distributed device to open its drop down menu
- Choose: “Write quantity to output”, “Drain”, “I” to select the Drain current as an output quantity
- Choose: “Write quantity to output”, “Drain”, “V” to select the Drain voltage as an output quantity
- Choose: “Write quantity to output”, “Gate”, “V” to select the Gate voltage as an output quantity



1.3.4. Result: Transfer Characteristics

Now we can run the simulation:

- Switch to the “Control” page and press the “Run” button
- Wait until the simulation has finished and switch to the “Curves” page
- Assign “DeviceVGate” to the X-axis via its drop down menu
- Assign “DeviceIDrain” to the Y-axis by checking the corresponding check-box



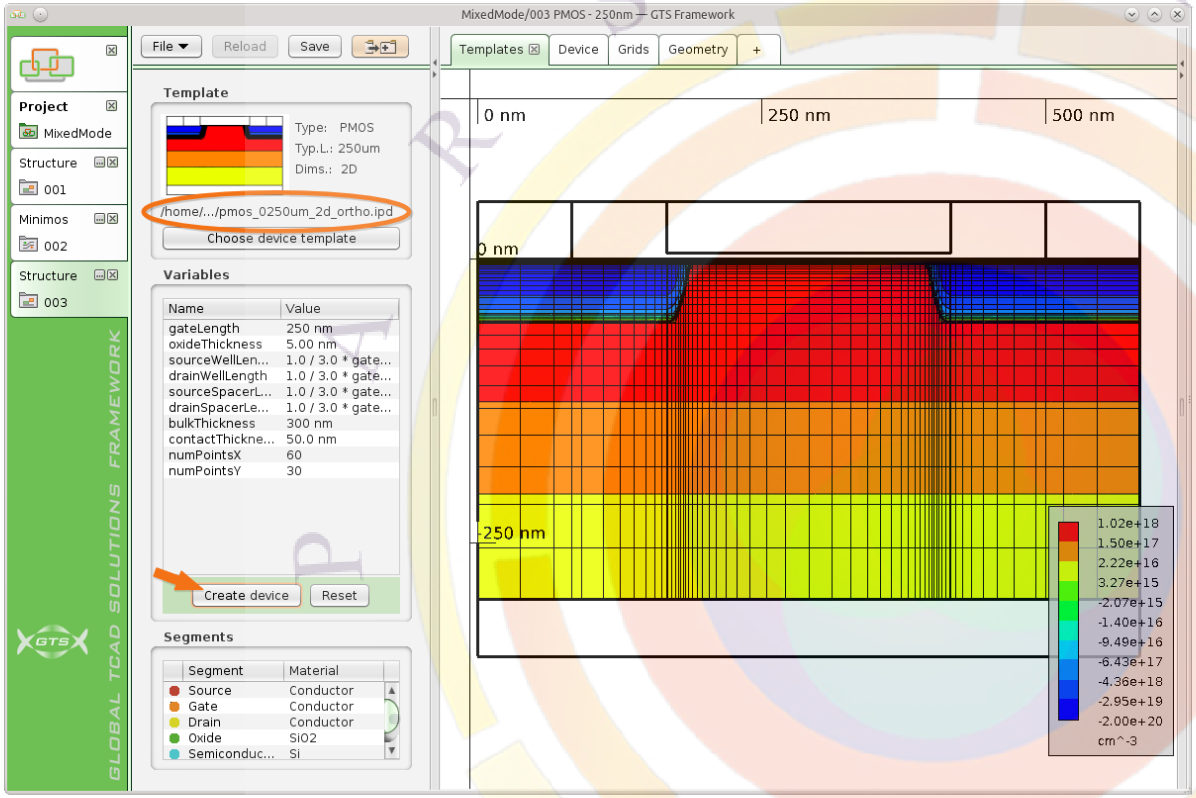
1.3.5. Result: Output Voltage

- Assign “DeviceVGate” to the X-axis via its drop down menu
- Assign “DeviceVDrain” to the Y-axis by checking the corresponding check-box

Part 2

CMOS with Distributed Devices

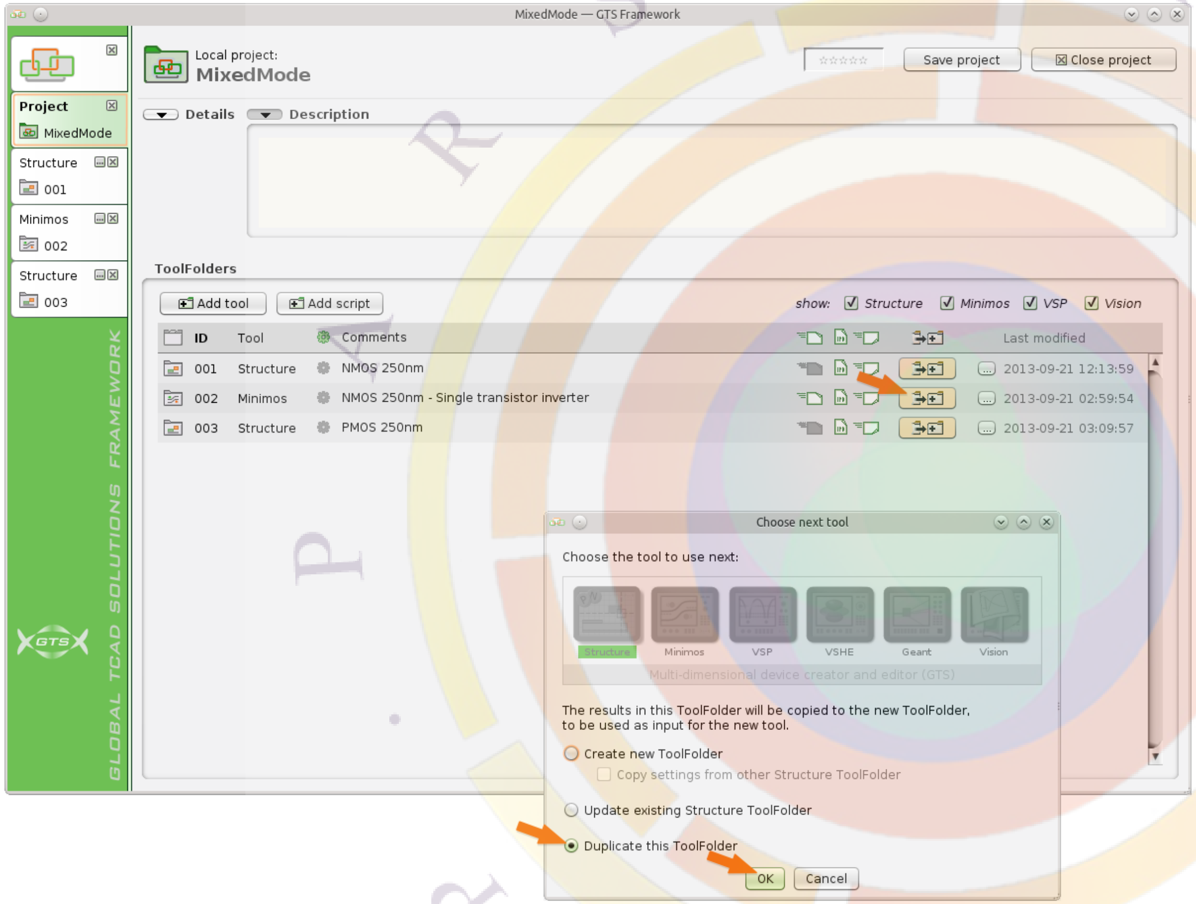
Now we will create a distributed PMOS transistor and modify the existing circuit to simulate the output voltage of a CMOS inverter.



2.1. Add Distributed PMOS

2.1.1. Create PMOS

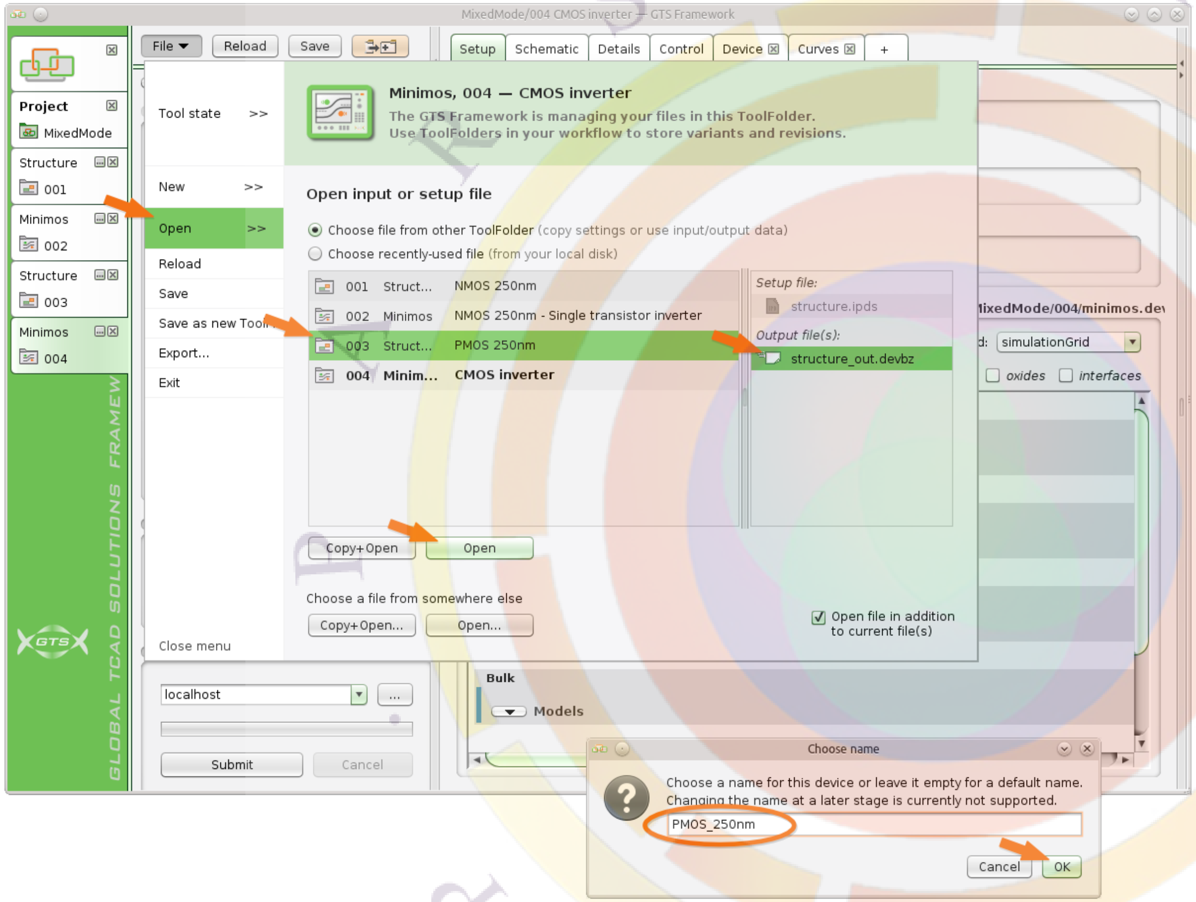
- Switch back to the project home in the tools column
- Use “Add tool” to add a ***Structure*** ToolFolder
- Create a 2D PMOS device with $0.250\,\mu\mathrm{m}$ channel length using ortho grid (`pmos_0250um_2d_ortho.ipd`)
- Use the button “Create device” to generate the device



2.1.2. Create Minimos-NT ToolFolder

In the following the circuit has to be modified, but as we want to keep the previous results we duplicate the ***mmnt*** ToolFolder:

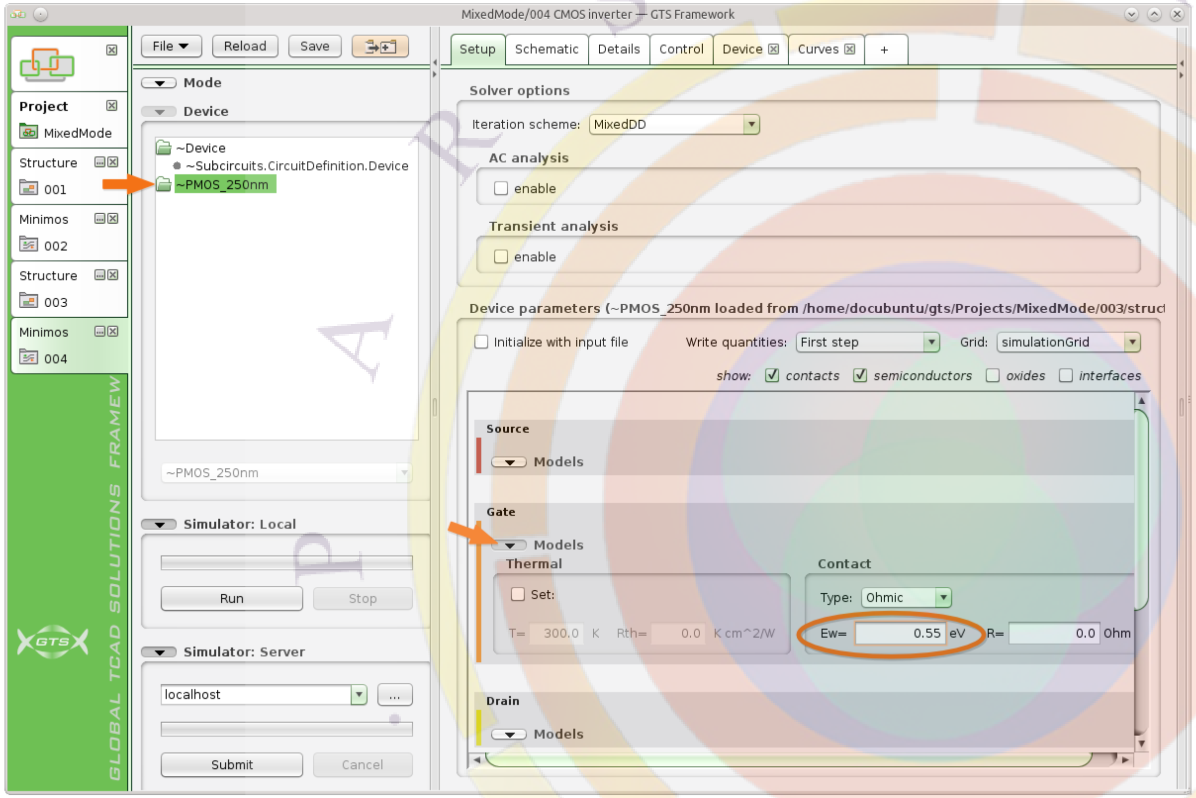
- Again switch to the project home in the tools column
- Use the "Next tool" button to duplicate the ***Minimos-NT*** ToolFolder



2.1.3. Load Distributed Device

We have to add the previously created PMOS:

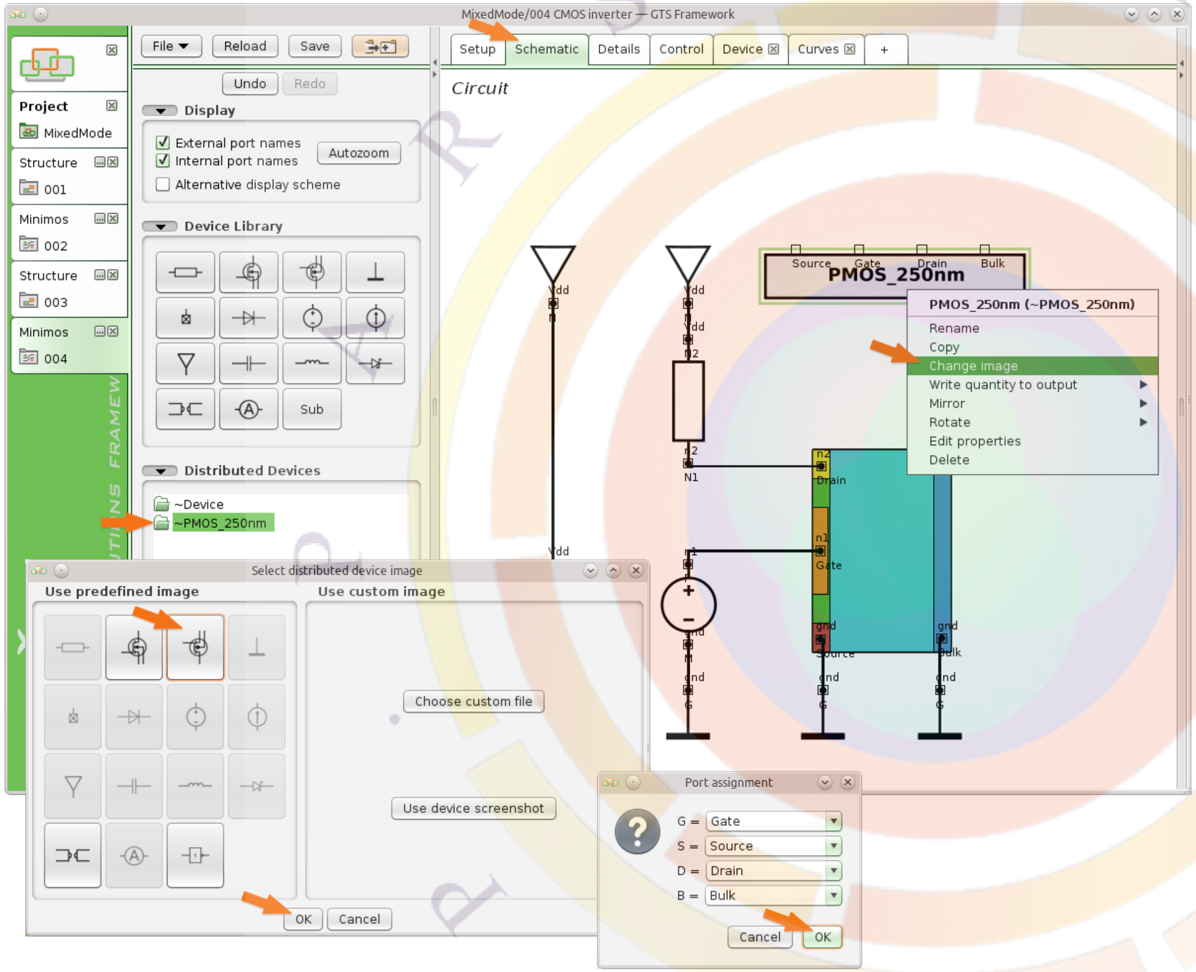
- Use the file menu in the control panel of the new ***Minimos-NT*** ToolFolder to open the structure file of the PMOS as shown in the figure
- Rename this device "PMOS_250nm"



2.1.4. Configure Distributed Device

The setup of the new device can be done as usual:

- Select the PMOS device in the "Device" section of the control panel
- Set the work function difference of the Gate contact of the PMOS transistor to $E_w = 0.55\,\mathrm{eV}$

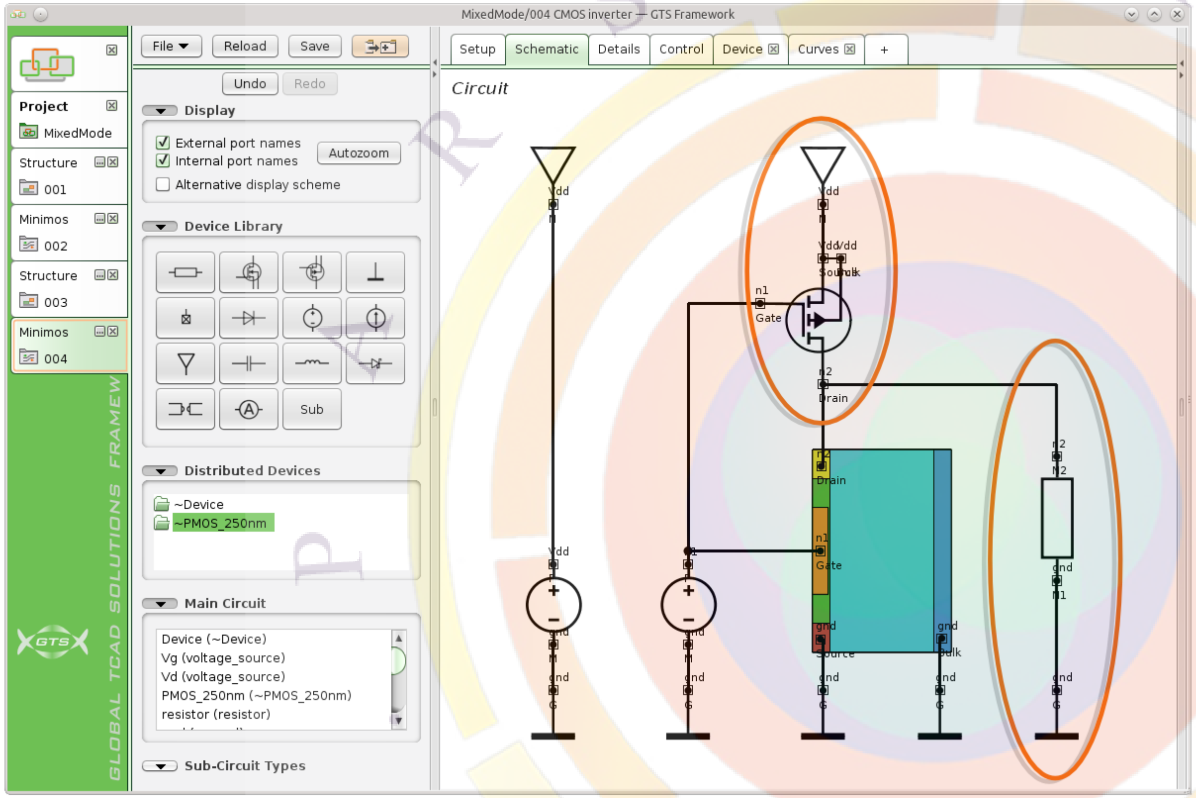


2.2. Edit Circuit

2.2.1. Add Distributed Device

- Switch to the "Schematic" page
- Left-click on "~PMOS_250nm" in the "Distributed Devices" section of the control panel
- Place it in the work area with a left-click as shown in the figure
- Right-click on the device
- Choose "Change image" in the drop down menu and select the schematic symbol of a PMOS
- Check the port assignment and confirm with "OK"

