

Agilent 85190A IC-CAP 2008

Target Modeling



Agilent Technologies

Notices

© Agilent Technologies, Inc. 2004-2008

No part of this manual may be reproduced in any form or by any means (including electronic storage and retrieval or translation into a foreign language) without prior agreement and written consent from Agilent Technologies, Inc. as governed by United States and international copyright laws.

Edition

March 2008

Printed in USA

Agilent Technologies, Inc.
5301 Stevens Creek Blvd.
Santa Clara, CA 95052 USA

Acknowledgments

UNIX ® is a registered trademark of the Open Group.

Errata

The IC-CAP product may contain references to "HP" or "HPEESOF" such as in file names and directory names. The business entity formerly known as "HP EEsof" is now part of Agilent Technologies and is known as "Agilent EEsof." To avoid broken functionality and to maintain backward compatibility for our customers, we did not change all the names and labels that contain "HP" or "HPEESOF" references.

Warranty

The material contained in this document is provided "as is," and is subject to being changed, without notice, in future editions. Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied, with regard to this manual and any information contained herein, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or of any information contained herein. Should Agilent and the user have a separate written agreement with warranty terms covering the material in this document that conflict with these terms, the warranty terms in the separate agreement shall control.

Technology Licenses

The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license.

Restricted Rights Legend

U.S. Government Restricted Rights. Software and technical data rights granted to the federal government include only those rights customarily provided to end user customers. Agilent provides this customary commercial license in Software and technical data pursuant to FAR 12.211 (Technical Data) and 12.212 (Computer Software) and, for the Department of Defense, DFARS 252.227-7015 (Technical Data - Commercial Items) and DFARS 227.7202-3 (Rights in Commercial Computer Software or Computer Software Documentation).

Safety Notices

CAUTION

A CAUTION notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in damage to the product or loss of important data. Do not proceed beyond a CAUTION notice until the indicated conditions are fully understood and met.

WARNING

A WARNING notice denotes a hazard. It calls attention to an operating procedure, practice, or the like that, if not correctly performed or adhered to, could result in personal injury or death. Do not proceed beyond a WARNING notice until the indicated conditions are fully understood and met.

Contents

1 Target Modeling

Graphic User Interface for Target Modeling 6

Using the Target Modeling Tool 9

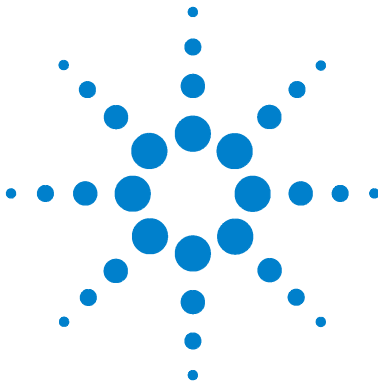
PCM Data 11

Diagrams 27

Report 31

Options 32

Adding Your Own Simulation Templates 34



Target Modeling

Graphic User Interface for Target Modeling 6

Using the Target Modeling Tool 9

Adding Your Own Simulation Templates 34

Target Modeling is a flexible IC-CAP toolkit that enables you to adjust a simulation model to definable targets. For example, you can use it to adjust a model to a new process or to view the overall trend of a production process.

In any semiconductor manufacturing environment, process data is monitored using process control monitors (PCM's). For example, those control monitors usually don't include device data from complete sweeps of gate voltages. Typically ID is measured at specific values of VG to get discrete pairs of VG/ID values. The goal is to use those *targets* to adjust a simulation model with extracted parameters from only a few wafers to the manufacturing process trend with a much larger number of measurements available through PCMs.

This toolkit enables you to:

- Adjust a simulation model that was extracted on one die of a wafer to data that represents the overall trend of this process.
- Generate a simulation model for a new technology based on an existing model card plus a reduced set of characteristic data of this new technology.
- Perform comparisons between different simulated models and silicon data.
- Generate documentation containing measured data (PCM, Device Data) and the appropriate simulations.
- Easily add your own methods, model definitions, circuit definitions, and so on.

Graphic User Interface for Target Modeling

The Target Modeling toolkit's GUI uses two windows:

- The control GUI to manage data, simulations, and plots.
- Plots/multiplots as the main objects to perform the target modeling.

The control module is an IC-CAP *.mdl* file, which manages the OTM objects and handles templates necessary for PCM simulation, report generation, and so on.

Control code and data are separated—control code is found inside the model *Target_Modeling* and data is stored inside data files using the naming convention *OTM_Data_(ProjectName)*.

The data files contain simulation setups, plots and multiplots, methods (plot optimizers, extractions), as well as user specific extensions. All relevant information regarding one project is stored in a single model file.

The control module contains 4 tabs: *Data*, *Diagrams*, *Reports*, and *Options*. The following figure shows the *Target Modeling* GUI.

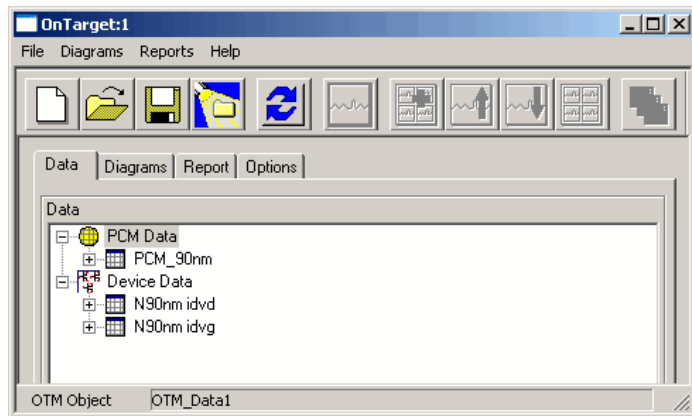


Figure 1 Target Control GUI

Using the *Data* tab, you can manage PCM as well as device data. PCM data format is read from files formatted as a table containing the following details:

The first n rows of the table are the KEYWORD rows. Keywords can be:

- TYPE with
 - ID=Identification
 - IP=Instance parameters
 - PCM=PCM parameters
- NAME with
 - Fixed Values for:
 - DeviceID
 - Name
 - Denominators for:
 - Instance Parameters (IP)
 - PCM values (PCM)
- SUBSET: specifies in detail the PCM values. Allowed names are *mean*, *min*, *max*. These identifiers are used to add statistical data.
- TEMP: Temperature of the PCM value

In the file, each column is separated by a comma (,). This enables you to import the file into table calculation programs like MS Excel.

A further rule for the rows *NAME* and *TEMP* is that if a cell does not contain a name, the value of the previous one is used. For example, T=25°C is used as a temperature for all PCM values as long as no other value is specified. Following is an extract from such a PCM data file.

	A	B	C	D	E	F	G	H	I	J
1	TYPE	ID		IP						
2	NAME	DeviceID	Name	Length	Width	AD	AS	PD	PS	NF
3	SUBSET									
4	TEMP									
5		0	N_W 5.000u	9.00E-08	5.00E-06	9.00E-14	9.00E-14	1.18E-06	1.18E-06	1
6		1	N_W 5.000u	1.30E-07	5.00E-06	1.30E-13	1.30E-13	1.26E-06	1.26E-06	1
7		2	N_W 5.000u	1.60E-07	5.00E-06	1.60E-13	1.60E-13	1.32E-06	1.32E-06	1
8		3	N_W 5.000u	2.50E-07	5.00E-06	2.50E-13	2.50E-13	1.50E-06	1.50E-06	1
9		4	N_W 5.000u	8.00E-07	5.00E-06	8.00E-13	8.00E-13	2.60E-06	2.60E-06	1
10		5	N_W 5.000u	1.00E-06	5.00E-06	1.00E-12	1.00E-12	3.00E-06	3.00E-06	1
11		6	N_W 5.000u	5.00E-06	5.00E-06	5.00E-12	5.00E-12	1.10E-05	1.10E-05	1
12		7	N_W 5.000u	1.00E-05	5.00E-06	1.00E-11	1.00E-11	2.10E-05	2.10E-05	1
13		8	N_W 180.0n	1.30E-07	1.80E-07	1.30E-13	1.30E-13	1.26E-06	1.26E-06	1
14		9	N_W 180.0n	2.50E-07	1.80E-07	2.50E-13	2.50E-13	1.50E-06	1.50E-06	1
15		10	N_W 180.0n	8.00E-07	1.80E-07	8.00E-13	8.00E-13	2.60E-06	2.60E-06	1
16		11	N_W 180.0n	5.00E-06	1.80E-07	5.00E-12	5.00E-12	1.10E-05	1.10E-05	1
17		12	N_W 180.0n	1.00E-05	1.80E-07	1.00E-11	1.00E-11	2.10E-05	2.10E-05	1
18		13	N_W 250.0n	1.30E-07	2.50E-07	1.30E-13	1.30E-13	1.26E-06	1.26E-06	1
19		14	N_W 250.0n	2.50E-07	2.50E-07	2.50E-13	2.50E-13	1.50E-06	1.50E-06	1

NOTE

Default content for empty cells depends on cell type:

- An empty cell inside the *NAME* row leads to an error message. For example, if the cell following *AD* is empty, you will get the error message *Instance Parameter Name AD is defined twice*. In this case, the name of the previous cell is used and double definitions are not enabled.
- An empty data value cell is assumed to have the value of zero.

The goal is to create an easy way to import data from different sources that are not in the IC-CAP data format. Using this comma separated value (.csv) format, any data format can be used with only minor changes. Those changes can be made using a table calculation program like MS Excel or similar programs.

The *Diagrams* tab enables you to open plots—single plots can be opened from the left side and multiplots from the right side of this tab.

On the *Report* tab, you can create customized reports of the projects.

The *Options* tab enables you to select the simulator and circuits.

Using the Target Modeling Tool

To better understand the tasks and to easily try the features, an examples directory is provided. This directory is located in `$ICCAP_ROOT\examples\model_files\target_modeling\examples`. It contains the `BSIM4_NMOS_example` and the `BSIM4_NMOS_from_scratch` directory, which contains a few subdirectories and specific example files that will enable you to try all aspects of Target Modeling.

The `iv_data` directory contains a `.dut`, `.mdl`, `.mdm`, and `.set` file. Another directory, `model_parameters`, contains an `.mps` file. The `simulation_template_configuration` directory includes the `SinglePoint.csv` and `Vth_GM.csv` simulation templates. The `pcm_data_files` directory includes 3 example PCM files.

To create a new project, open `$ICCAP_ROOT\examples\model_files\target_modeling\TargetModeling.mdl`. Double choose `Target_Modeling` to open the Target Modeling GUI, then choose `File > New` from the menu. The `Create New Data Model` window opens prompting you to enter a name for the new data model. Standard naming convention applies and every new model begins with the prefix `OTM_`, but you can overwrite this setting.

CAUTION

Do not rename the data model file as it is displayed in the main window after creation: It will be unusable!

Next, select a `Simulation Model` from the list on the left side of this window. Once you've selected a model, the right side of this window lists the available simulators (see the following figure).

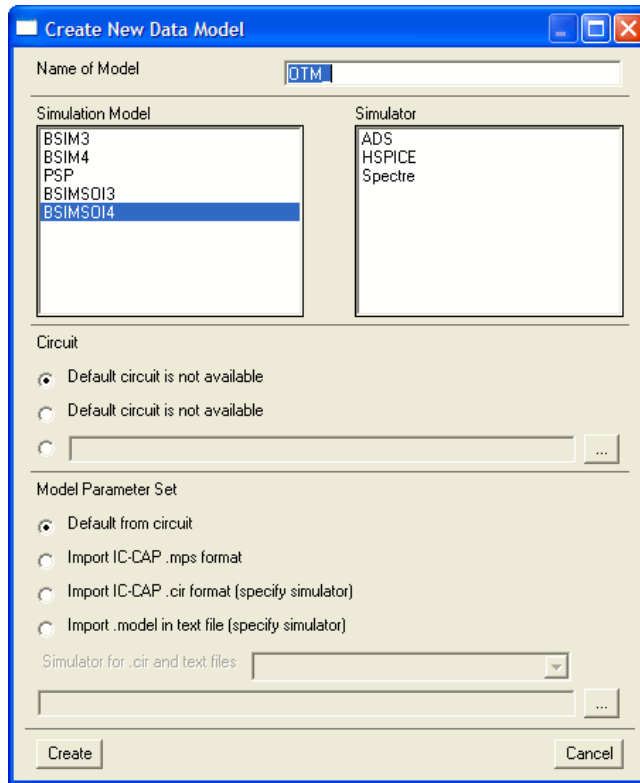


Figure 2 Startup Wizard to create a new data model

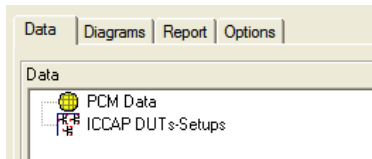
Select a simulator then select if you would like to use the default NMOS or PMOS circuit from the respective Modeling Package. This selection is only available after you select a simulator. If you would like to select another circuit, you can browse to the location where the circuit is located.

Next, select the model parameter set. You can use the default model parameter set included in the circuit or import one in IC-CAP .mps or .cir format. For example, import the example model parameter set from $\$ICCAP_ROOT\examples\model_files\target_modeling\examples\BSIM4_NMOS_from_scratch\model_parameters$.

You can also import a model parameter set in the form of a text file. This could be a library file containing only one *.model* statement. In this case, select the simulator of origin for the model file. For example, this selection gives you the flexibility to generate a model for a new technology based on an existing model card. You can check if the used model obtains adequate results for, let's say, reduced gate dimensions.

PCM Data

After creating a new project, the Target Modeling window shows two entries on the Data tab.



For the first step, right click *PCM Data* and choose *New PCM Table*. The *New PCM DUT* dialog box appears prompting you for the name of the new PCM DUT and the location of the data file, which needs to be in comma separated value (*.csv*) format as previously explained.