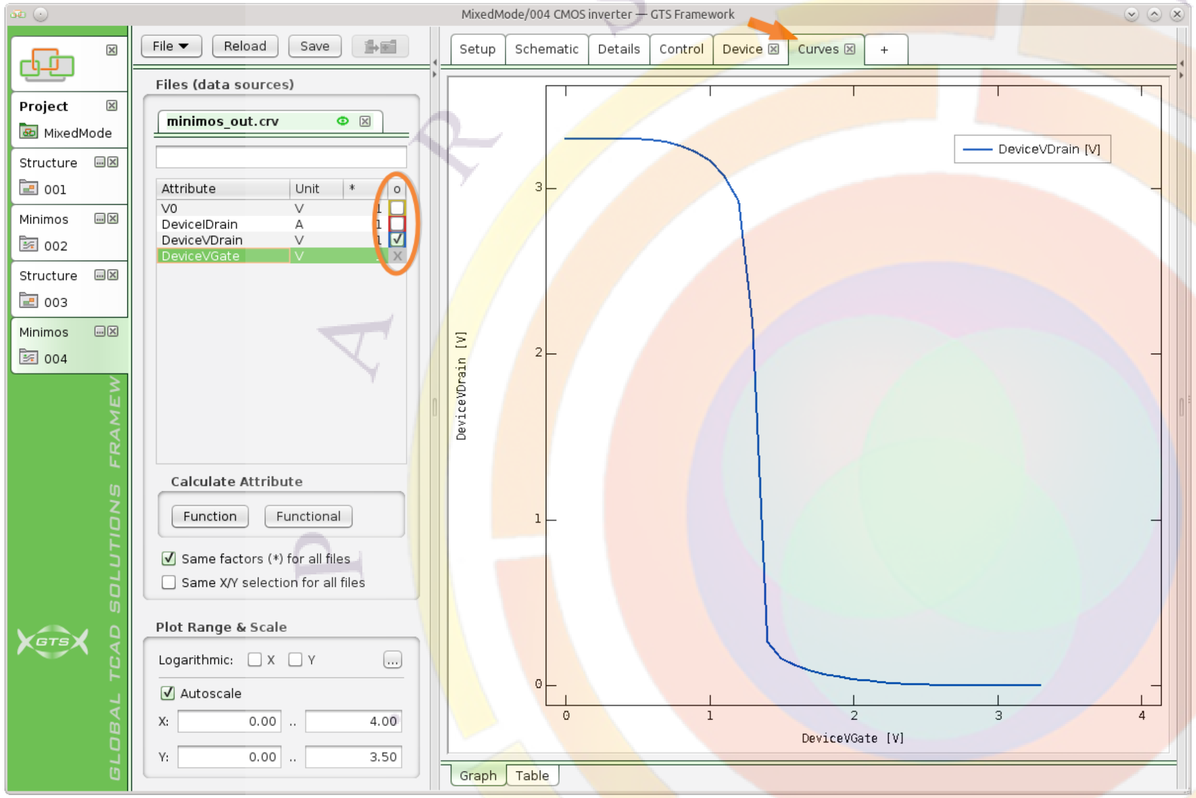
We will used the schematic transistor image this time, but one could use the device screenshot again as well.



2.2.2. Complete Circuit

- Add a GND node for the load resistor and
- Move and connect the components as shown in the figure

The output quantities are the same as before so we don't have to change or add any.



2.3. Simulate Output Voltage

2.3.1. Result: Output Voltage

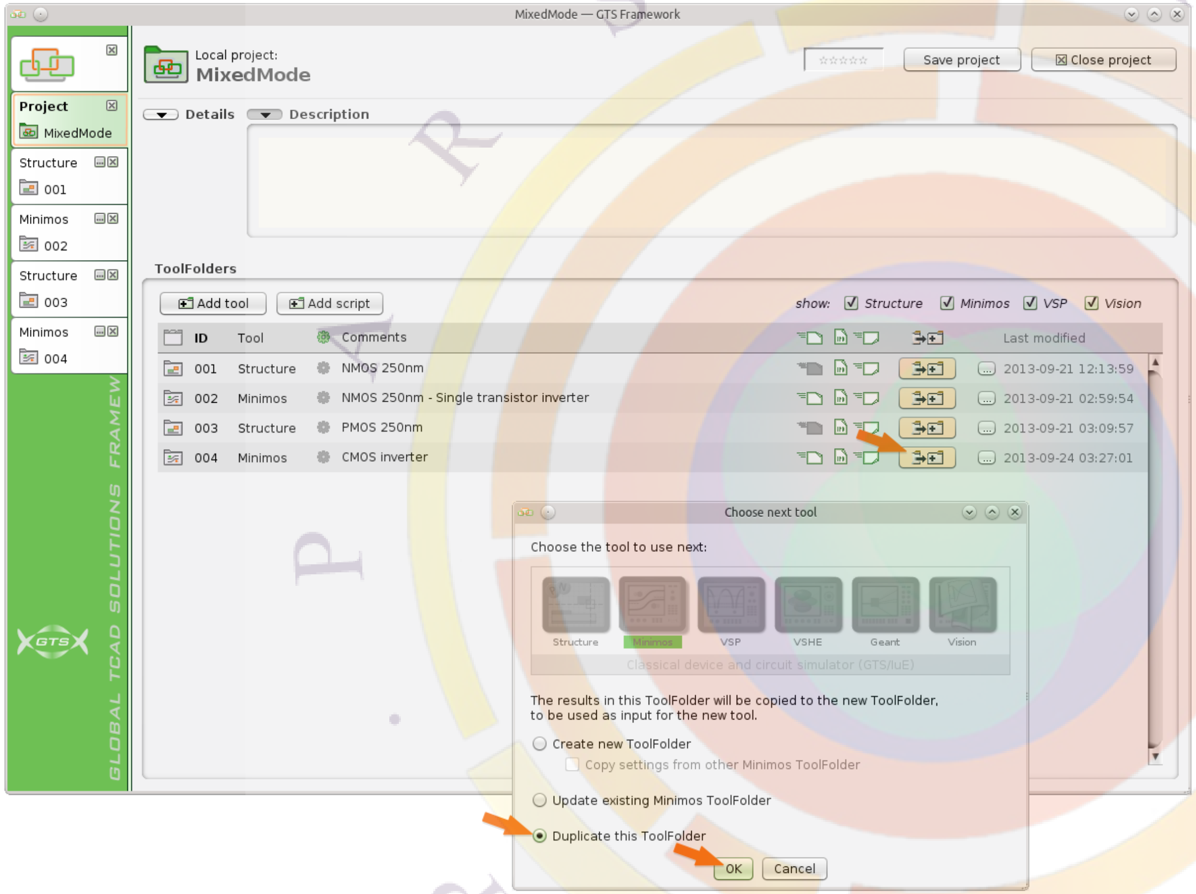
Now we can run the simulation:

- Switch to the "Control" page and press the "Run" button
- Wait until the simulation has finished and switch to the "Curves" page
- Assign "DeviceVGate" to the X-axis via its drop down menu
- Assign "DeviceVDrain" to the Y-axis by checking the corresponding check-box

Part 3

CMOS with Mixed Transistors

In this chapter we will replace one of the distributed devices with a ***Spice*** MOS Level 3 model. Again we will simulate the output voltage of the CMOS inverter.

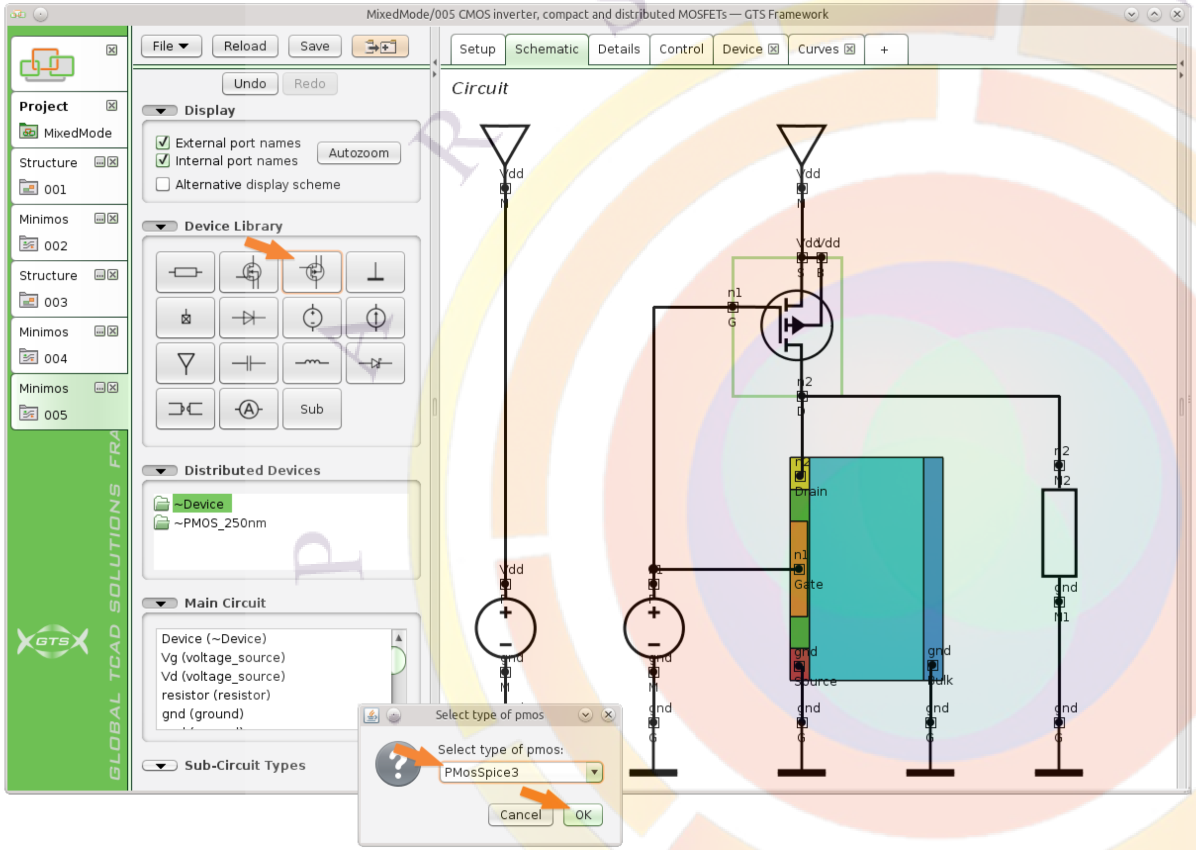


3.1. Edit Circuit

3.1.1. Create Minimos-NT ToolFolder

Again the circuit has to be modified, but we want to keep the previous results:

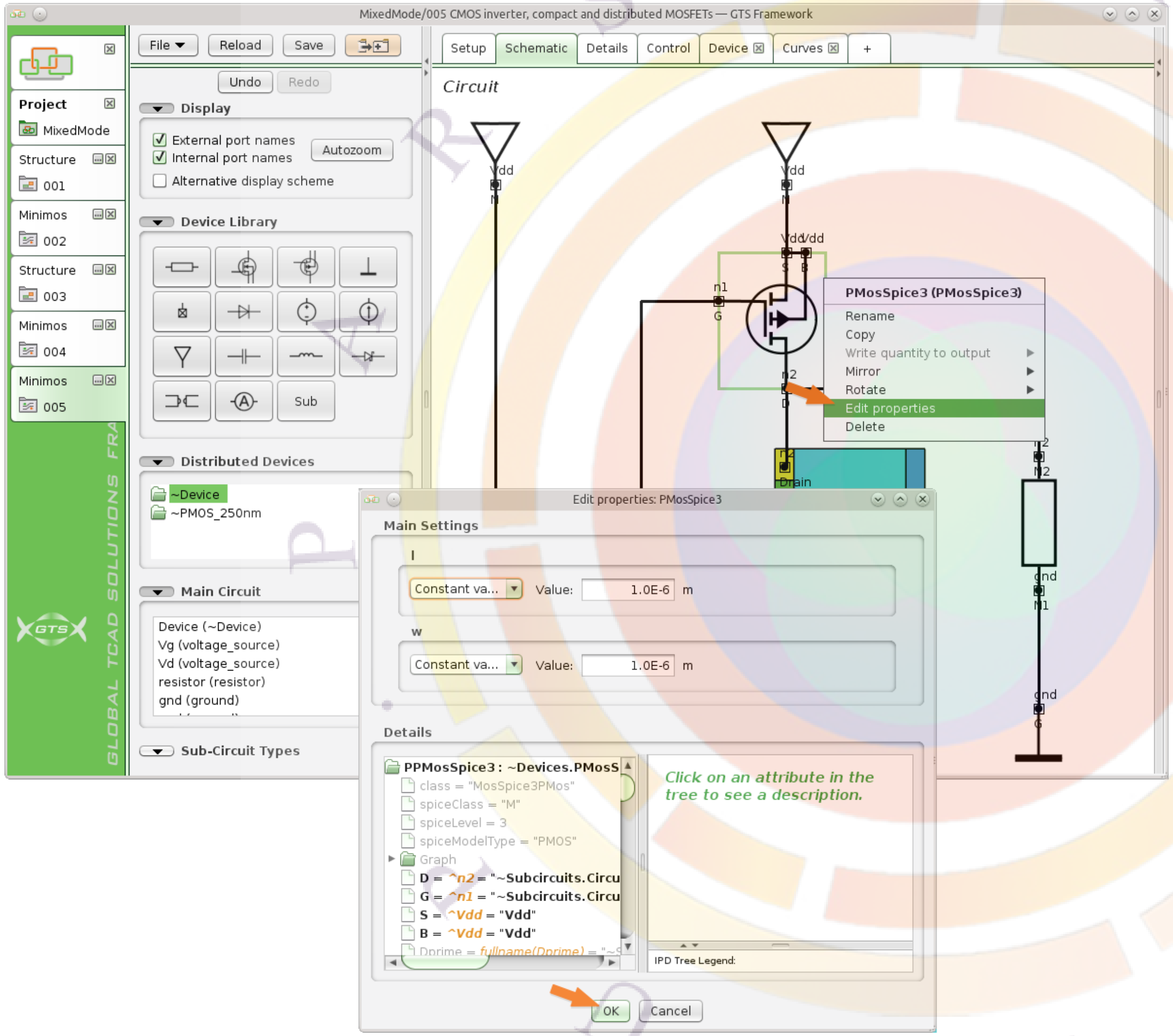
- Switch back to the project home in the tools column
- Use the "Next tool" button to duplicate the latest ***Minimos-NT*** ToolFolder



3.1.2. Add Compact Device

We want to replace the distributed PMOS with a compact model:

- Switch to the "Schematic" page
- Select the PMOS with a left-click and press "Del"
- Add a compact PMOS to the circuit by left-clicking on the corresponding symbol of the device library in the control panel
- Select the model "PMosSpice3" and confirm with "OK"
- Place and connect the new component as shown in the figure



3.1.3. Configure Compact Device

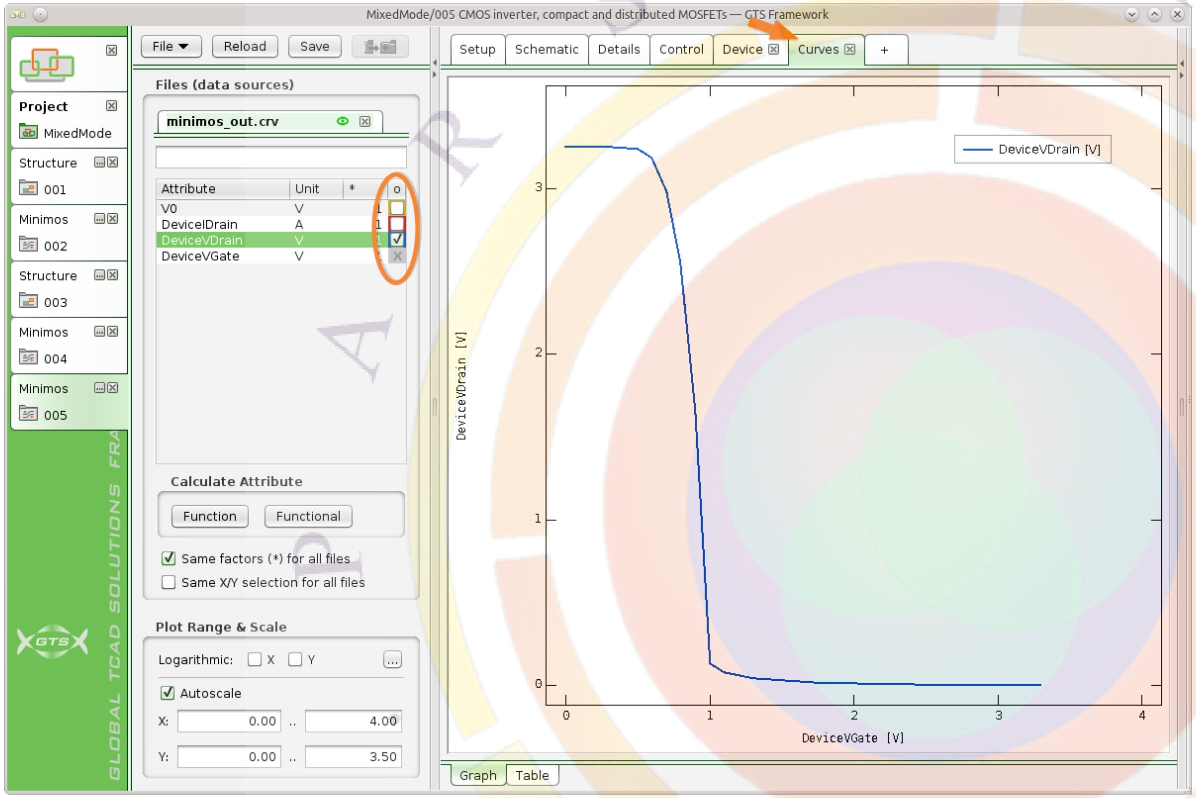
The ***Spice*** MOS level 3 model offers various parameters to determine its characteristics.

- Right-click on the new PMOS an select "Edit properties" in its drop down menu and take a look at the parameters. We don't have to change anything now, so just close the menu afterwards.

The section "Details" of this menu shows all parameters of this device. The most important parameters are also editable via the "Main Settings" section.

`l` defines the length and `w` the width of the device.

Notice that for a proper internal setup of the ***Spice*** model it is of relevance if a parameter is explicitly given by the user or not. So every parameter except the width `w` and the length `l` has a `...Given` parameter. Those `...Given` parameters have to be set to `true` if the corresponding parameter has been changed.



3.2. Simulate Output Voltage

3.2.1. Result: Output Voltage

Again we don't have to change the output parameters, so we can run the simulation:

- Switch to the "Control" page and press the "Run" button
- Wait until the simulation has finished and switch to the "Curves" page
- Assign "DeviceVGate" to the X-axis via its drop down menu
- Assign "DeviceVDrain" to the Y-axis by checking the corresponding check-box

Conclusion

In this tutorial, basic mixed-mode simulation capabilities of Minimos-NT have been presented. Distributed and compact devices have been used to simuate the output voltage of a single transistor inverter and a CMOS inverter.

As a follow-up, we recommend to continue with the *Mixed-Mode II* tutorial, covering transient analysis of a COMS inverter and an entire SRAM cell.

Further Reading

We welcome you to have a look at further *GTS tutorials* and *examples*, which you can open in ***GTS Framework***. Next to the basic ones included with the release, you can download more sophisticated tutorials and examples from MyGTS at `https://globaltcad.com/mygts`. Extracting the archives to your projects folder makes the tutorials visible in the projects list (highlighted yellow). Previews are provided at `http://www.globaltcad.com/en/solutions.html`.