See `$ICCAP_ROOT\examples\model_files\target_modeling\examples\BSIM4_NMOS_from_scratch\pcm_data_files` for example PCM data files.

After choosing *OK*, a new PCM table is inserted below *PCM Data* on the Data tab. Now, right click this new entry to:

- *Delete PCM Table*
- *Derive a Subset or Device Data*
- *Re-import* data for this PCM table by selecting the desired data files. This routine checks the data file format and reports any errors.
- open a *Device Info* table showing the names and some parameters of the devices read in from the *.csv* table.

NOTE

Derive device data is available only after adding and configuring a Simulation Template, defining at least one working condition.

Now, the Data tab contains the information for the PCM below the named PCM in the form of a tree structure. The branches are:

- *Available Outputs*

The available outputs are read in from the PCM data file – they represent your PCM data.

- *Simulation Templates*

A configured Simulation Template is required to simulate PCM data and to derive device data. Therefore, it is recommended to add and configure a Simulation Template first.

- *All*

The *All* branch contains device data as specified inside the *.csv* file just read in. The Device Info table shows the devices configured for this PCM set.

Simulation Templates

From imported PCM data, two kinds of diagrams can be generated using predefined simulation templates:

- PCM values versus Instance Parameters (L, W,..) or temperatures

Those plots are generated by configuring Simulation Templates.

- I-V like diagrams with only a few measured points but full simulation

Those plots are based on the configuration of the Simulation Templates plus the subset configuration done while deriving device data.

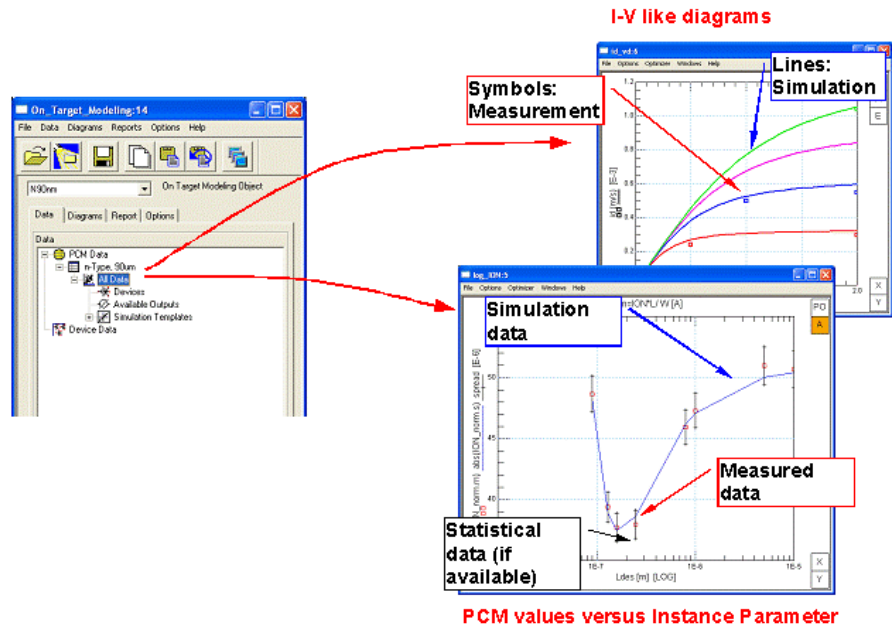


Figure 3 Possible kinds of diagrams

Two types of simulation templates are implemented for the BSIM3, BSIM4, and PSP models:

- Single Point current
- $V_{th}(I_{ref}, gm)$, $max(gm)$, $gm(V_{th}+offset)$

Two similar templates for the BSIMSIO3 and BSIMSIO4 models are implemented (for floating body SOI):

- Single Point current
- $V_{th}(I_{ref}, gm)$, $max(gm)$, $gm(V_{th}+offset)$

Additional templates can be added using the *Target_Modeling* file.

A support document named “Simulation Templates for Target Modeling” describes how to implement a user defined model.

The following figures show the two different types in more detail:

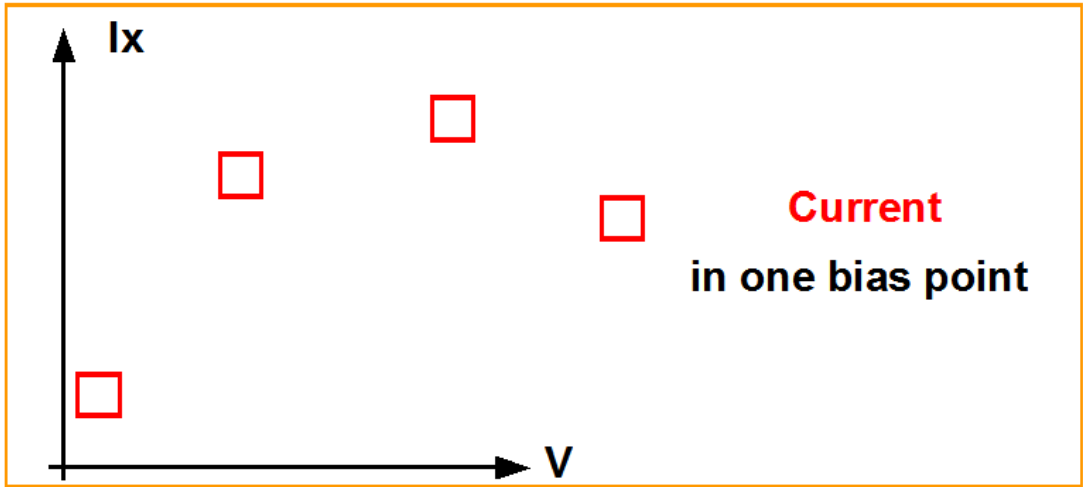


Figure 4 MOS single point simulations for I/V-diagrams

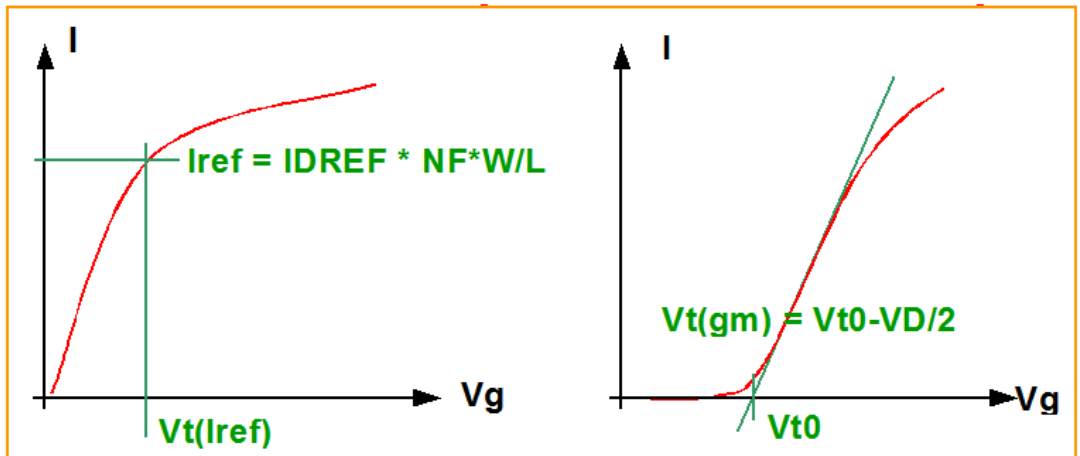


Figure 5 MOS threshold voltage related simulation V_{th}

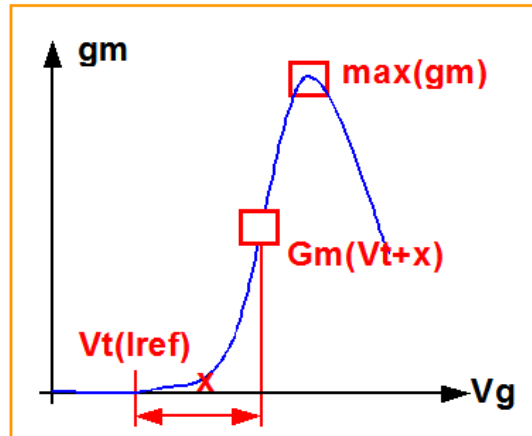


Figure 6 MOS threshold voltage related simulation conductance

Now that PCM data has been read in and subsets have been created (if desired), the next step is to configure the simulation templates.

NOTE

Although you can create subsets, you can not create device data until at least one simulation template is configured.

Right click the *Simulation Templates* entry below the PCM Data set you just created, then choose *Add Template*. The Simulation Templates dialog box appears. Select a template defined for the model and simulator, then choose *OK*. The selected simulation template is inserted into the PCM data tree. Right click this template then choose *Configure Template*. This opens the *Simulation Assignment* window to let you assign specific parameters as well as inputs and outputs for this simulation configuration. See the following figure for details.

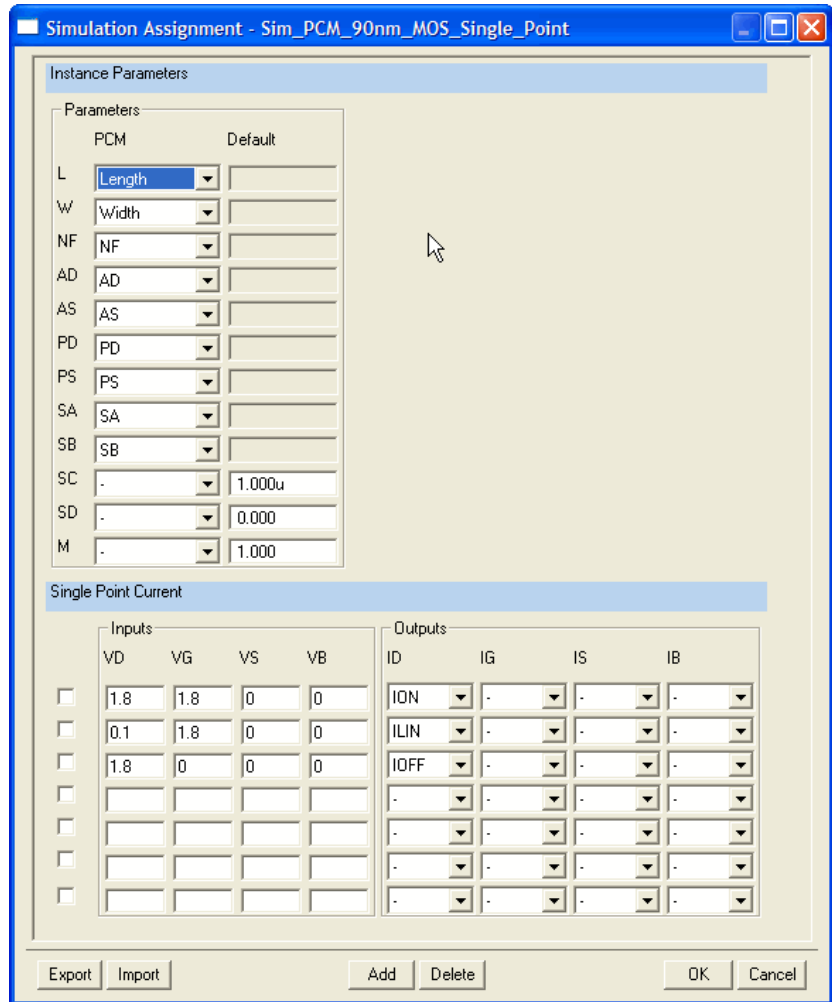


Figure 7 Simulation Assignment

A number of parameters are available for the model chosen. You can change the allocation of the parameter to any other parameter of this set or you can enter a specific value by selecting no allocation (-) from the drop down list. In this case,

the column named *Default* is highlighted and you can enter a value for the selected parameter. If no parameter value was specified, the default value defined in the circuit is used.

The next step is to add input values for the simulation sources. In templates for single-point simulations, the values entered are the simulation values for the exact points (values of VD, VG, VS, and VB) given.

Inputs		Outputs						
VD	VG	VS	VB	ID	IG	IS	IB	
<input type="checkbox"/>	1.8	1.8	0	0	ION	-	-	-
<input type="checkbox"/>	0.1	1.8	0	0	ILIN	-	-	-
<input type="checkbox"/>	1.8	0	0	0	IOFF	-	-	-

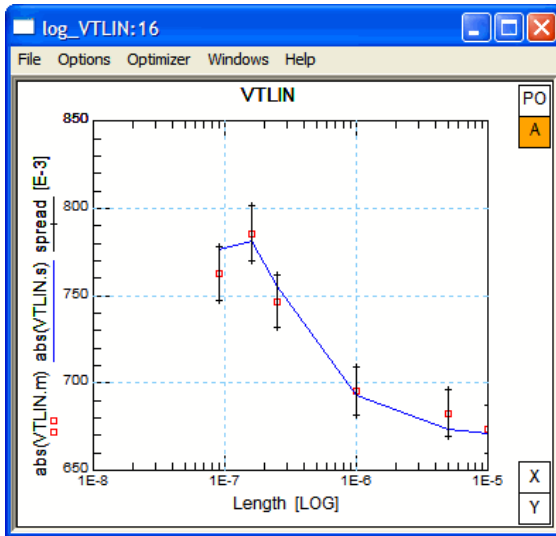
You can add more points (lines) or you can delete points by choosing the appropriate button at the bottom of the *Simulation Assignment* window. Mark the lines you would like to delete by selecting the check box in front of the line(s).

In Vth simulations for example, enter the start, stop, and step values for VG, then enter the rest of the voltages for swept simulations.

Inputs		Outputs				Parameter					
VGstart	VGstop	VGstep	VD	VB	VS	Vt(lref)	Vt(gm)	Gm(Vt+x)	max(gm)	IDREF	x
<input type="checkbox"/>	0	1.8	0.05	0.1	0	0	VTLIN	-	GMMAX	100n	
<input type="checkbox"/>	0	1.8	0.05	1.8	0	0	VTSAT	-	GMDEL	100n	0.5

As shown in the figure above, the first order sweep VG can only be entered into the first row. The following rows are grayed out for VG sweep – you can only enter VD, VB, and VS.