For additional information, please refer to `http://globaltcad.com/` or feel free to contact us at `info@globaltcad.com`.

Appendix A

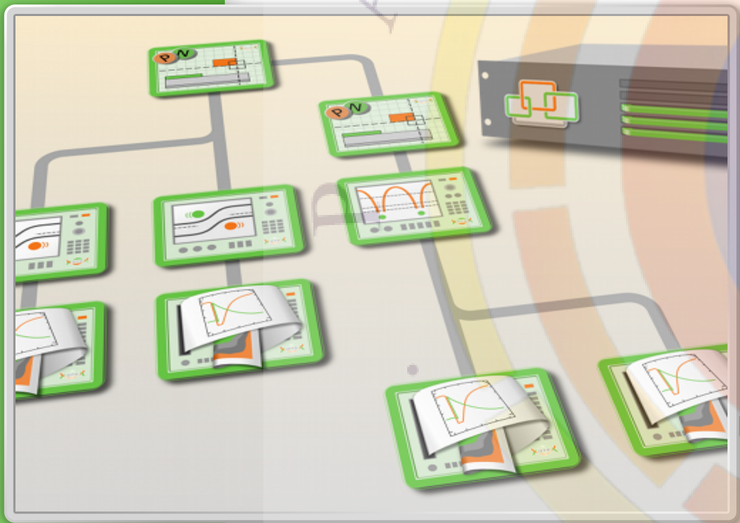
ToolFolder List

The project **MixedModeI** contains the following ToolFolders (TF):

TF	Tool	Description
T01	Structure	NMOS 250nm
T02	Minimos	NMOS 250nm - Single transistor inverter
T03	Structure	PMOS 250nm
T04	Minimos	CMOS inverter
T05	Minimos	CMOS inverter, compact and distributed MOSFETs

GLOBAL TCAD SOLUTIONS — GTS FRAMEWORK — TUTORIAL

GTS FRAMEWORK



DOE, Optimizer, Post-Processing

Tutorial

global TCAD solutions®

DOE, Optimizer, Post-Processing – Tutorial, GTS Framework Release 2014.09
Revision of September 29, 2017

Global TCAD Solutions GmbH
Bösendorferstraße 1/12
1010 Wien
Austria / Europe

Phone: +43 1 9255049
Email: info@globaltcad.com

Additional information is provided at: http://www.globaltcad.com/

Cover & layout: prausedesign

Contents

Introduction

This tutorial demonstrates the design-of-experiments (DOE) and parameter-fitting capabilities (optimizer) of ***GTS Framework***, which are aggregated in the *Scripting* tool. Its functionality is provided by the GTS JobServer.

In the second example, it illustrates the usage of the post-processing tool to show the influence of a device parameter to specific figures of merit of a MOSFET, such as V_{th}, I_{ON}, I_{OFF}, and the sub-threshold slope.

Using This Tutorial

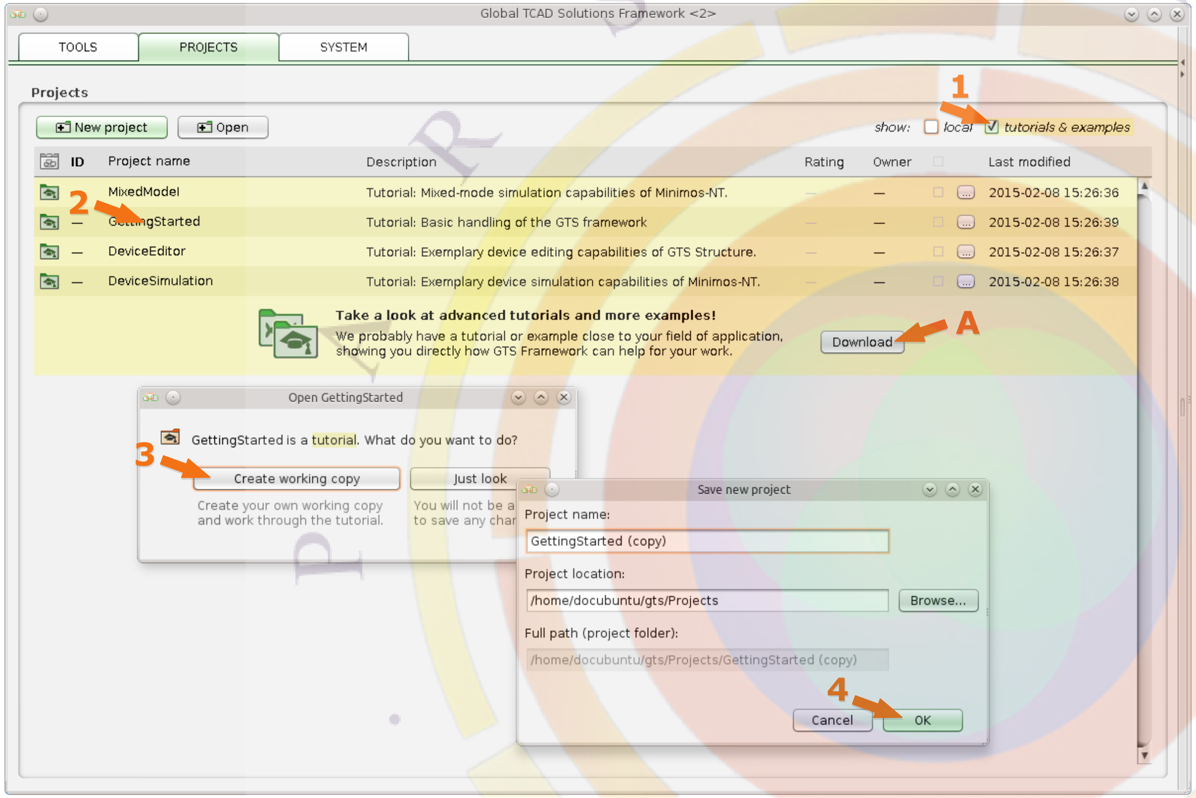
GTS *tutorials* generally consist of a written description (this text) and a GTS *project* (simulation data) which you can open in ***GTS Framework*** (yellow items in project list). As shown below, we recommend to create a working copy from the project, and proceed step by step, guided by this text. If not yet familiar, please refer to the *GettingStarted* tutorial.

Working in a copy of a tutorial project, you can open the yellow Txx ToolFolders at any time. They contain the data of the described simulations, which you can use for reference or as starting points for your own simulations.

Only a few tutorials are included with the release; the others are available via *MyGTS* at `https://globaltcad.com/mygts`. When logged in, please download the respective file and extract it to the gts projects folder on your PC. After restarting GTS Framework, the tutorial project will appear in the *Projects* list.

This is an intermediate (level 2) tutorial.

This tutorial was created using GTS Framework Release 2014.09. Other releases might need adjustments or have slightly different user interfaces.



Project / Working Copy

If you have not already done so, go to the *Framework Home* and locate the project associated with this tutorial in the *Projects* list, and create your working copy from it:

1. Check “tutorials & examples”
2. Click on the respective tutorial project: ***DOE, Optimizer, Post-Processing***
3. Choose “Create working copy”
4. Check the project name, click “OK”

The project is created and opened, so that you see the *Project Home*. — *Ready to start!*

A. If you miss the respective project, please download it via *MyGTS* (click “Download”, see previous page).

Alternatively, you can start the tutorial with an empty project – just create a new project. (The advantage of the tutorial project is that you have the results at hand any time.)

Part 1

DOE & Optimization

DOE and Optimization are implemented in the *Scripting* tool, which runs on the GTS Jobserver. Thus, the first chapter shows how to easily start a GTS Jobserver from the graphical user interface.

The second chapter will show how to design a very basic experiment for varying a structure parameter like the threshold dopant, which results in a shifted CV curve.

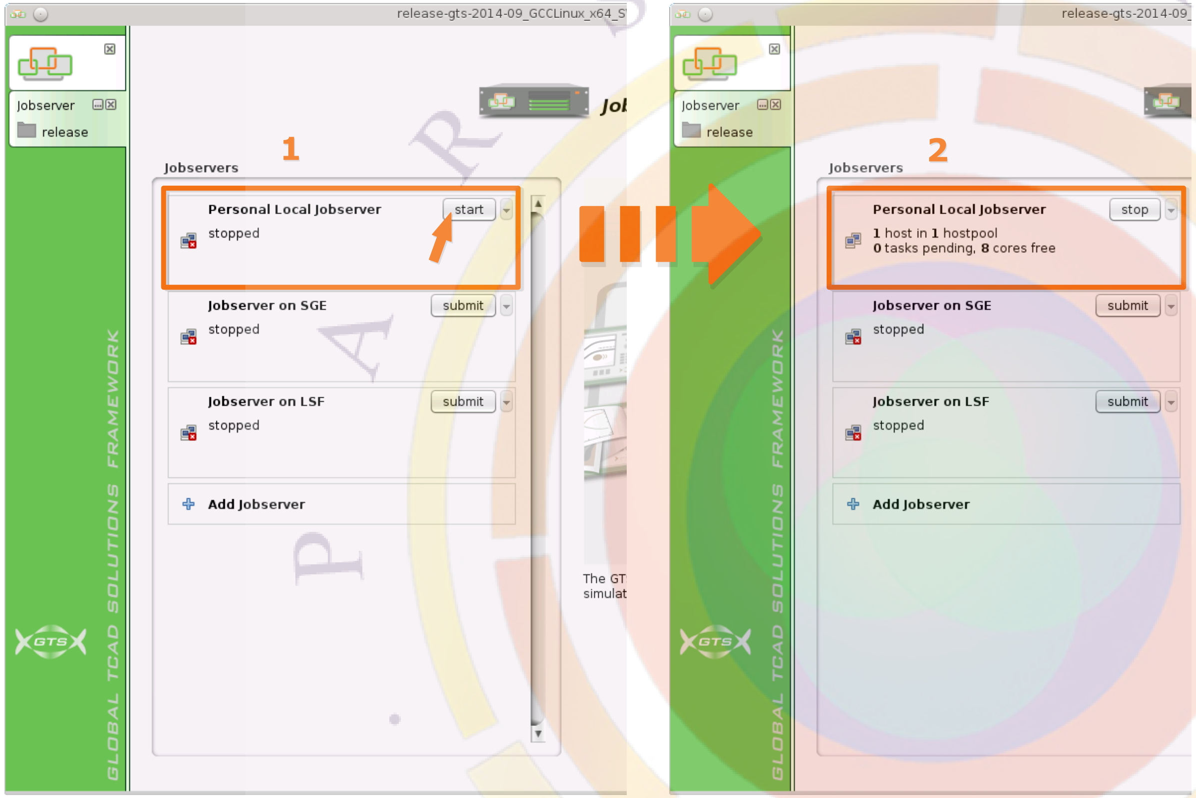
The third chapter will use the optimizer for parameter fitting. Exemplarily, two structure parameters will be used to fit to a measured CV curve.

1.1. Personal Job Server

GTS Framework allows to run a *Personal* Jobserver directly on your machine, which can be used by you exclusively to queue and run *complex* simulation jobs, such as jobs that include a specific DOE or use the optimizer.

1.1.1. The Jobserver Tool

1. At the *Framework Home*, click "Jobserver" to open the *Jobserver* tool

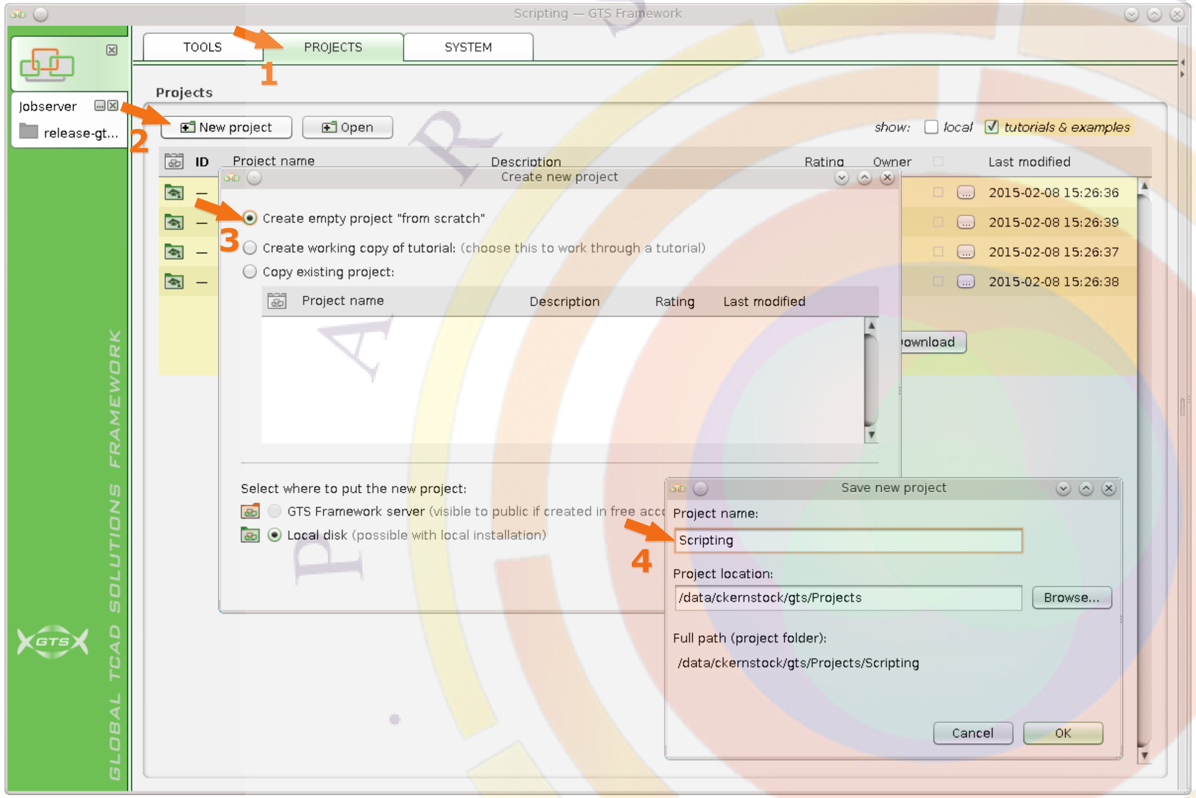


1.1.2. Jobserver

The Jobserver Manager shows a list of Jobservers you can start or stop. Per default, there is only the *Personal Local Jobserver* in the list, which is used in this example.

1. On the *Personal Local Jobserver*, click *Start*
2. After a short time, you should see that the Jobserver has started, as shown in the right-hand side part of the image

If you have a grid engine (SGE, LSF), a respective Jobserver item is shown in the list, and we recommend to use the Grid-Jobserver – click *Submit* on the respective list item.



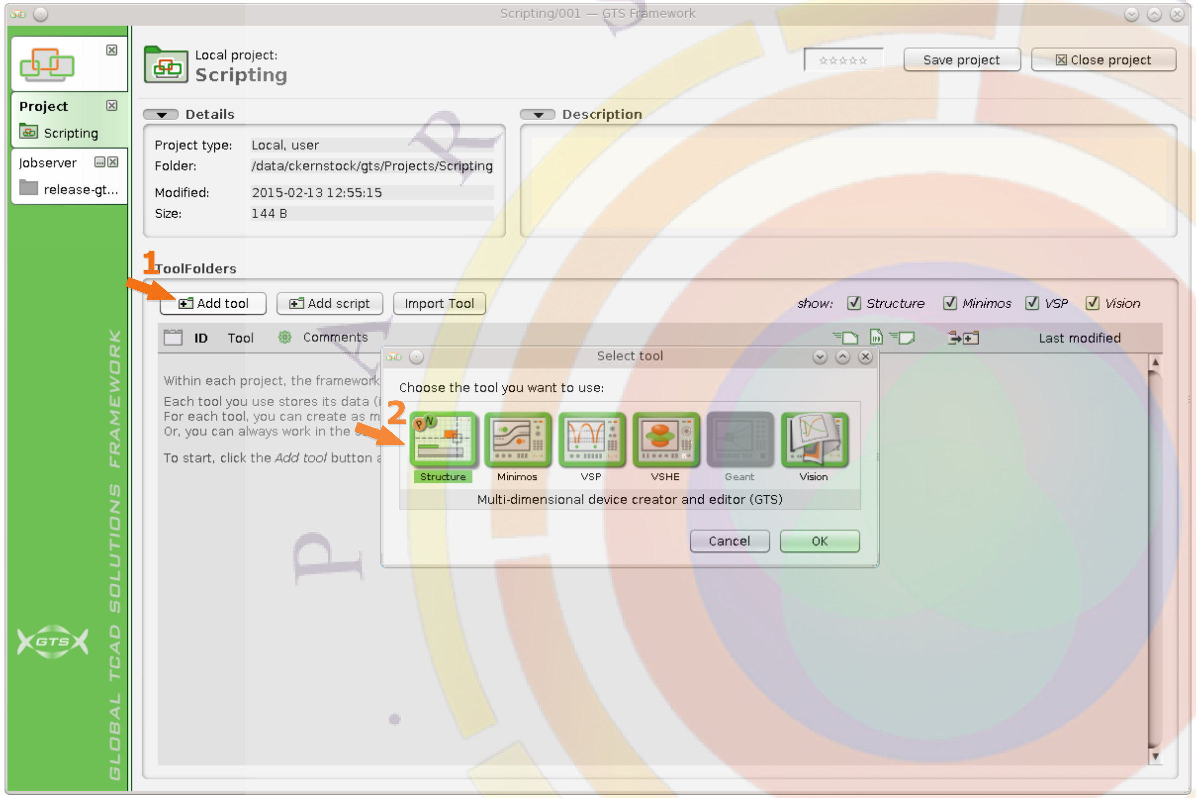
1.2. DOE: Parameter Stepping

This section shows the usage of the *Script* tool for designing experiments. For a simple example, we will study the effect of varying one structure parameter (the threshold dopant) on the CV curves.

1.2.1. Project

We recommend to use a working copy of the respective tutorial project, as explained in the introduction. If you have already done so, just switch to the *Project Home* and skip this step. Otherwise, please create a working copy (for details, please see the *GettingStarted* tutorial).

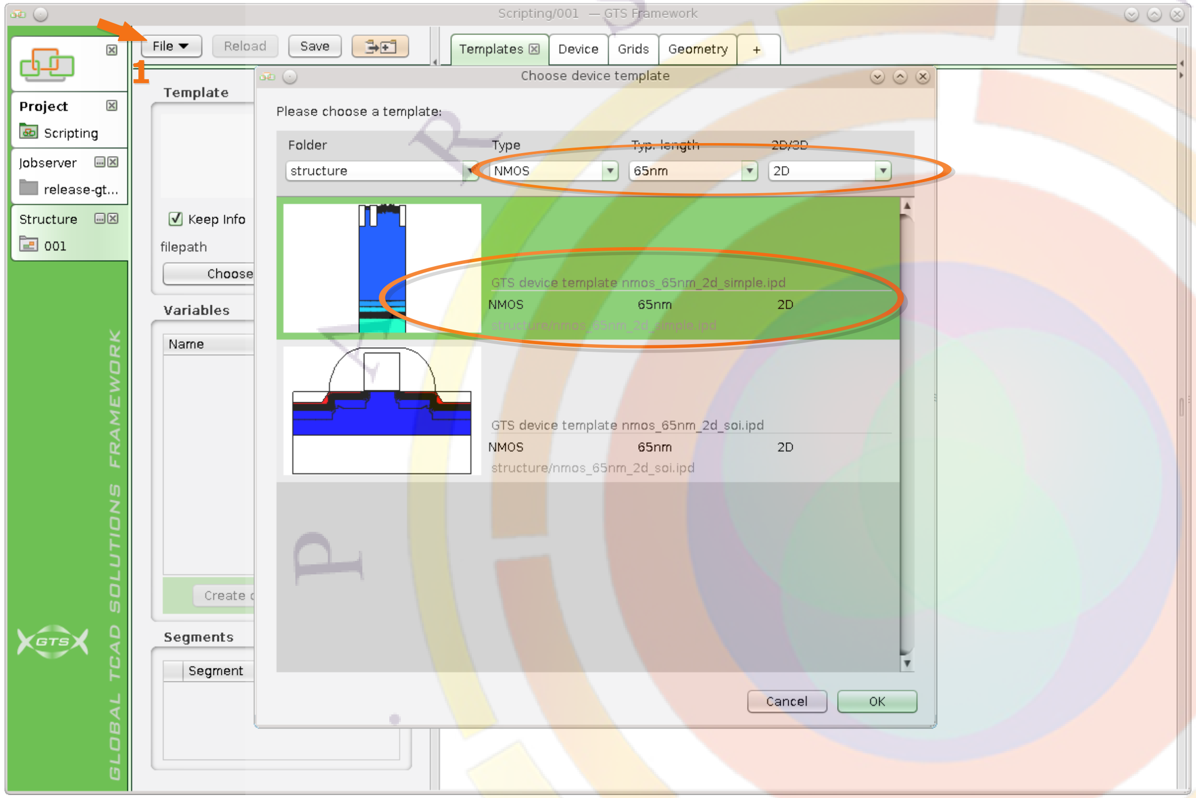
Alternatively, you can create a new Project, and name it as you like, f.i. “Scripting”.