The following plot shows the simulation of part of our example PCM data using the settings from the figure above. The plot shows the value of VTLIN vs. device length (on a logarithmic x-axis) for a subset of devices using a width of 5 μm and a temperature of 25°C.



Now choose the outputs from the drop-down list. Enter parameters to the right of the outputs. For example, if you are interested in the value for g_m at the point ($V_{th} + x$), enter the x-value into the applicable column named x. For $V_{th}(I_{Dref})$ simulations, you have to assign a value for I_{Dref} into the specified column labeled IDREF. If you would like to extract $V_{th}(I_{Dref})$ and g_m in one step, the I_{Dref} value is valid for both simulations. For different values of I_{Dref} , use an additional line and enter different values as desired into the I_{Dref} column.

You can also Export or Import settings from this window to use in other simulations.

From the directory `$ICCAP_ROOT\examples\model_files\target_modeling\examples\BSIM4_NMOS_from_scratch\simulation_template_configuration`, you can import one of the simulation templates provided: `SinglePoint.csv` or `Vth_GM.csv`

Derive Subset

To derive a subset from all devices, right click the *All* branch (or the PCM tree top) then choose *Derive Subset*. In the Prompt Dialog, enter the name of the new subset. You can derive subsets based on preferred criteria selected in the *Select Criteria* window. See the following figure.

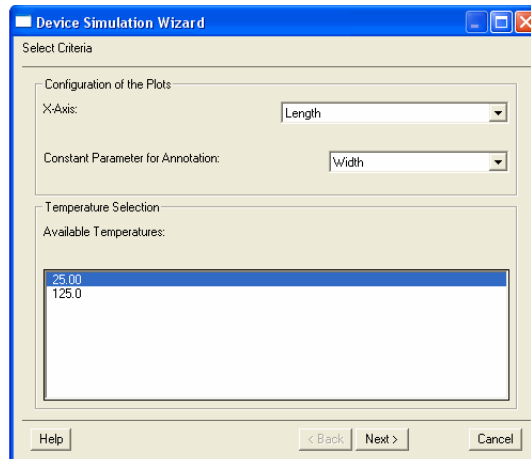


Figure 8 Selections for a subset

You can configure plots for the new subset based on different selectable X-Axis parameters as well as annotation parameters and temperatures, if available.

After selecting the subset criteria and choosing *OK*, a window opens showing the devices that are included in the subset. You can select one or more devices from this window. Choose *OK* and the new subset is inserted into the tree.

Derive Device Data

To derive device data, right click a PCM table or a subset then choose *Derive Device Data*. The *Device Simulation Wizard* window opens. Enter a name for the simulation and choose an available simulation template from the list, then choose *Next*.

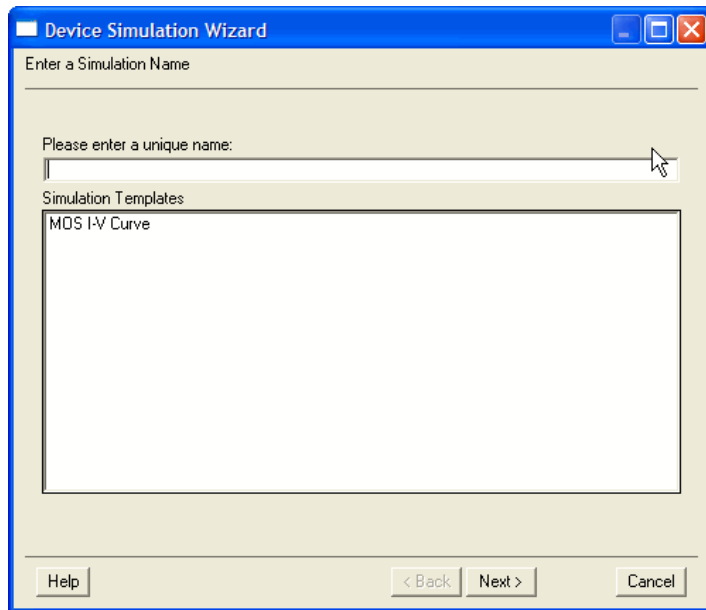


Figure 9 Device Simulation Wizard

The Device Simulation Wizard window now displays items you can include in this device simulation task. Choose one item from the list then click the right arrow to assign this item to the IV-Curve section. To remove an item from this simulation, simply select the item then click the left arrow.

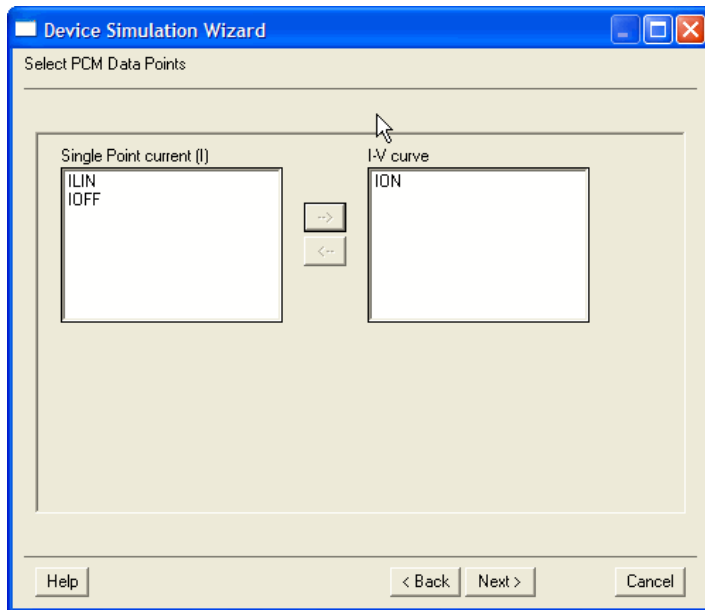


Figure 10 Assignment of predefined simulations

See [Figure 4](#) and [Figure 5](#) for a graphical explanation of Single Point or V_{th} related items.

When you're done assigning data points, choose *Next* to enter simulation conditions. See [Figure 14](#).

Enter the sweep order as well as the conditions for the first order sweep. The values for the second order sweep come from the PCM data values – they can't be changed in this window.

1 Target Modeling

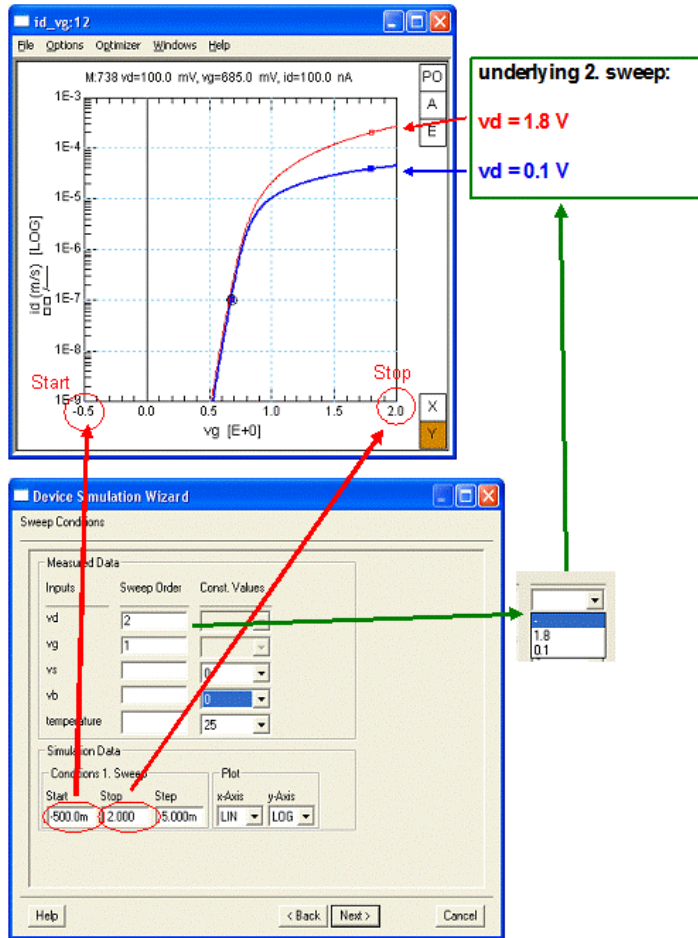


Figure 11 First and second sweep order entries and their relationship to the plot

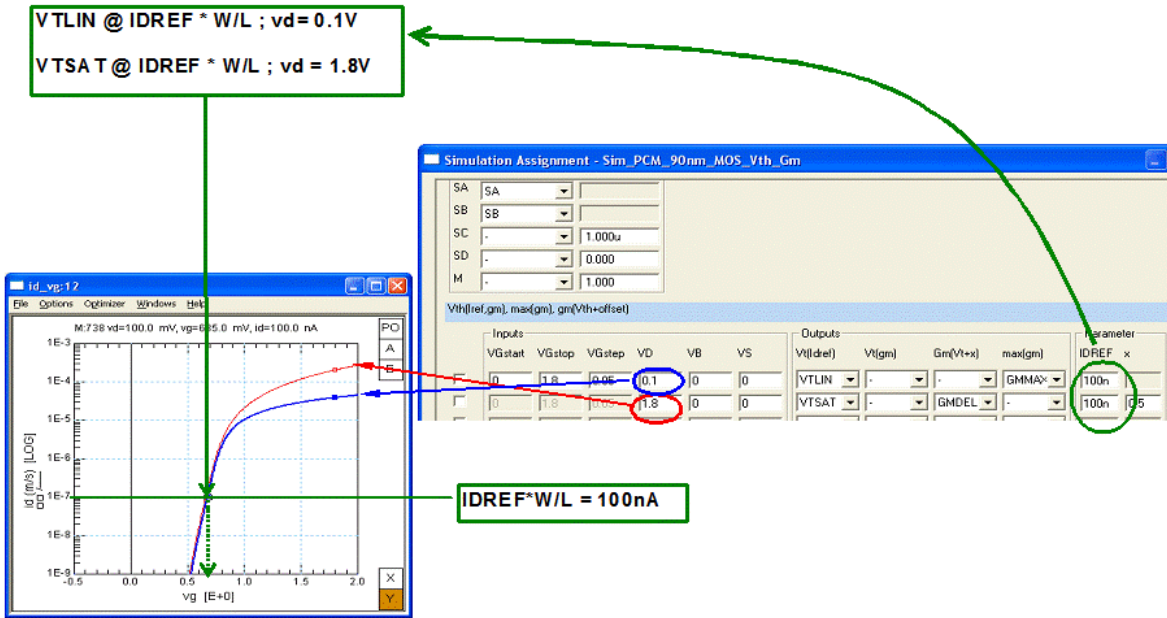


Figure 12 Simulation assignment and related outputs at a predefined parameter value, IDREF

In the example shown, the value of $IDREF$ is identical for the determination of $VTLIN$ and $VTSAT$. However, it is possible to have different $IDREF$ values. Simply enter the required values into the proper field.

NOTE

The value of $IDREF$ is multiplied by W/L for each transistor.

1 Target Modeling

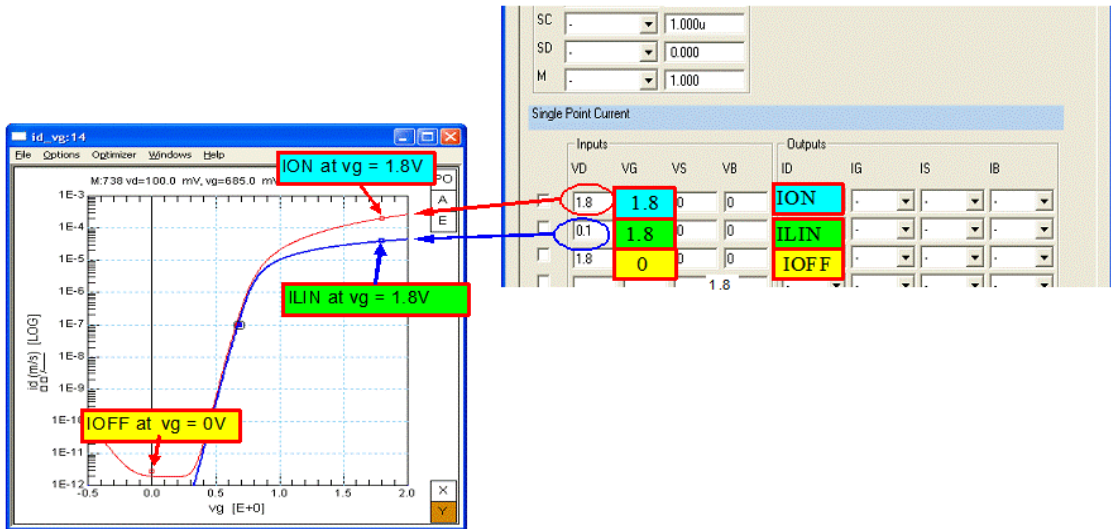


Figure 13 Single Point currents: Relationship between inputs and outputs and their representation in the form of single measurements