1.2.2. Structure

First, let's use *Structure* to create a device from one of the GTS-supplied templates:

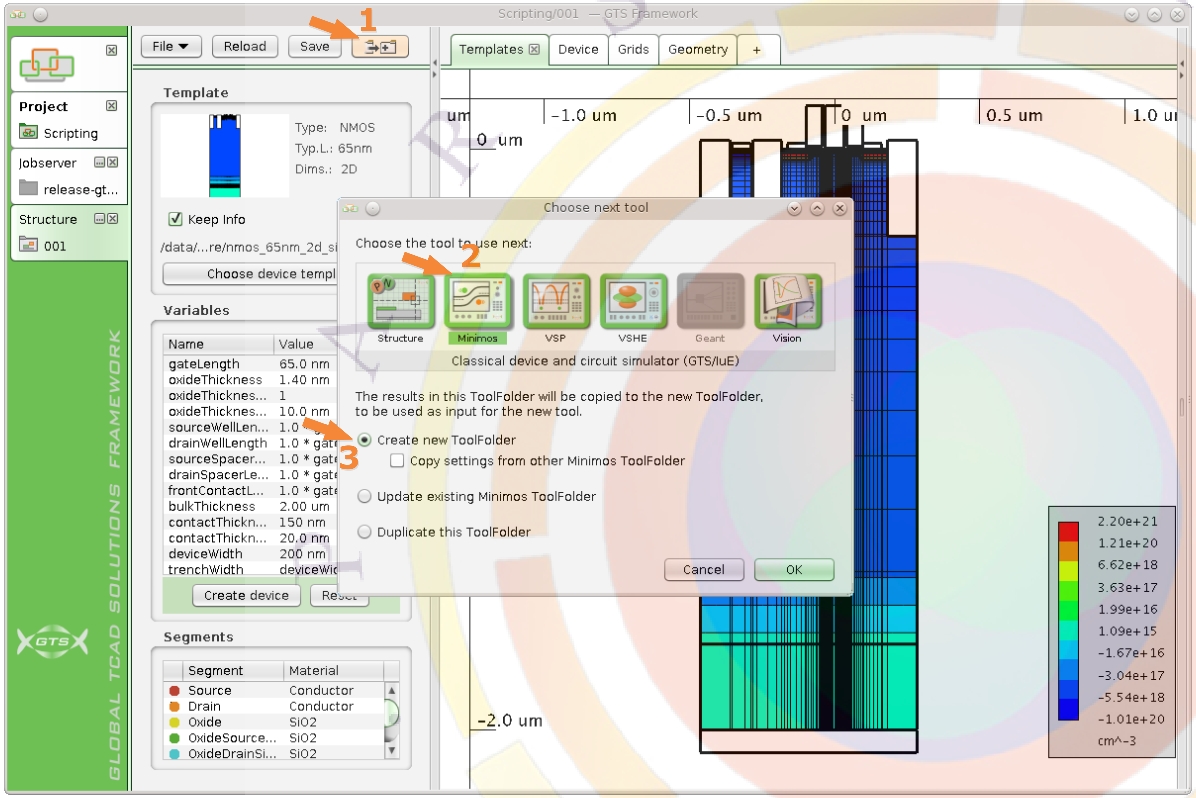
1. Click *Add tool*
2. Select *Structure* and click *OK*



1.2.3. Template

Create an 65nm NMOS FET from a template:

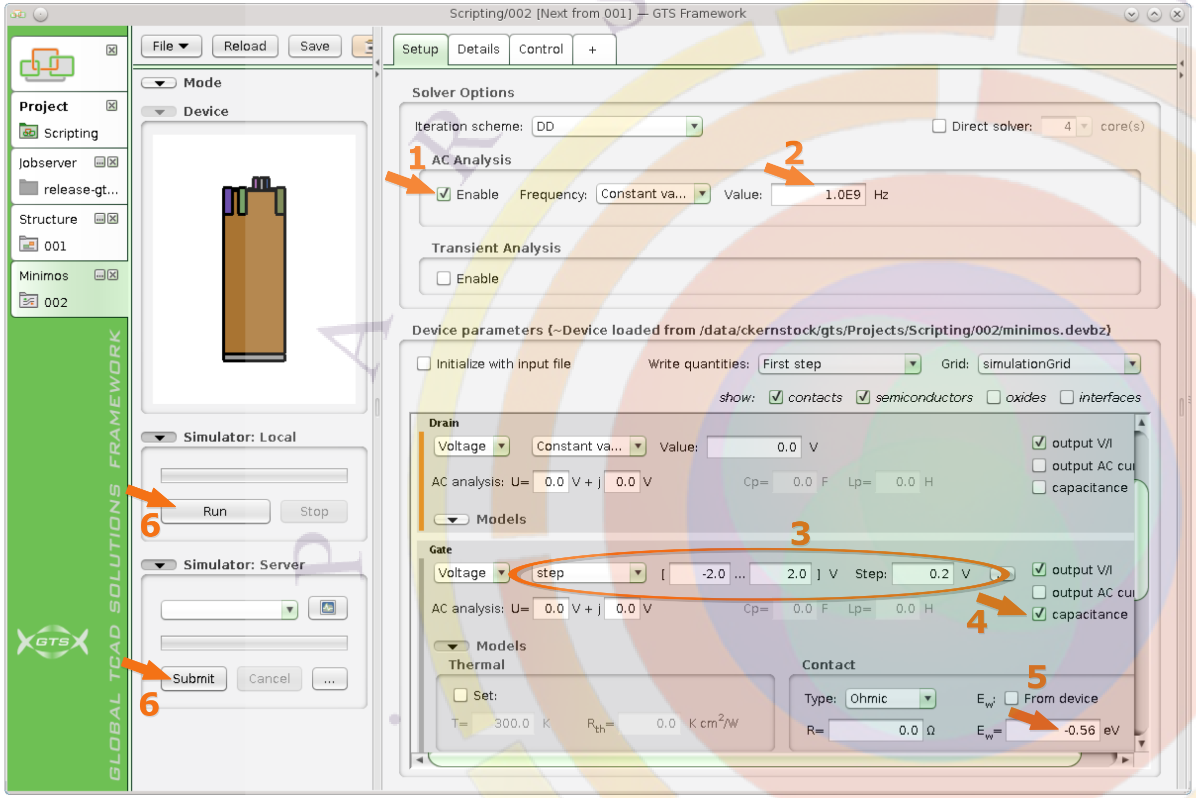
1. Open the *File* menu,
   - Choose *New*
   - Click *Create device from template*
2. Use the filters to search (NMOS, 65nm, 2D), and select the template `nmos_65nm_2d_simple` and click OK
3. Back in the *Templates* page view, click *Create device*



1.2.4. Next Tool: Minimos-NT

To set up the simulation, we create a Minimos-NT ToolFolder:

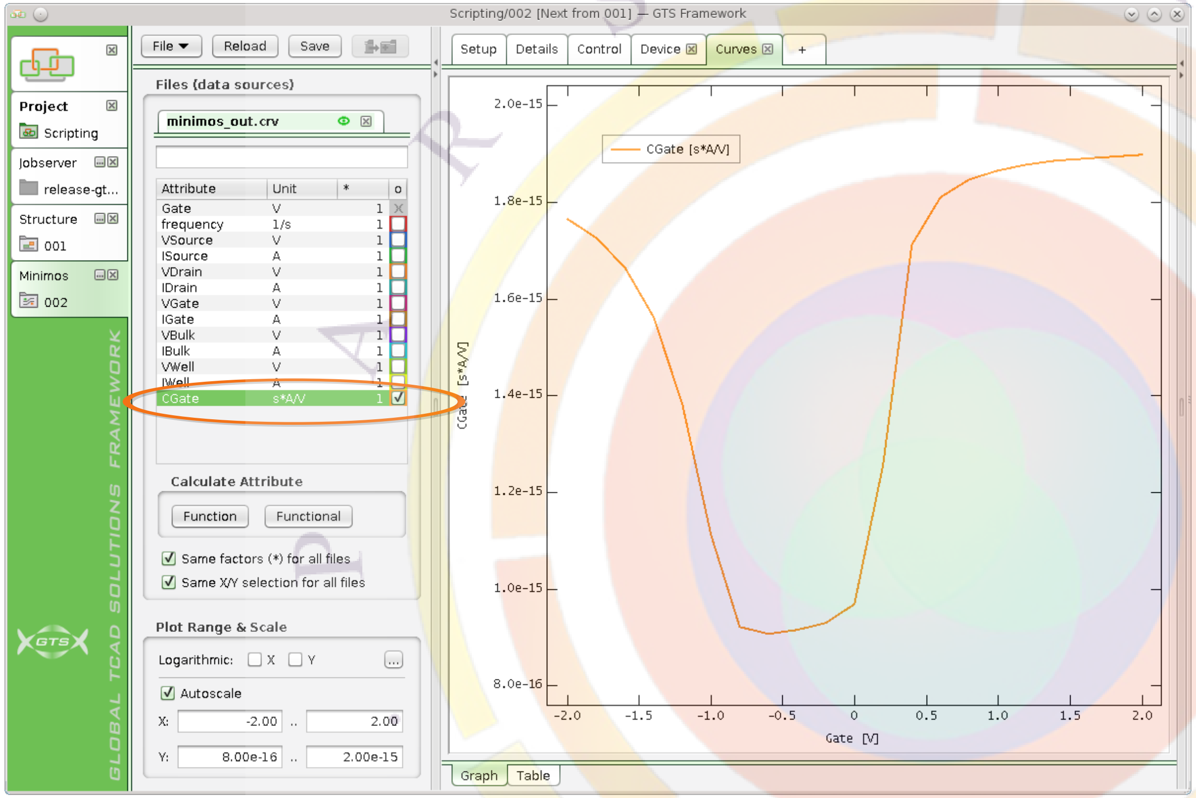
1. Click the *Next tool* button,
2. Select *Minimos* and click OK (or double-click *Minimos*)
3. Select *Create new ToolFolder*, click *OK*



1.2.5. Setup CV Curve

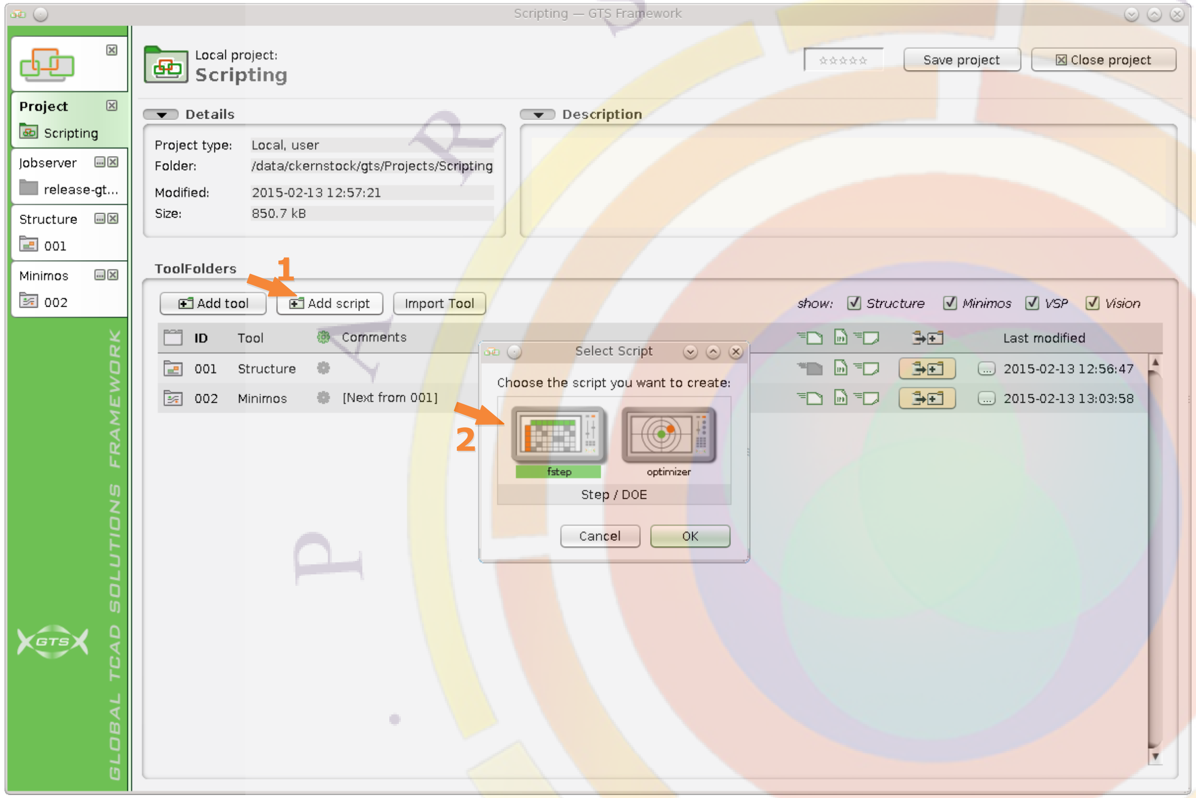
1. Enable *AC Analysis*
2. Set the frequency to 1 GHz
3. Step the gate voltage from -2 V to +2 V
4. Enable writing the capacitance
5. Set the work function difference to -0.56 V
6. Click Run or Submit.

If you click *Run* the simulation will start immediately on your machine. If you click *Submit* the simulation will be started via the Jobserver which may save resources on your local machine and allows to close the framework while the simulation is running.



1.2.6. Result: CV Curve

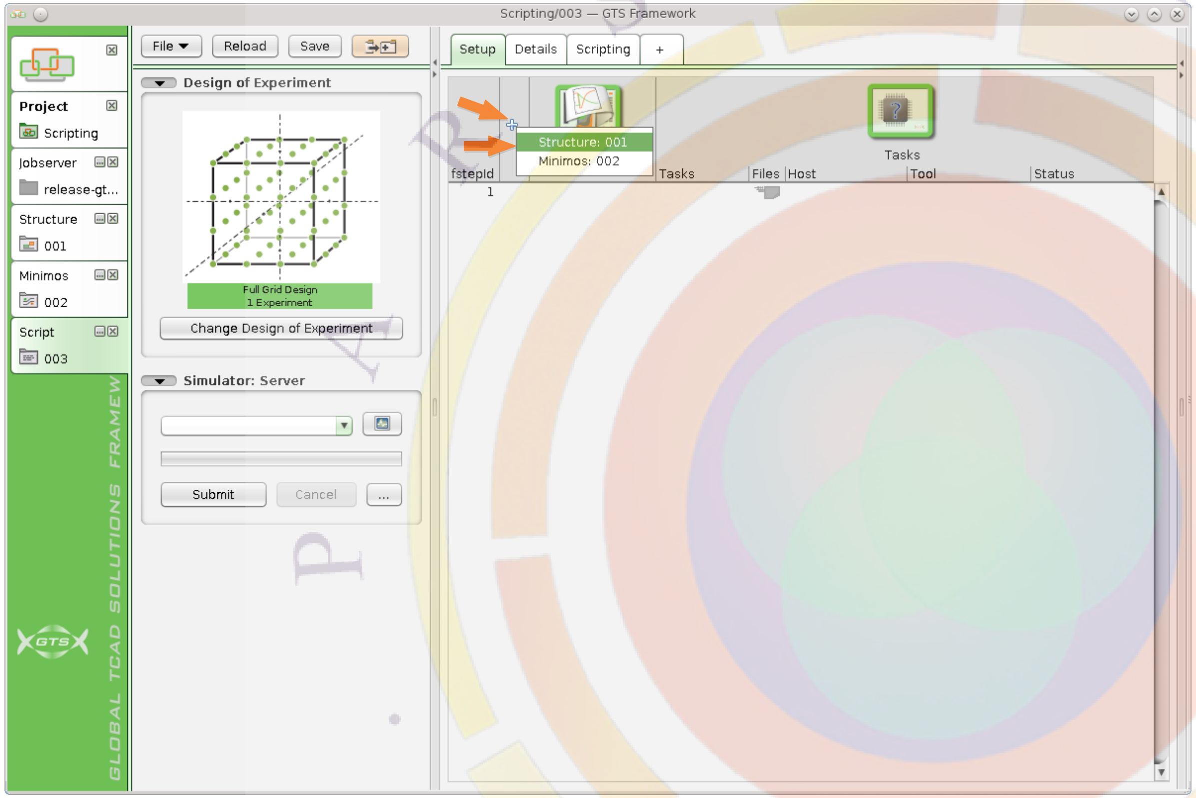
In the control panel of the *curves* page, select *CGate* to show the resulting CV curve.



1.2.7. Adding a Step/DOE Script

Now, we go back to the *Project Home*

1. Click *Add script*
2. Select *fstep* and click *OK*.



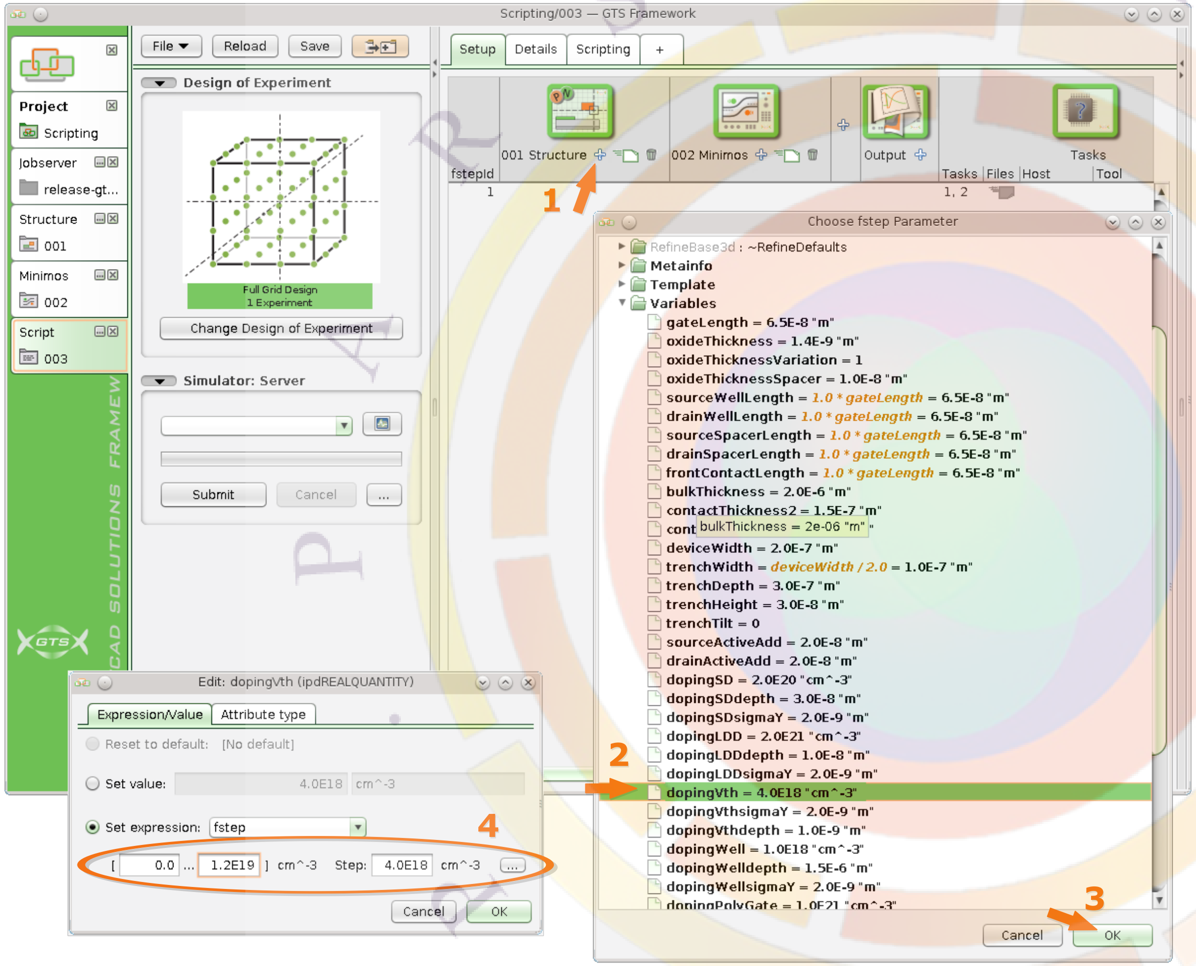
1.2.8. Script: Adding Tools

In the Script tool, the table allows to specify the experiment you want to do, i.e. the tools that will be called and the respective parameter sets.