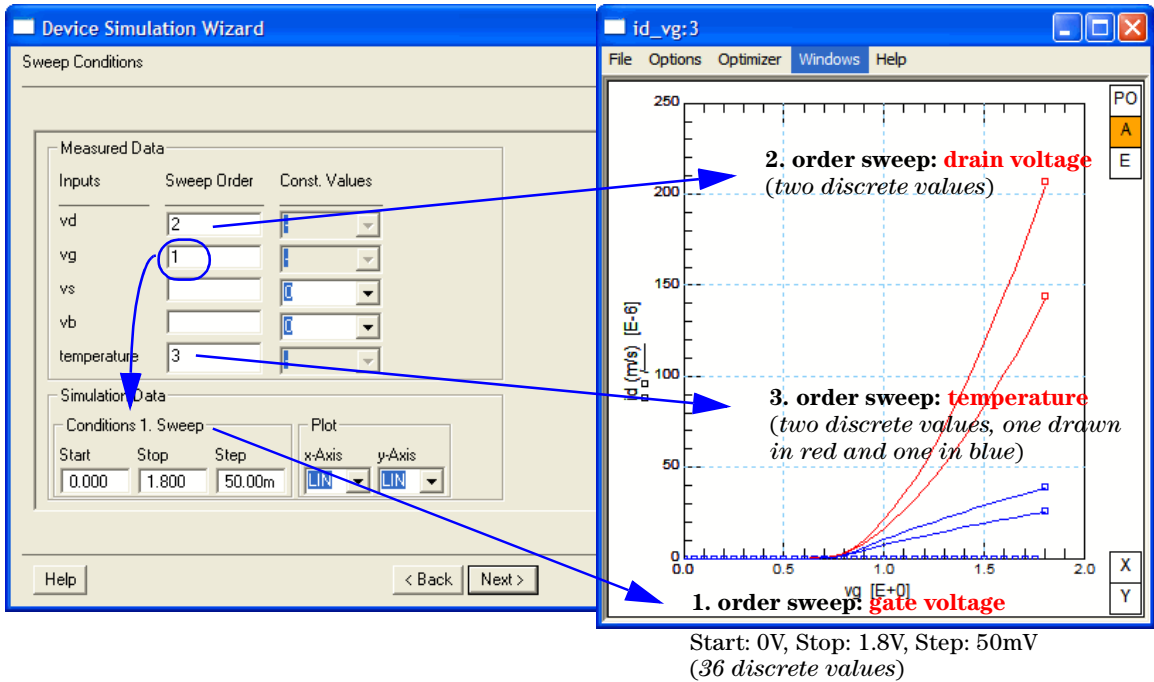
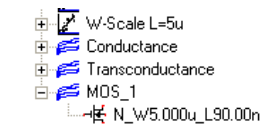


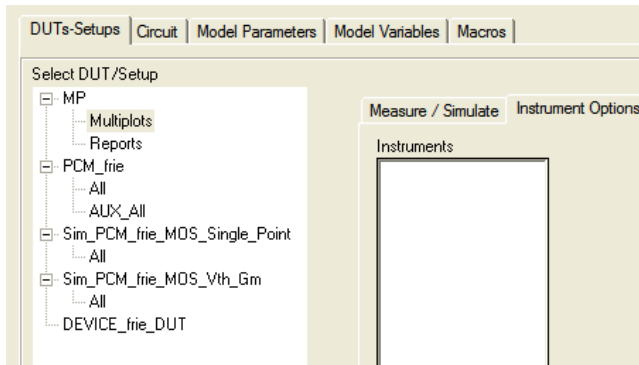
Figure 14 Entering Simulation Conditions (left) and resulting I_D - V_G diagram (right, shown for two different temperatures). The squares show actually measured points.

Again, choose *Next* and you are prompted to select devices to include in this simulation setup. After choosing the devices, choose *Finish* and the simulation setup is added to the tree of PCM data using a specific symbol.



ICCAP DUTs-Setups

Right click ICCAP DUTs-Setups to display a menu that enables you to create a new DUT, edit an *.mdl* data file, or import an *.mdl* or *.dut* file. Choose *Edit Data MDL* to display the model file in edit mode. Since you can define everything inside this *.mdl* file, you must be careful to create a functional model. The following figure shows the generated IC-CAP model file.



After creating a new DUT, you can import *.mdm* files by right clicking and choosing *Import .mdm*. An Import Wizard is opened and you are prompted to enter a name for the DUT as well as the name and path of the *.mdm* file to be imported. See the following figure.

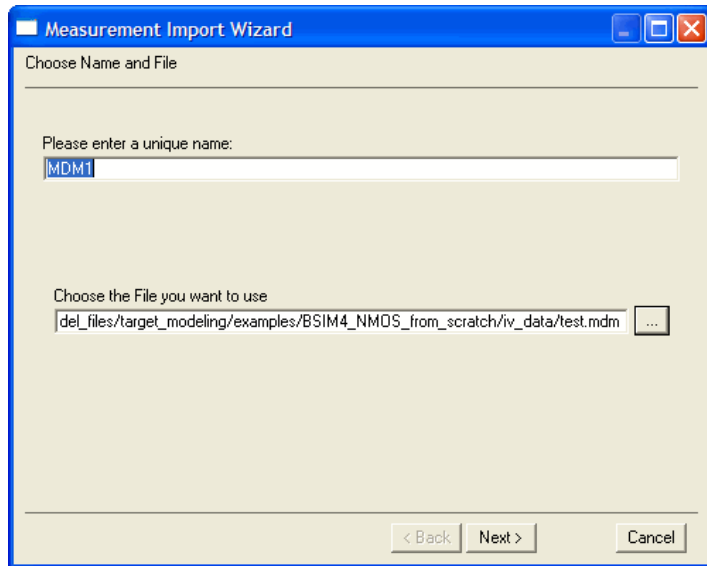
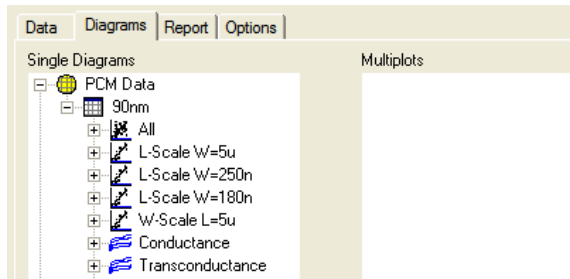


Figure 15 Measurement Import Wizard, shown is the example .mdm file provided for import

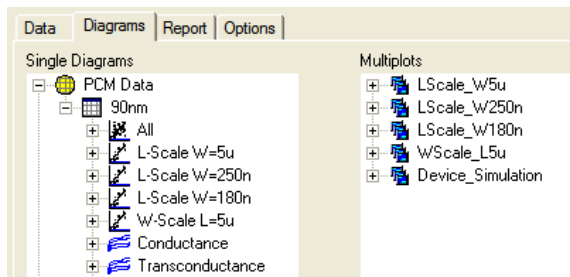
After entering the information, choose *Next* and you get a warning to check nodes, dimensions, and simulation settings. Choose *Finish* to import data into the setup and open the IC-CAP model file for editing.

Diagrams

Use the *Diagrams* tab to define plots – *Single Diagrams* on the left and *Multiplots* on the right. The Multiplots column is empty until you add a *New Multiplot* using the Diagrams menu or the icon.



When you add a new multiplot, you are prompted to enter a name. To configure a new or existing multiplot, first select the multiplot, then select the plots on the left side of the tab and click the arrow to add the plots to the selected multiplot.