The table header shows all ToolFolders that are used in the experiment, i.e. each column represents one tool. (It is pre-populated with a special *Output* tool for extracting and calculating results from crv file data.) The *Tasks* column is special, it will show the status of the tasks. Use the **+** button in the table header to add tools, or the trash icon in the tool header to remove it.

Our experiment (threshold dopant variation, CV curves) involves two Toolfolders (*001 Structure* and *002 Minimos*):

1. Click **+**
2. Choose *Structure 001*
3. Click **+** again, choose *Minimos 002*



1.2.9. Script: Parameters

Per default, each tool uses the parameter set specified in the respective ToolFolder. Here in the Script tool, you can specify parameters to be overwritten by the Script.

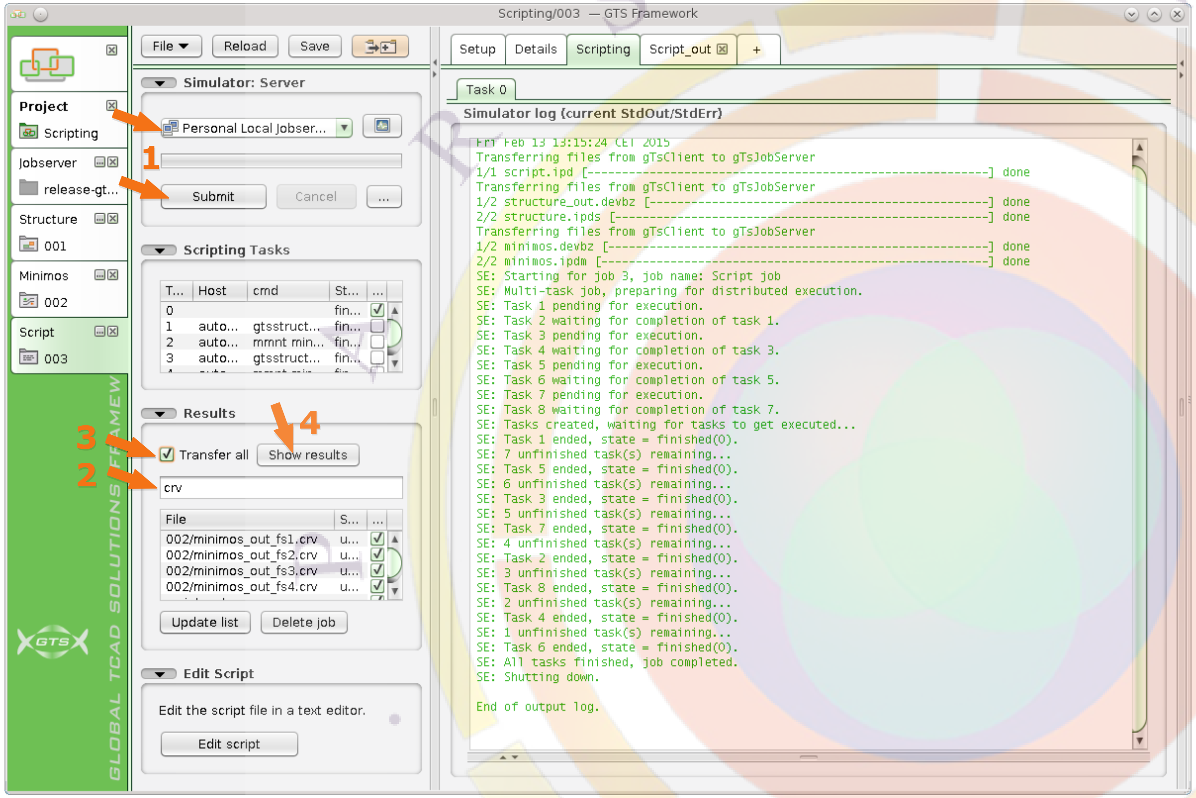
To choose a parameter, click the **+** button below the tool icon in the table header.

We want to vary *dopingVth*:

1. Click **+** below *001 Structure* to add a paramter for this tool
2. For the parameter to vary, select ~Variables.dopingVth
3. Click *OK*
4. In the next dialog, choose *fstep* from 0 to 1.2E19 cm^{-3} with a step size of 4E18 cm^{-3}

Note the second row in the table header below the tool icons: It shows the chosen parameters. To edit a parameter, click the yellow *Edit* icon. Use the *trash* icon if you do not want to use a parameter any longer.

For more complex experiments, you would probably add additional parameters for a tool by clicking **+** again.



1.2.10. Run Script

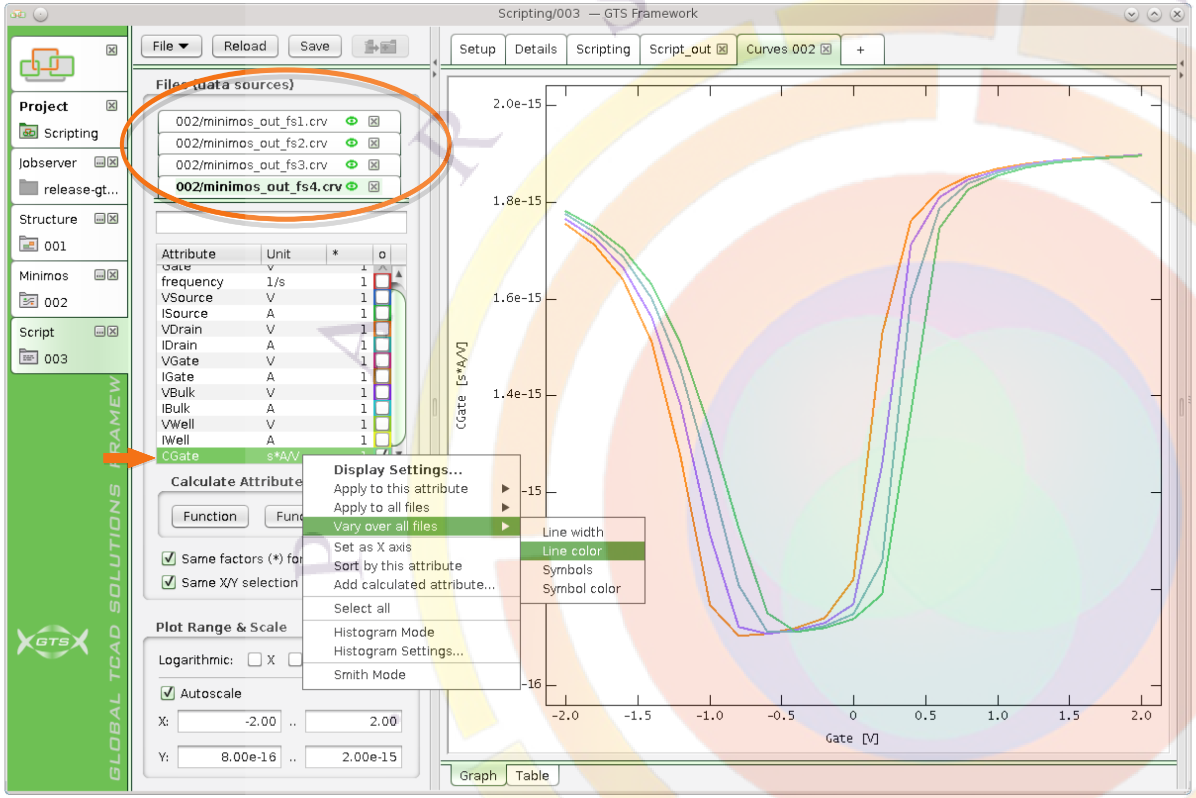
In the *Scripting* page, we can now run the Experiment:

1. Click *Submit* to run queue the simulation on the Jobserver

You should now be able to watch the log of the running job and its tasks in the working area at the right.

2. After/while running, set fiter to "crv"
3. Enable *Transfer all*
4. Click *Show results*

You can press the *Show results* button while the simulation is still running. In this case, results will be visible as far as they are available.

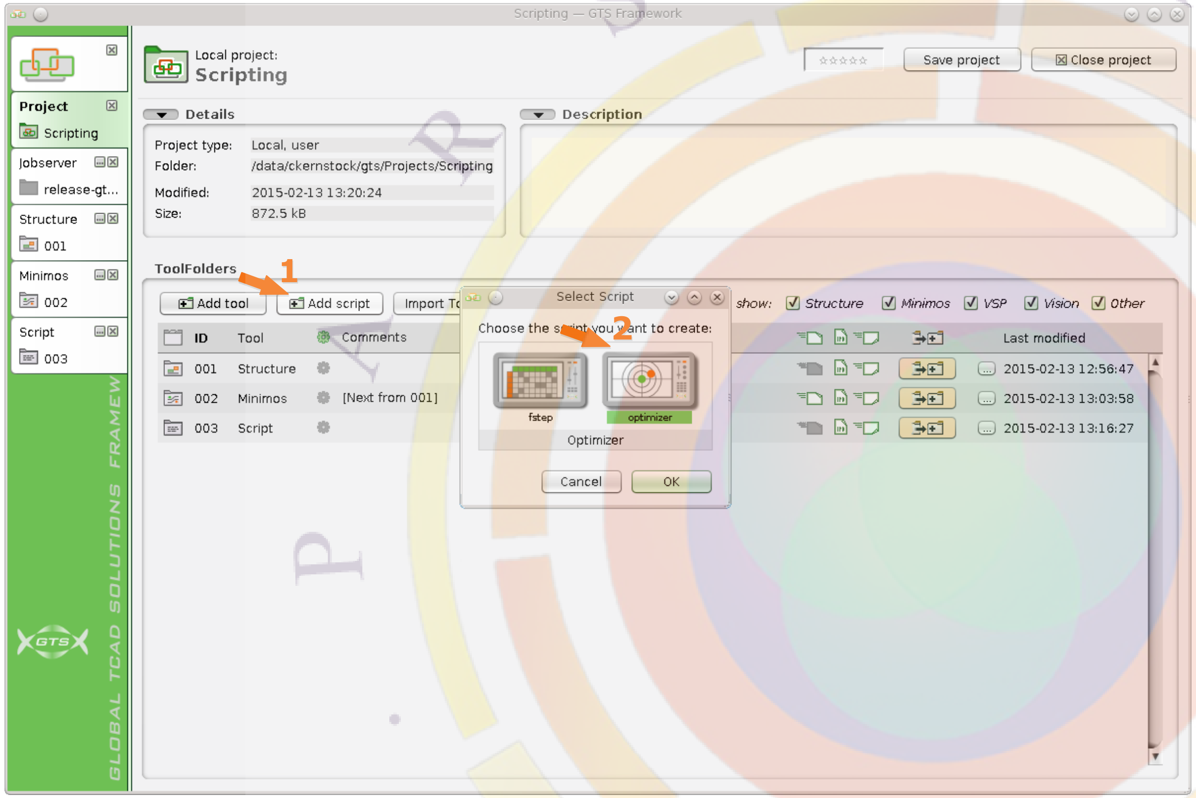


1.2.11. DOE Result: CV Curves

When the job has finished, the *Curves* page shows the resulting CV curves. At the top, the control panel shows the results of each tool invocation. Clicking on the green eye allows to show/hide the curves of the respective simulation.

In the table, right-click *CGate* and choose to vary the colors over all files (for details see the *GettingStarted* tutorial).

We now see the CV curves for varying threshold doping concentration.



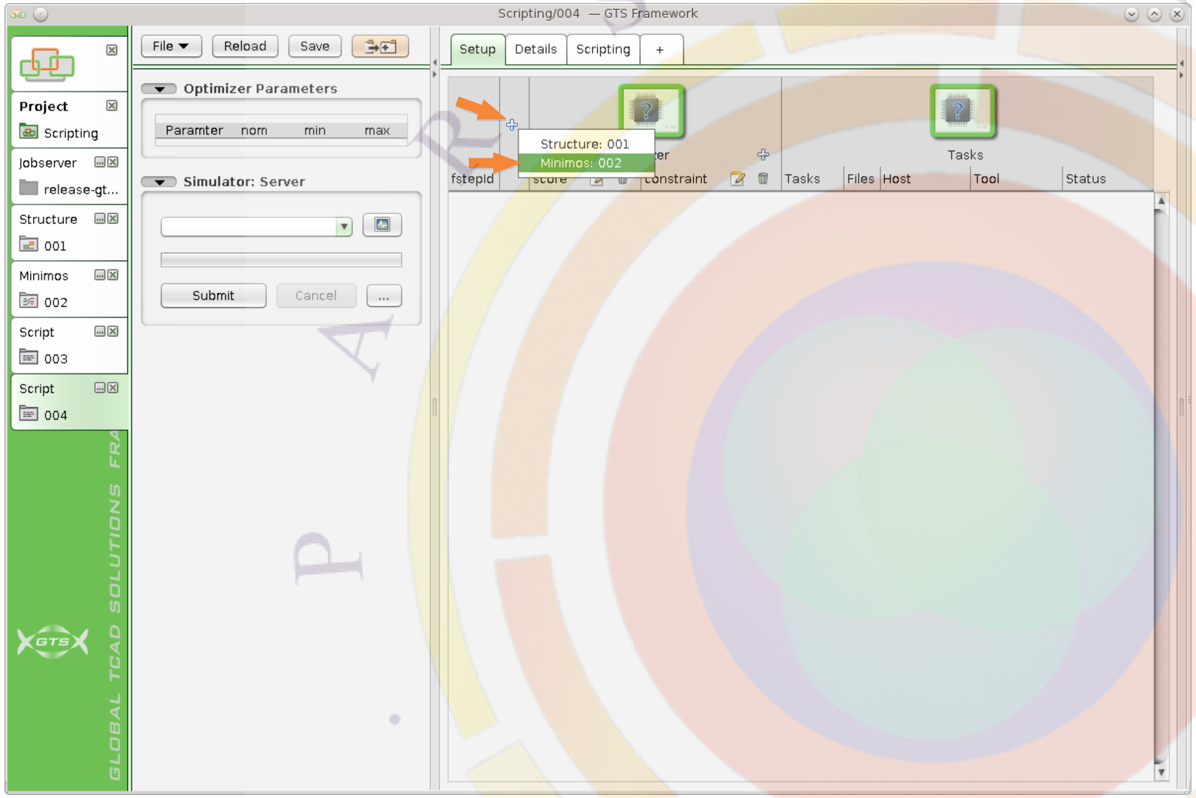
1.3. Optimizer: Parameter Fitting

Usually measurements are used to calibrate a TCAD model. This section demonstrates this use case in a simplified example. It will modify two structure parameters – threshold dopant and oxide thickness – to fit a given CV curve.

1.3.1. Adding an Optimizer

To Create an Optimizer Script, switch back to Project page. Then:

1. Click *Add script*
2. Choose *optimizer* and click *OK*

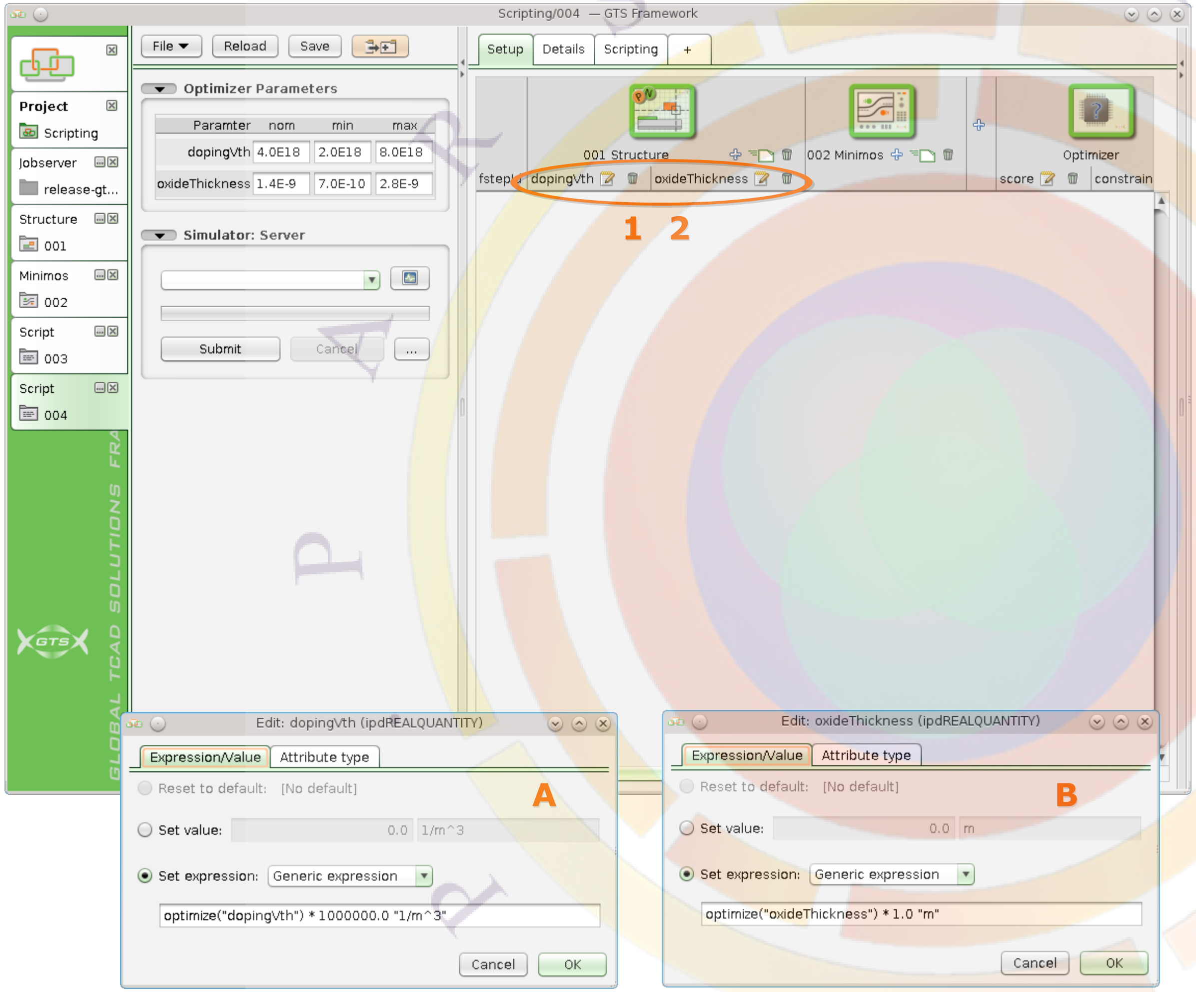


1.3.2. Script: Tools

For the parameter fitting, two Toolfolders are involved:

- Structure 001.
- Minimos 002.

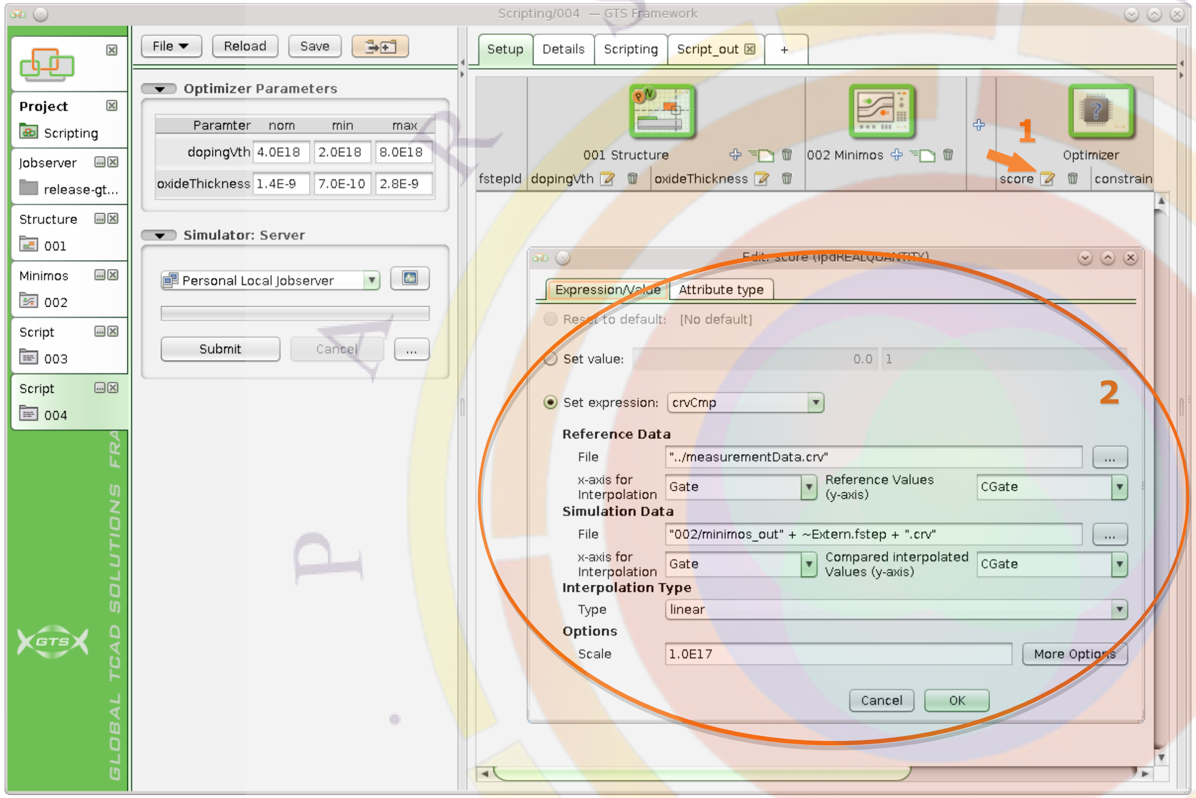
Add these ToolFolders to the Script, as we did in step 1.2.9.



1.3.3. Script: Parameters

For the *Structure 001* ToolFolder, we add two parameters. To do so, click the **+** icon below the *Structure 001* tool (as we did in step 1.2.9). Alternatively you can add both parameters at the same time via multiselect (using the *Ctrl* key). Choose the parameters:

1. ~Variables.dopingVth ()
   A – Accept default values from dialog
2. ~Variables.oxideThickness
   B – Accept default values from dialog

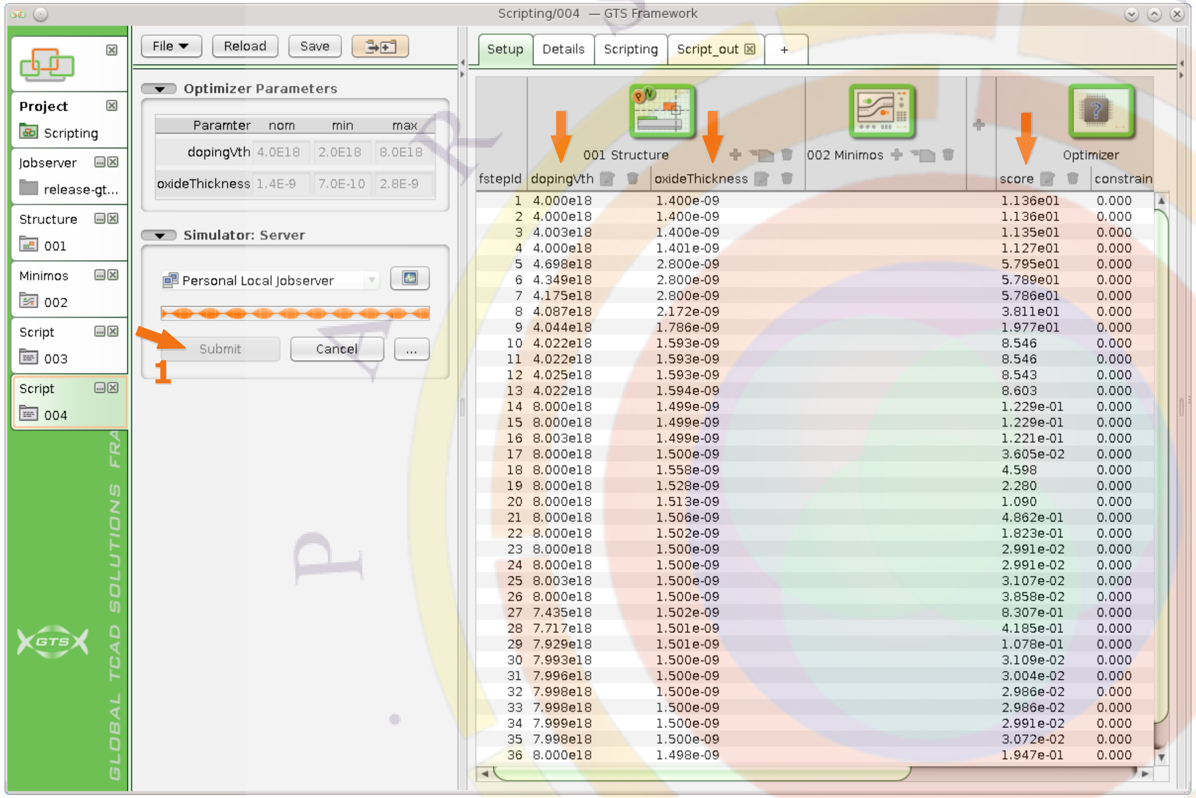


1.3.4. Edit Score Function

The optimizer minimizes a given *score* function by varying the input parameter. Thus this function has to be specified:

1. Click the *Edit* icon next to *score*
2. Set parameters as shown in the dialog
   - The file measurementData.crv is delivered in the base directory of the tutorial.

Use the ... button to search for the files. The ~Extern.fstep will be inserted automatically and will be replaced by the fstep file postfix while running the optimization.



1.3.5. Submit

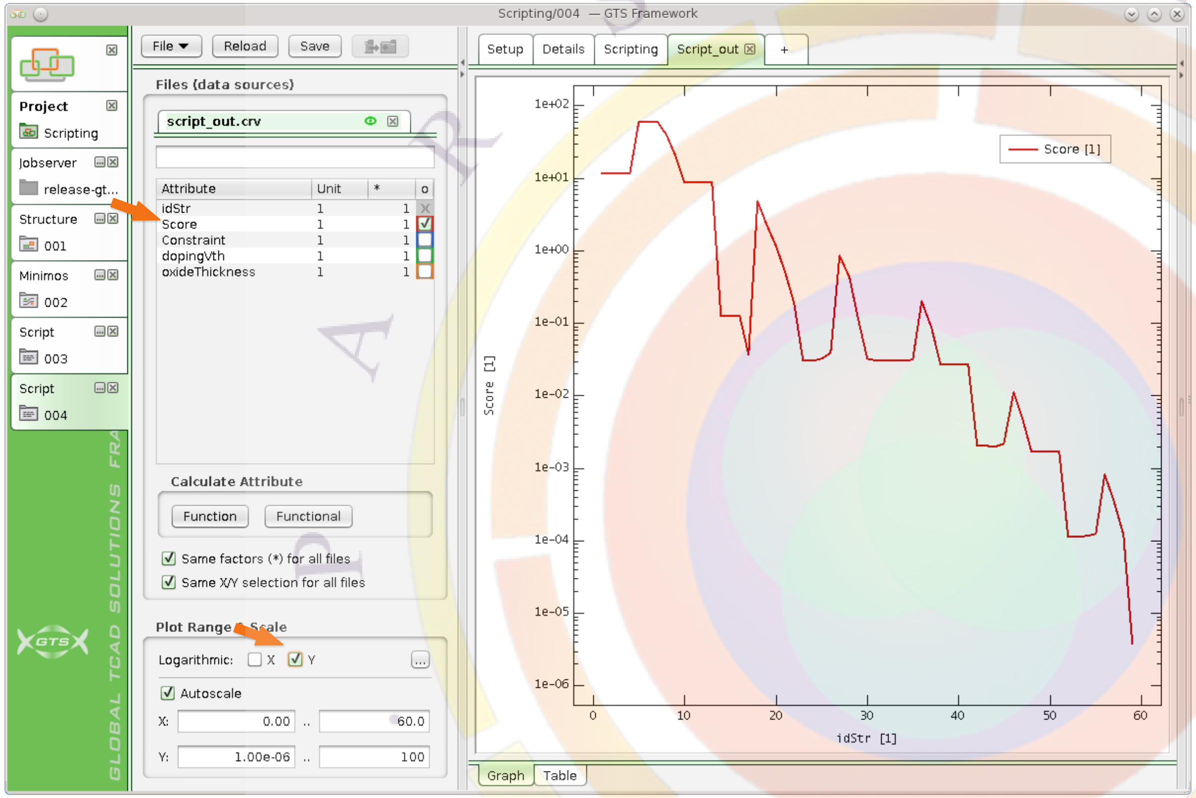
Now, let's submit the experiment to the Jobserver:

1. Click *Submit*

Watch the results shown in the columns.

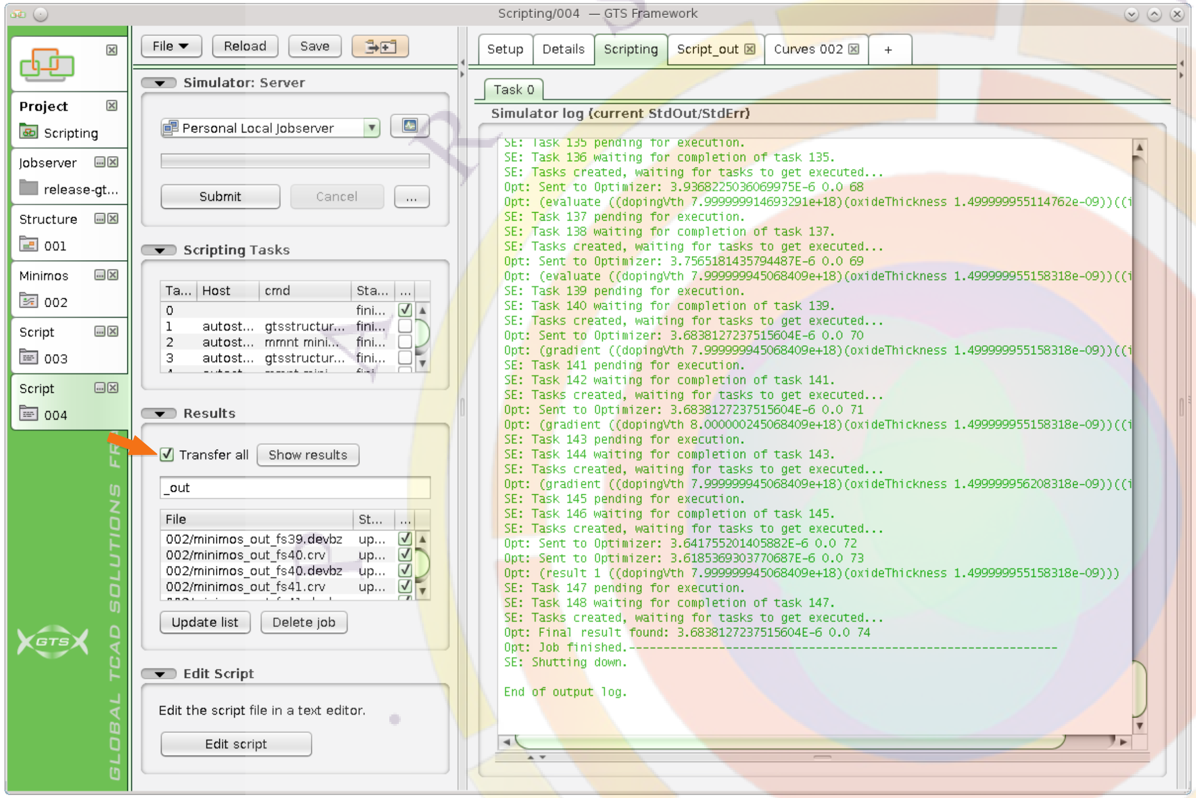
You can continue the tutorial or other work in ***GTS Framework*** while the Jobserver performs its work.

You can close/open ***GTS Framework*** while the job is running.



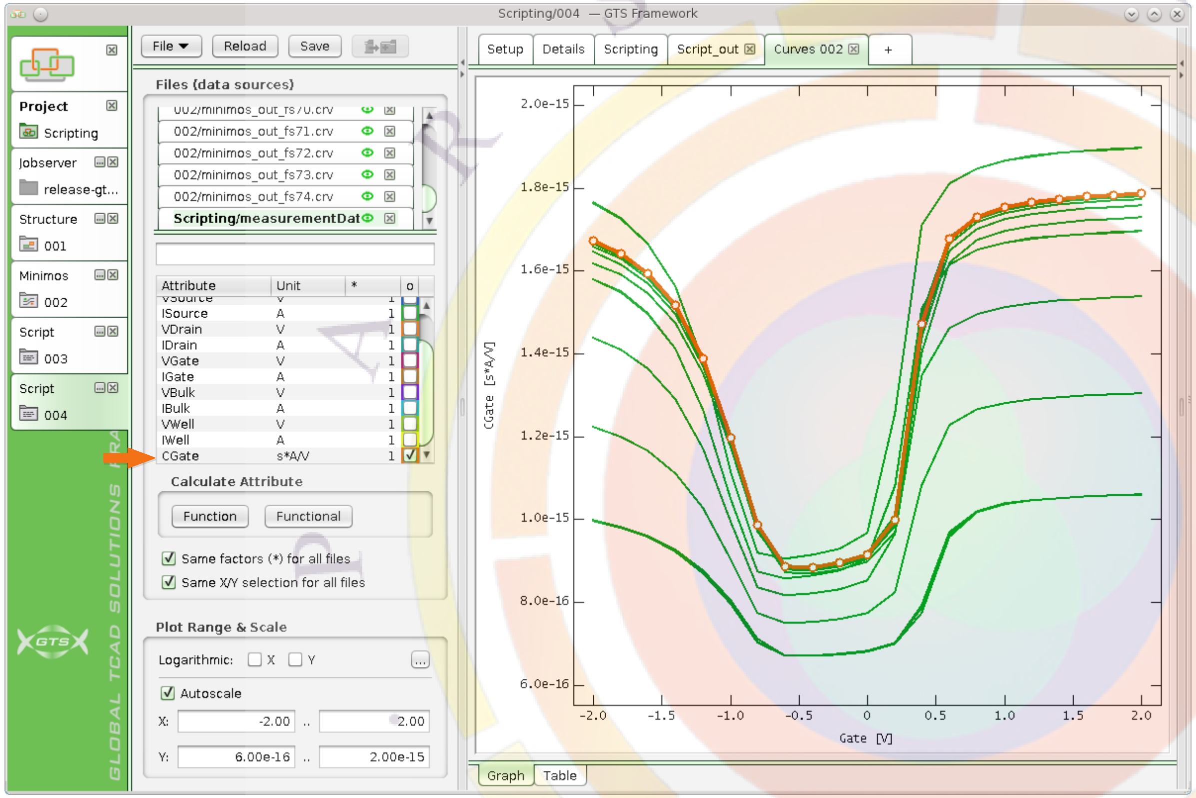
1.3.6. Score

On the *Script_out* page, the achieved score is shown.



1.3.7. Result

To retrieve all result files from the Jobserver, select *Transfer all* and click *Show results*.



1.3.8. Optimizer Result

We can now see the resulting curves of the optimizer steps, approaching the measured curve from the file `measurementData.crv`; in the screenshot, the calculated curves are colored green, whereas the measured curve is colored orange.

To display the final result more clearly, we can also create a comparison plot of the final optimized curve against the measured data.

To show the `measurementData.crv` on top close the file and reopen it via File → Open...

For details on how to compare two curves, please refer to the *Getting Started* tutorial.

Part 2

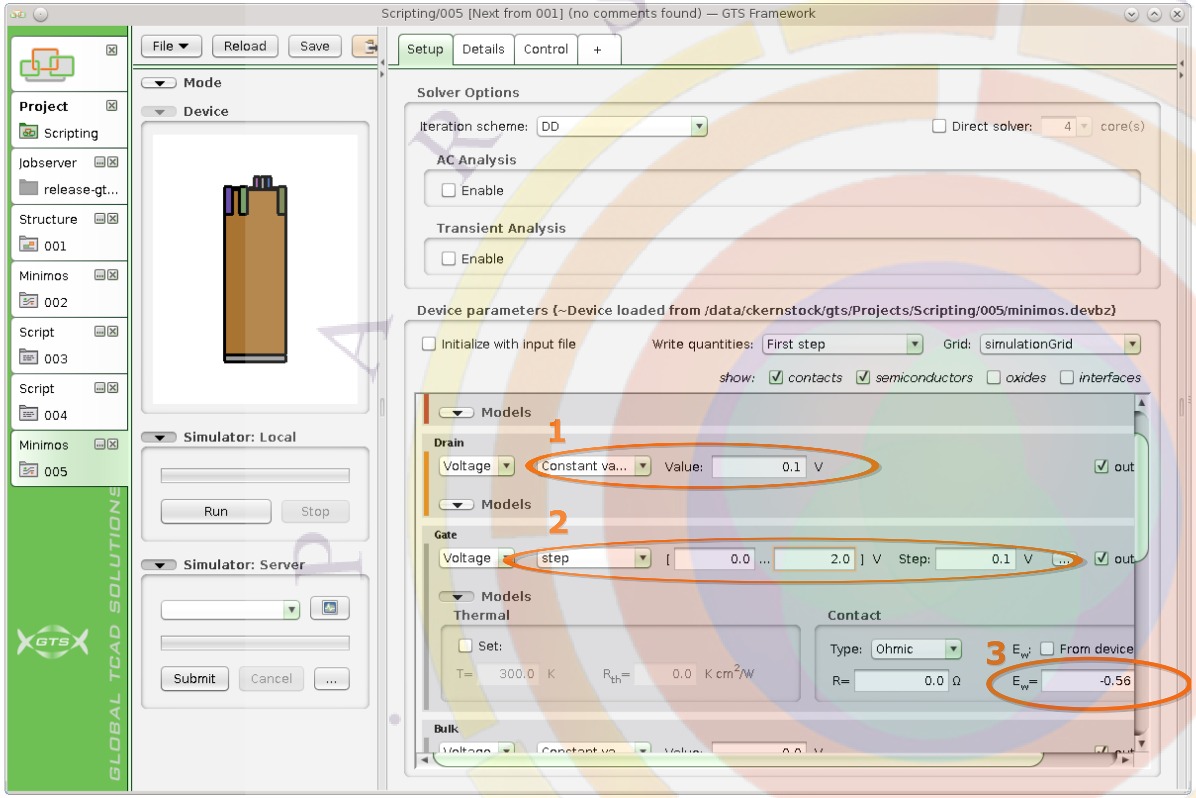
Post-Processing

This section illustrates usage of the post-processing tool to show the influence of a device parameter (mobility) to specific figures of merit of a MOSFET, such as V_{th}, I_{ON}, I_{OFF}, and the sub-threshold slope.

2.1. Vth, Ion, Ioff, Sub-Threshold Slope

2.1.1. Minimos

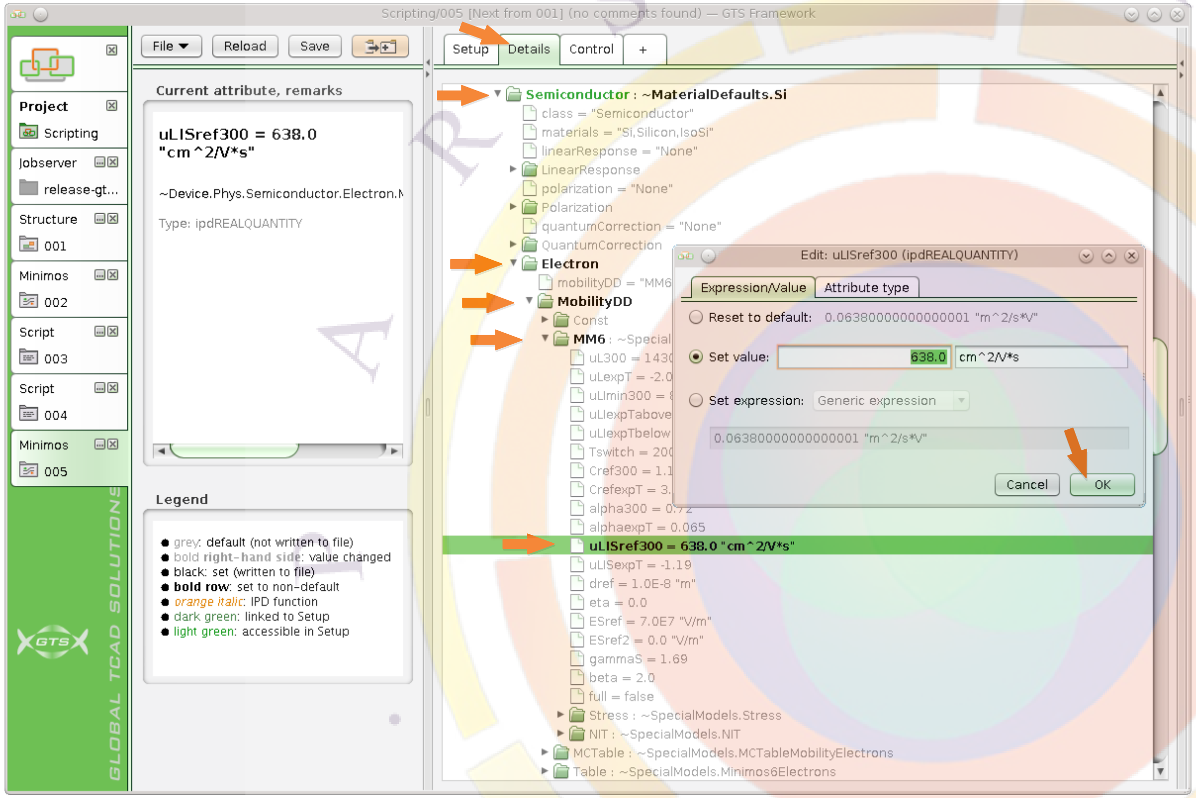
1. Switch back to the *Project Home*
2. Locate the *Structure 001* ToolFolder, and click its *Next tool*
3. Choose *Minimos*
4. Select *Create new Toolfolder*



2.1.2. Transfer Characteristics

Setup a transfer characteristics simulation with:

1. Drain Voltage = 0.1 V.
2. Gate Voltage = step(0 V, 2 V, 0.1 V)
3. Work function difference = -0.56 V
4. Run or Submit

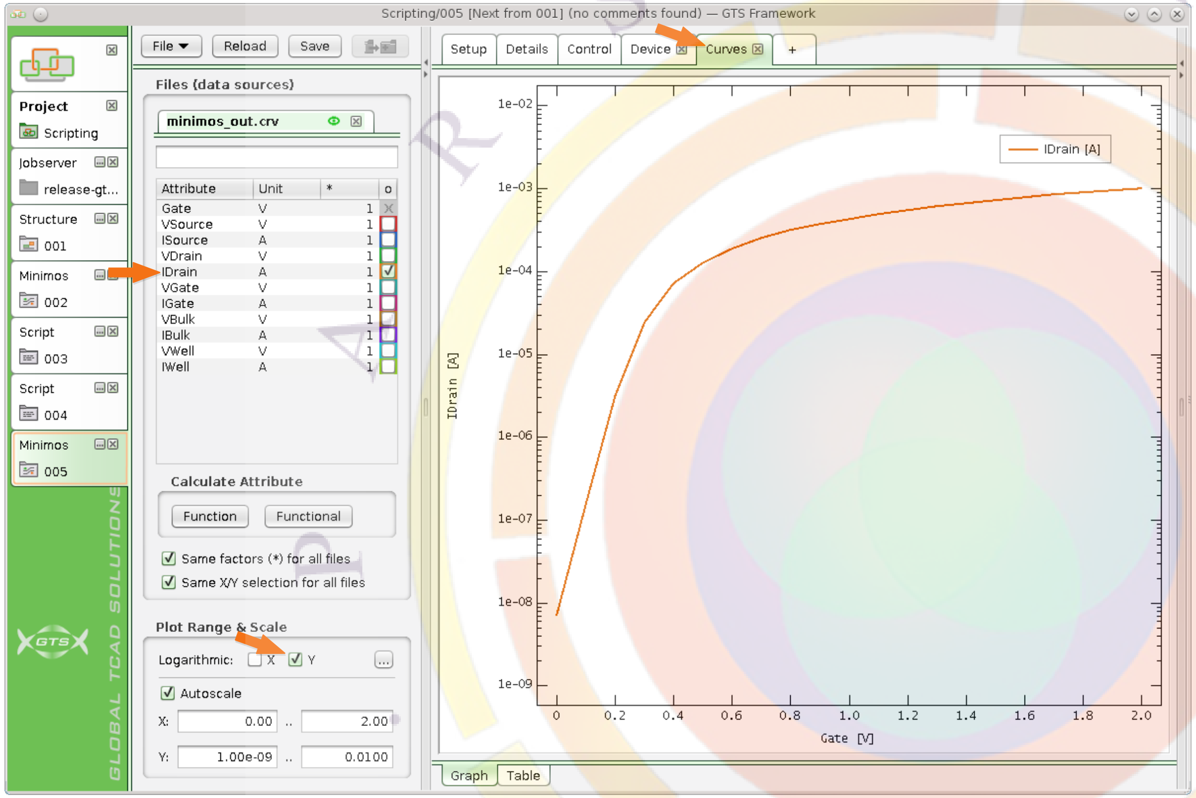


2.1.3. Mobility

To be able to easily choose a parameter later in the Script, we need to touch it in the current tool, so that it is explicitly present in the setup (not just an inherited default value):

- Go to the *Details* page
- Touch the surface mobility parameter: Navigate to ~Device.Phys.Semiconductor.Electron. MobilityDD.MM6.uLISref300 and double-click it, then click *OK* without any modification.

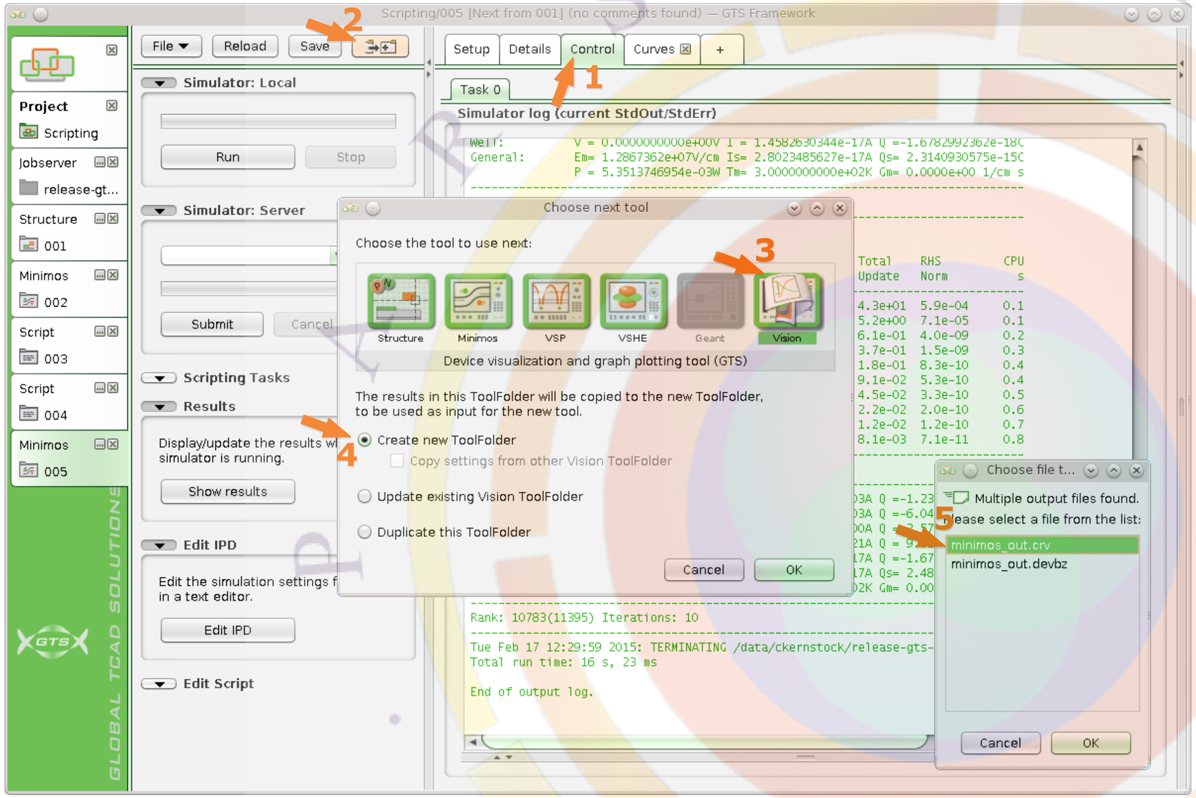
Note that the touched item is now black, which indicates its explicit presence in the setup.



2.1.4. Result: IV-curve

- Switch to the *Setup* page and Submit the experiment.
- Modify the view accordingly, as shown in the figure.

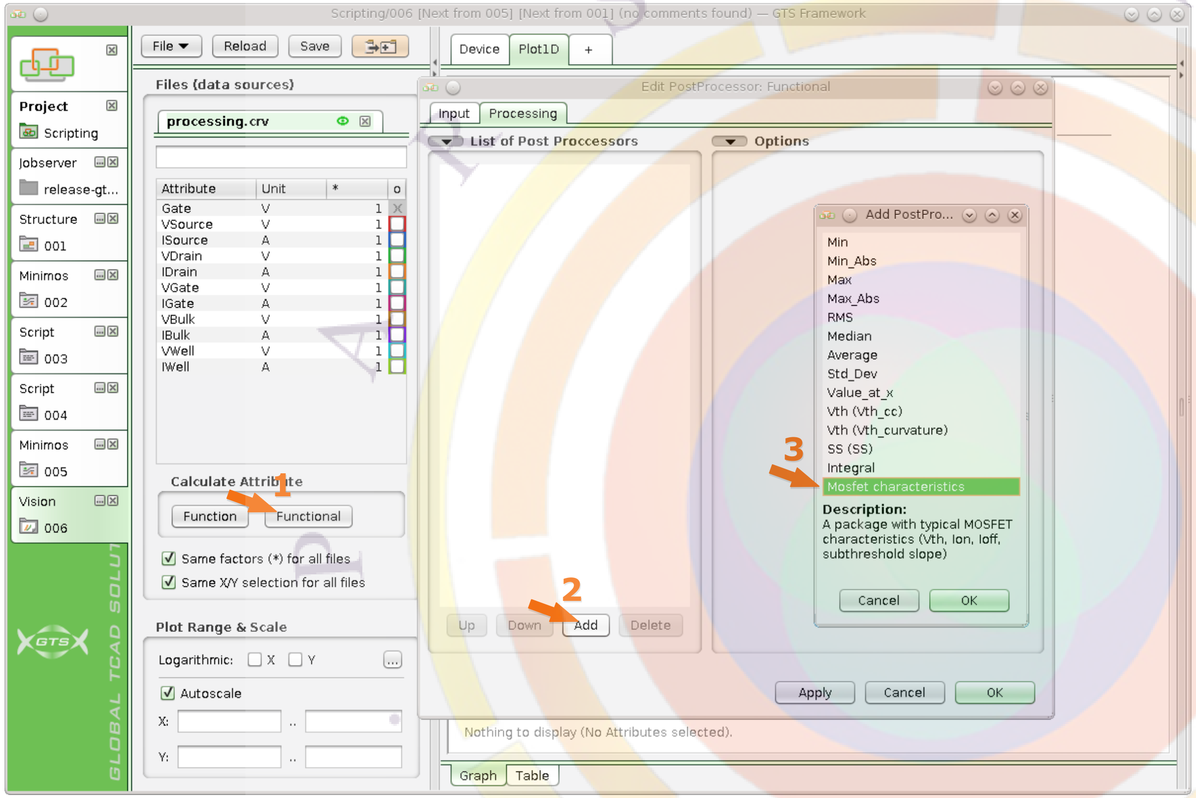
We now see the transfer characteristics.



2.1.5. Next Tool: Vision

Post-processing is done in the Vision tool.

1. Switch to the *Control* page (optional)
2. Select *Next Tool*
3. Select *Vision*
4. Create new ToolFolder
5. When asked, select the .crv file

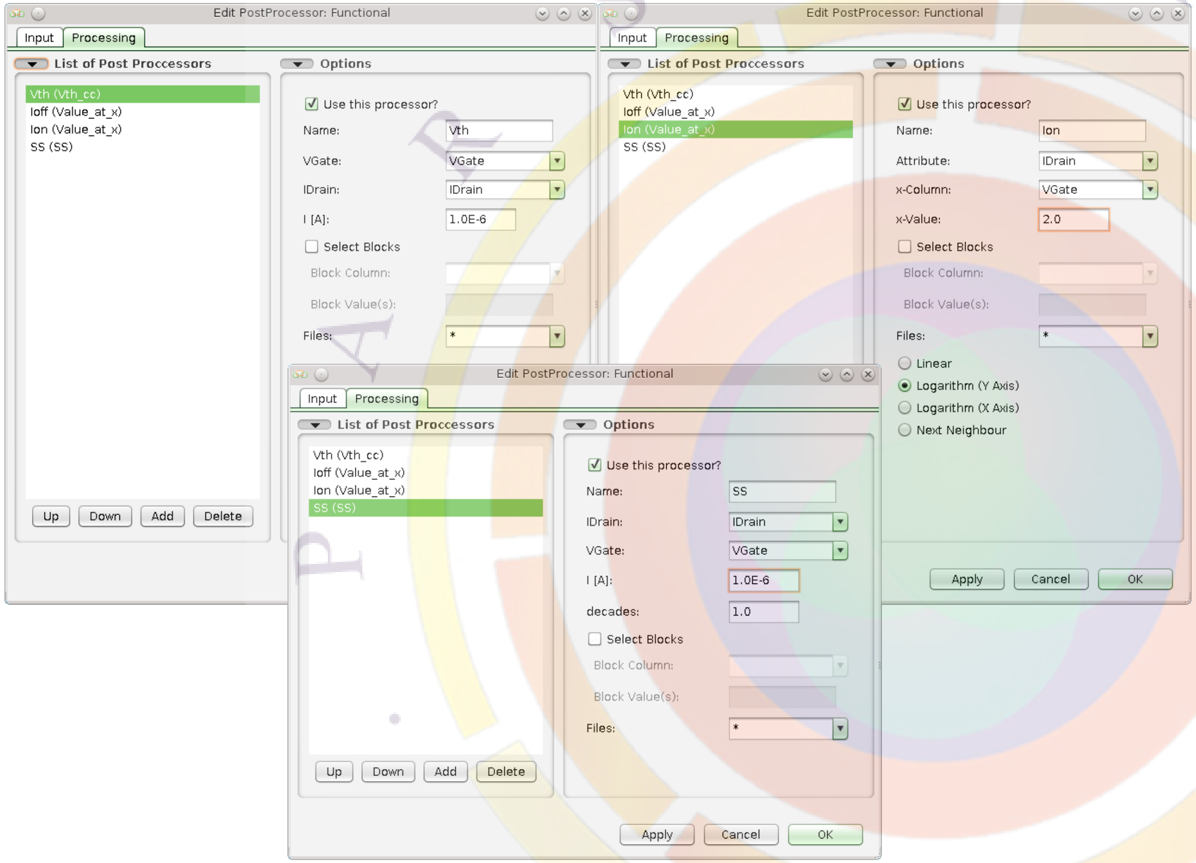


2.1.6. MOSFET Characteristics

Add the MOSFET characteristics:

1. In the *Post-Processor* group, click *Functional*
2. In the post-processors list, click *Add*
3. Choose *Mosfet characteristics*

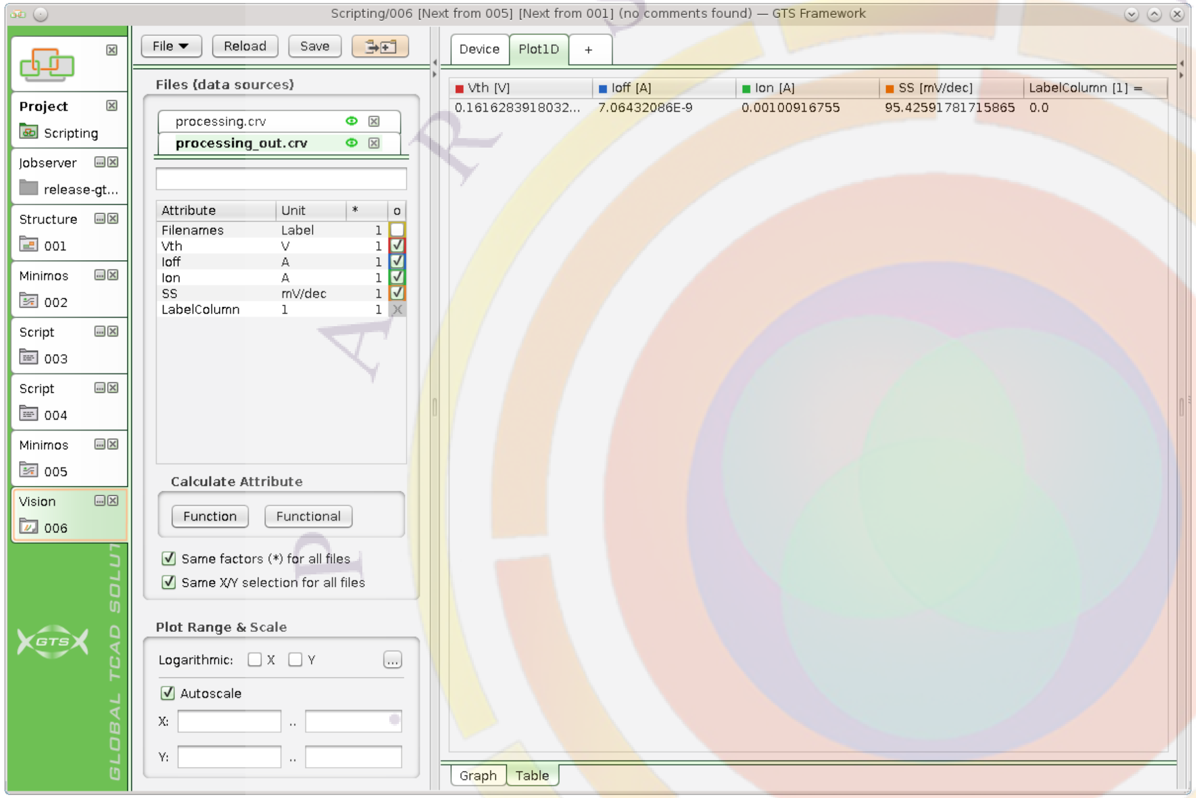
A respective set of post-processors is added to the list.



2.1.7. MOSFET Characteristics Settings

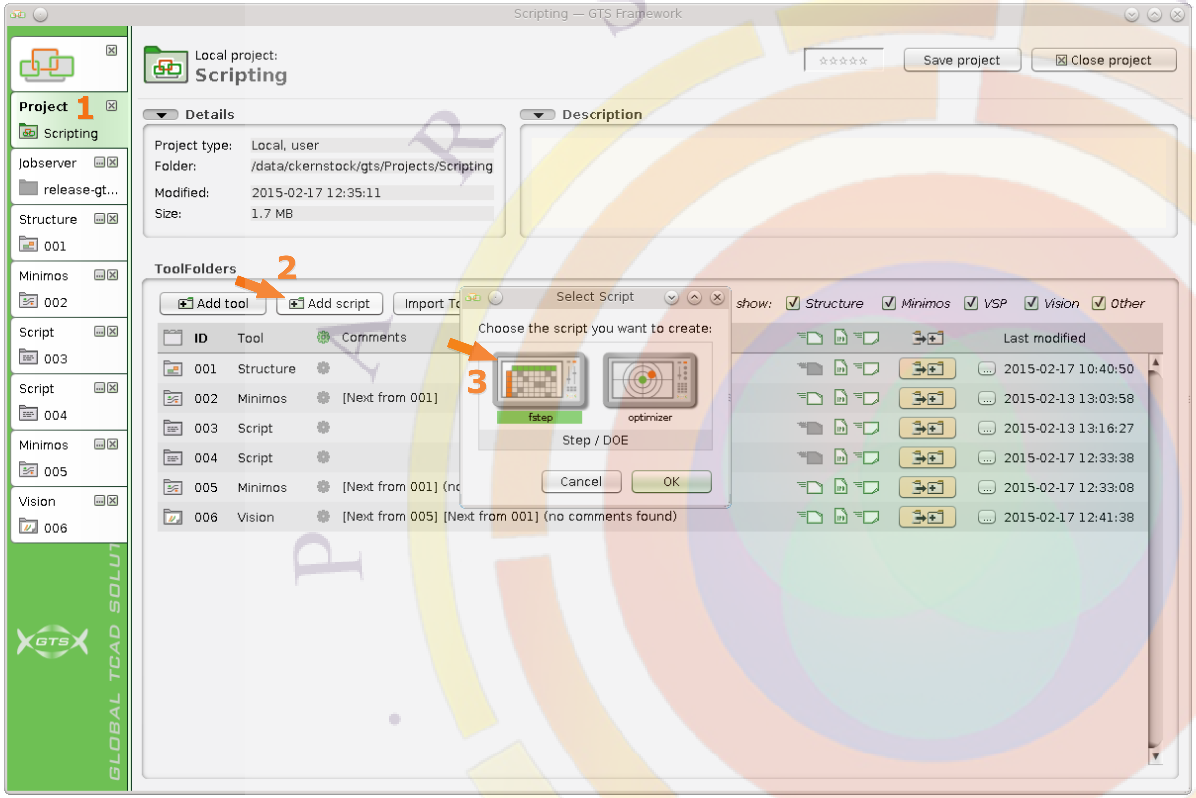
To fit to the current device, edit the post processors according to the figure. In particular, change following values:

- Vth: I = 1E-6 A
- Ion: x-Value = 2.0
- SS: I = 1E-6



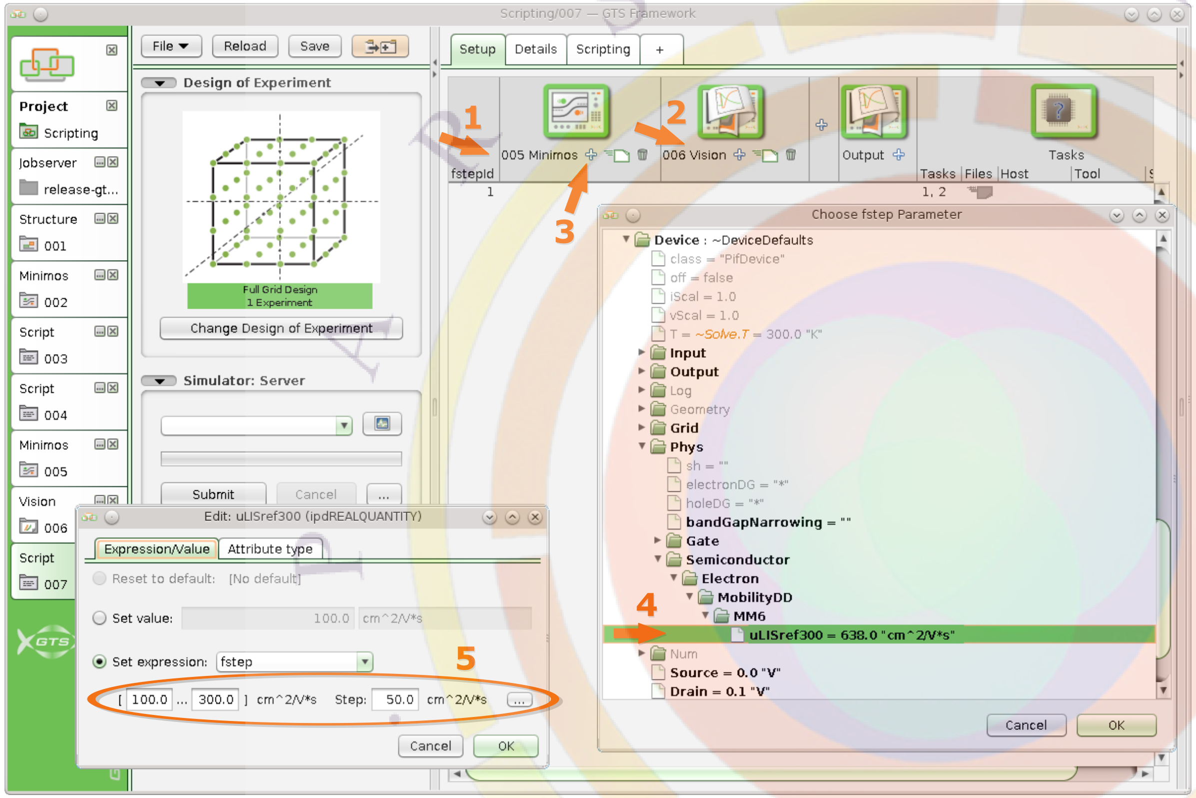
2.1.8. Result

The resulting values of V_{th}, I_{ON}, I_{OFF}, and the sub-threshold slope are shown here.



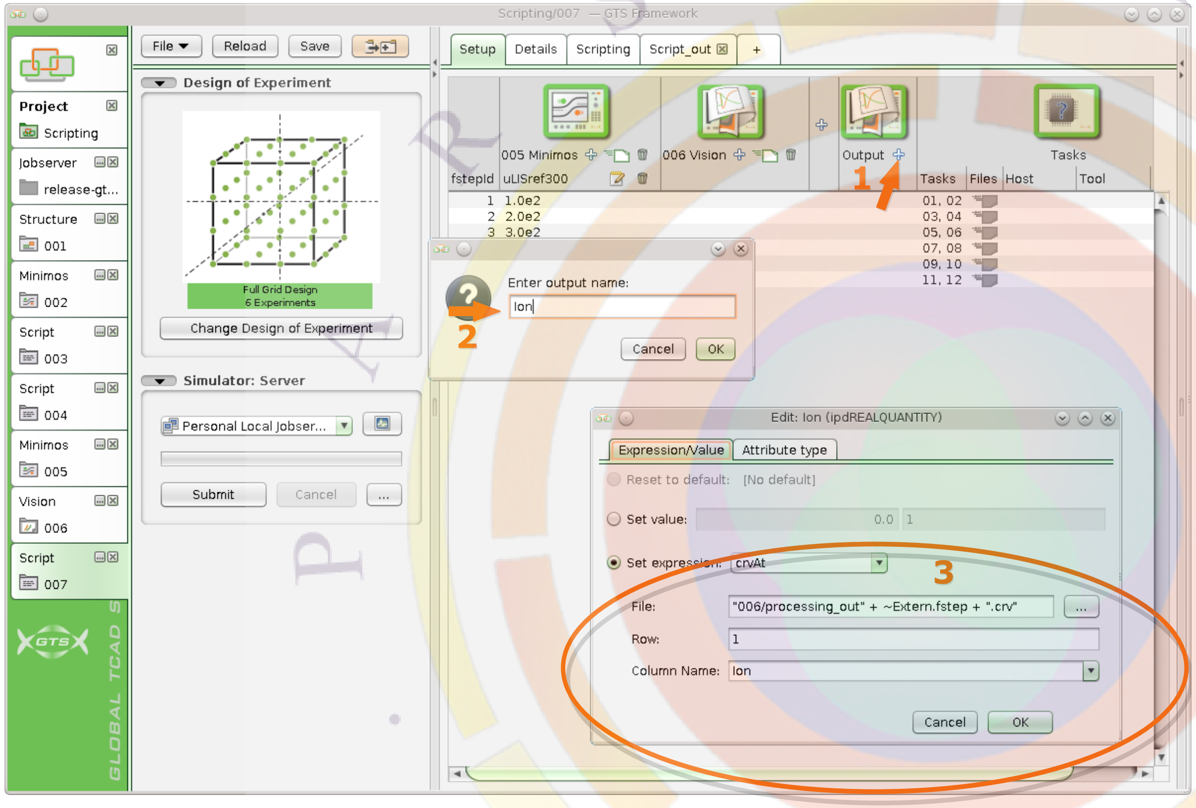
2.1.9. Adding the DOE Tool

1. Switch back to the *Project Home*
2. Click *Add script*
3. Choose *fstep*



2.1.10. Setting up DOE

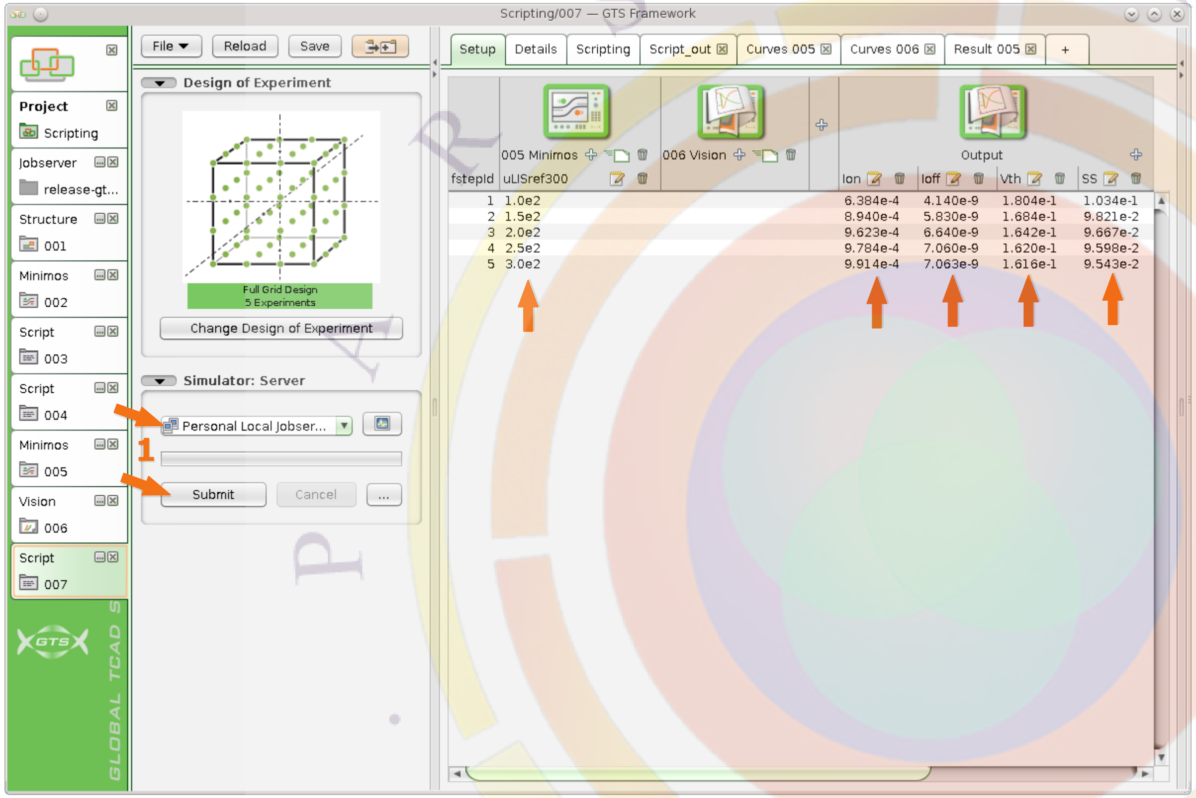
1. Add ToolFolder: Minimos 005
2. Add ToolFolder: Vision 006
3. Click *Add parameter*:
4. ~Device.Phys.Semiconductor.Electron.MobilityDD.MM6.uLISref300
5. Select fstep(100 cm^2/V*s, 300 cm^2/V*s, 50 cm^{-2}/V*s)



2.1.11. Output Values

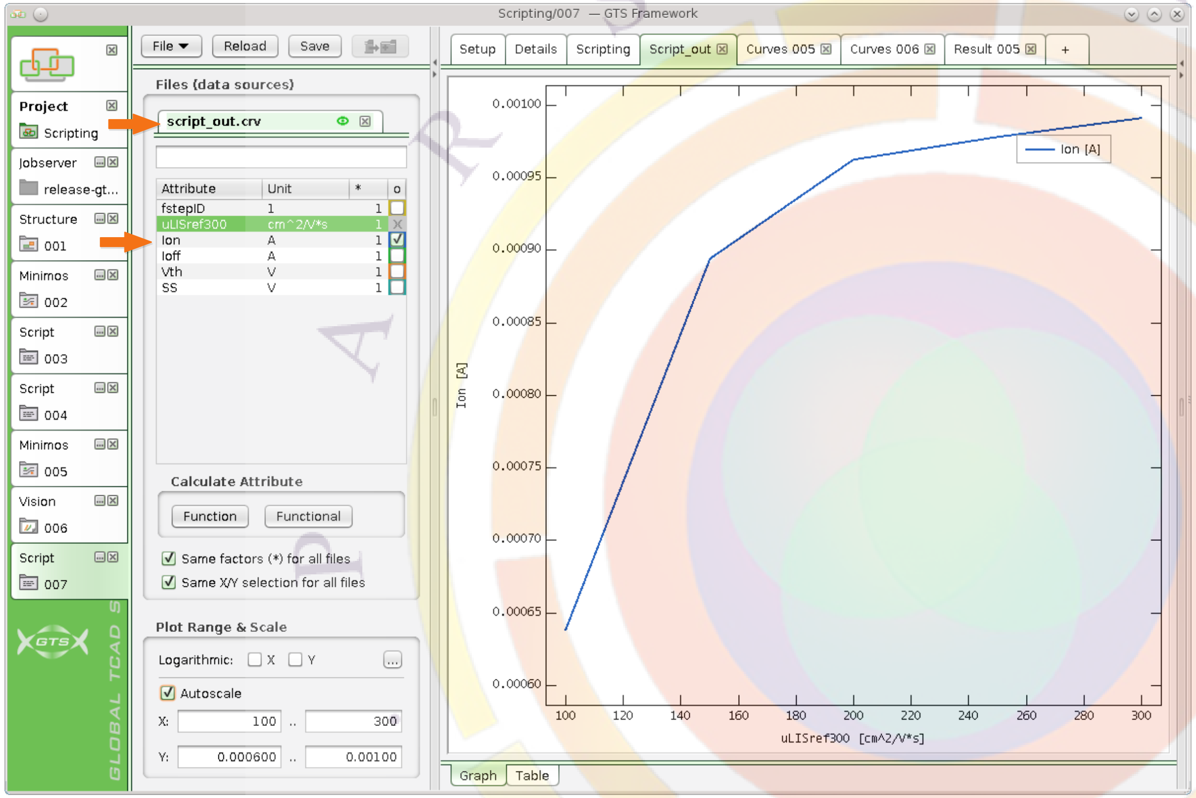
1. Add *Output*
2. Enter name: *Ion*
3. Select the appropriate parameters:
   file = 006/processing_out.crv
   Column Name = Ion

- If desired, perform step 3 for Ioff, Vth and the sub-threshold slopes accordingly.



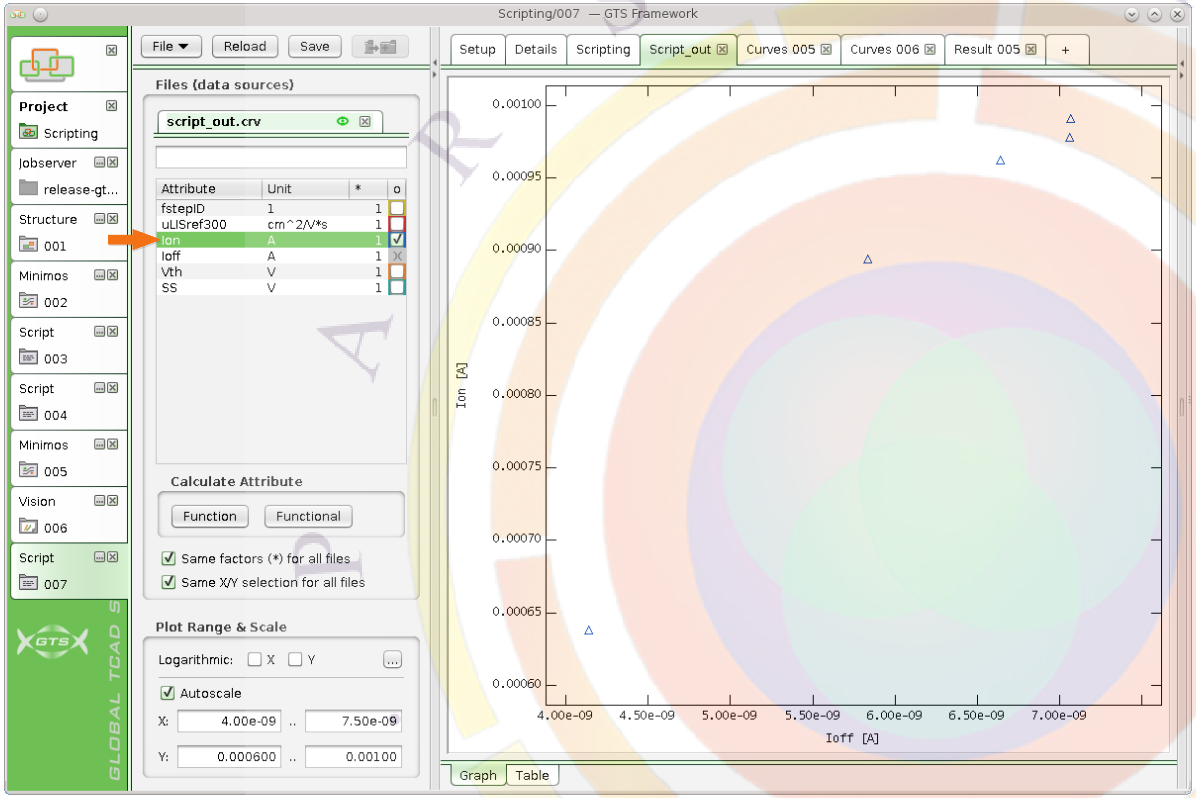
2.1.12. Submit

Submit the experiment to the Jobserver and see the results in the various columns.



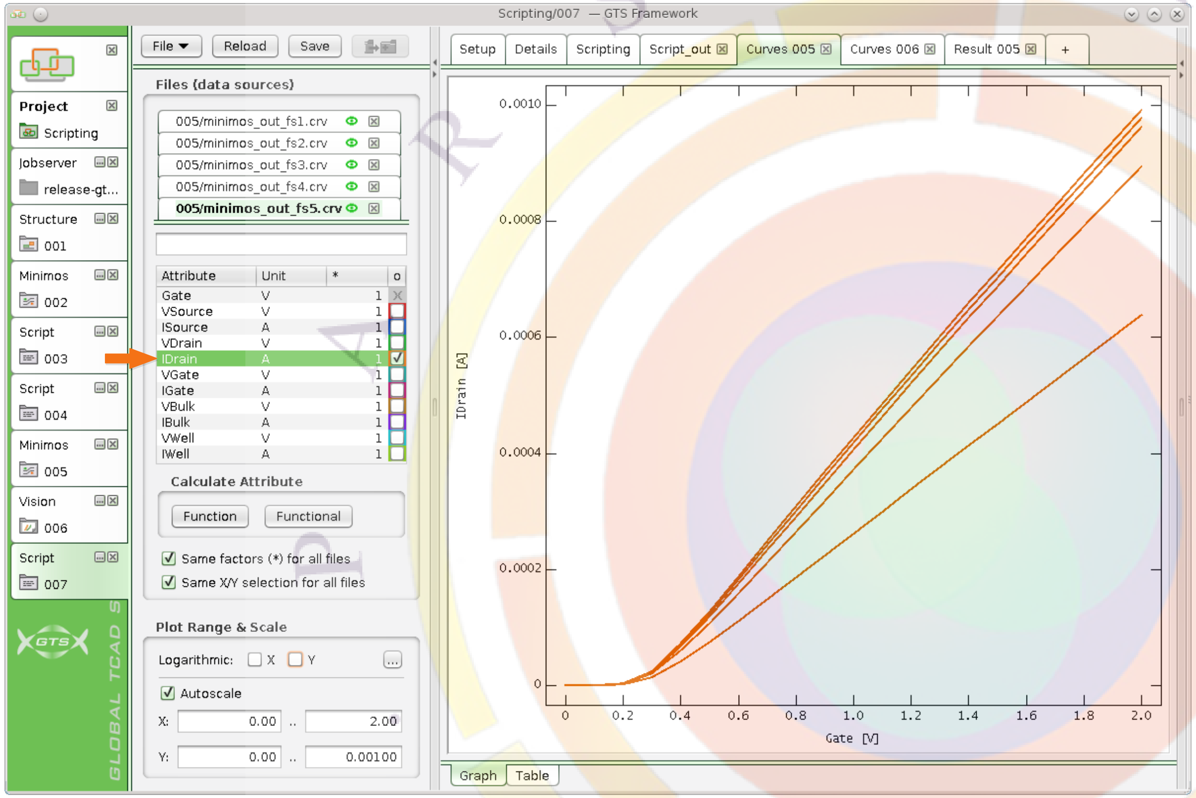
2.1.13. Result: Ion

The resulting on-current over the surface mobility is ready to be shown (set up the plot accordingly, refer to the figure).



2.1.14. Result: Ion/Ioff

To get the resulting on-current over off-current plot, right click *Ioff*, click *Set as X axis* and tick *Ion*. Select a symbol style to your preference, for instance like shown in the figure.



2.1.15. Result: Transfer Characteristics

The detailed results are not retrieved automatically.

- You can choose to retrieve all results (or only .crv files) on the *Scripting* page.
- You can then show the transfer characteristics (Idrain over Gate) for all steps

Conclusion

By some basic examples, this tutorial illustrated the use of the DOE / fstep functionality as well as the optimizer and the post-processing capabilities in ***GTS Framework***. We hope it was useful, and we invite you to try more, such as exploring DOE with more parameters.

Further Reading

We welcome you to have a look at further *GTS tutorials* and *examples*, which you can open in ***GTS Framework***. Next to the basic ones included with the release, you can download more sophisticated tutorials and examples from MyGTS at `https://globaltcad.com/mygts`. Extracting the archives to your projects folder makes the tutorials visible in the projects list (highlighted yellow). Previews are provided at `http://www.globaltcad.com/en/solutions.html`.

For additional information, please refer to `http://globaltcad.com/` or feel free to contact us at `info@globaltcad.com`.

Appendix A

ToolFolder List

The project `Scripting` contains the following ToolFolders (TF):

TF	Tool	Description
T01	Structure	Template: nmos_65nm_2d_simple.ipd
T02	Minimos	Tutorial: [Next from T01] CV curve
T03	Script	Tutorial: Parameter variation: dopingVth –> CV curve
T04	Script	Tutorial: Parameter fitting: dopingVth + oxide thickness –> CV curve (measured)
T05	Minimos	[Next from T01] transfer characterics
T06	Vision	[Next from T05] MOSFET charactercis
T07	Script	Parameter Variation: surface mobility –> MOSFET characterics: Ion, Ioff, Vth, sub threshold slope