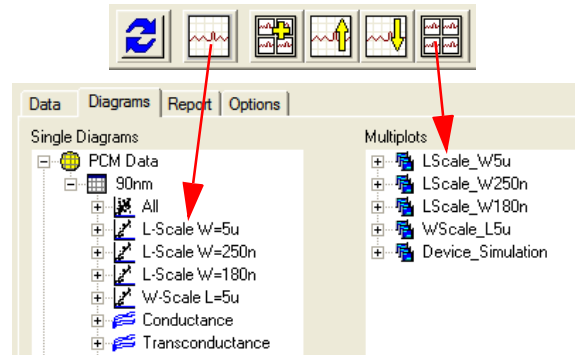
Another method to configure a multiplot is to right click a plot on the left and choose *Copy*, then right click on the desired subfolder on the right and choose *Paste*.

If you would like to change the arrangement of the plots, click the *Move Up* and *Move Down* icons (yellow arrows) or right click a multiplot and choose *Move Up* or *Move Down*.

After the plots are configured, you can open single plots by double clicking on the plot or select the plot then choose *Diagrams > Display Single Plots* from the menu or choose the *Display Plots in Single Diagrams* icon. You can open more than one single plot if you select multiple plots while pressing and holding *Shift* or *Ctrl* on your keyboard.

You can open multiplots by double clicking on the multiplot or select the multiplot then choose *Diagrams > Display Multiplots* or choose the *Display Plots in Multiplots* icon.

The *Display Plots in Single Diagrams* icon is shown on the left and the *Display Plots in Multiplots* icon is shown on the right.



Click the + sign in front of one of the PCM data entries to open the tree of single diagrams. You can then open the underlying plots by double clicking on them. However in the example above, if you open the Conductance or Transconductance diagram tree, the plots below that folder are separated by devices. Therefore, before you can open the plots, you must first open the device folders.

1 Target Modeling

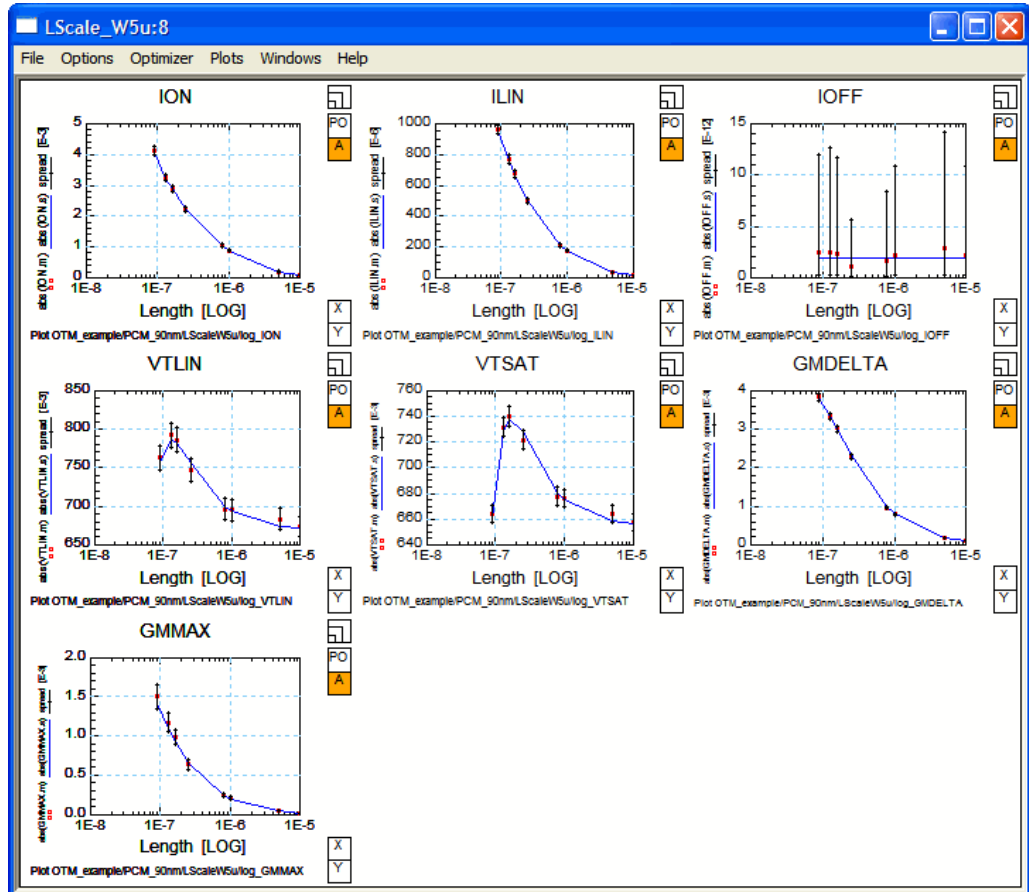
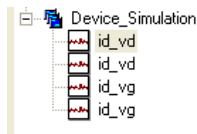
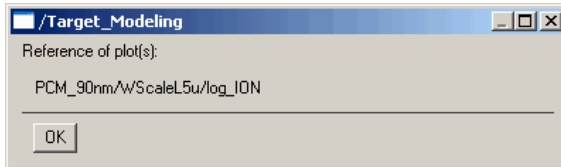


Figure 16 Multiplot for Lscale_W5u for a Target modeling project

You can also open a single plot in a multiplot by clicking the + sign in front of the multiplot to open the tree, then double click one of the single plots. See the following figure.

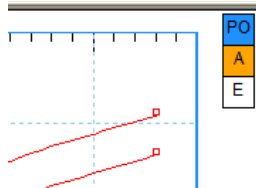


To get information regarding the origin of a plot, right click on the plot then choose *Show Origin*. A dialog box similar to the following appears.



To reconfigure plots, right click a plot then choose *Edit*. This enables you to change axis settings and so on for selected plot.

After you open a single or multiplot, you can use all the features of an IC-CAP plot window like the plot optimizer, tuner, and so on. Click the PO-field inside the plot to enable changes in the optimizer area.



Report

Use this tab to create reports for the project. Choose *Report > Configure* to open the *Configuration of Report* window. In this window you can enter a headline and comments to be entered into the report, as well as the path for the report to be stored. Moreover, you can choose to include the plot legend in the report and to set the plot size for the report.

For a new project, the *Report Structure* field on the *Report* tab is empty. Choose *Report > Set Report Structure to Default* to set the report structure to a predefined setup. This enters a default structure that you can adapt to your needs.

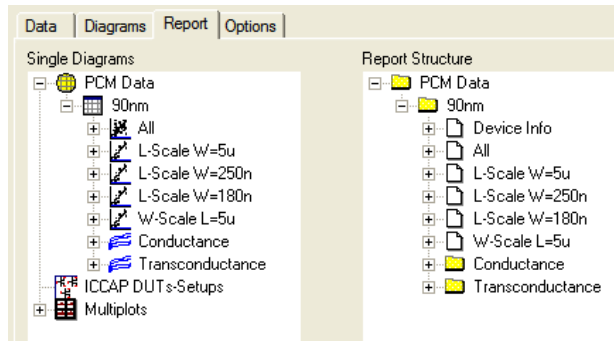


Figure 17 Default report structure

To adjust the default structure, use the right mouse button to change the order, delete, rename, and add subsets or new folders to the structure.

When finished, choose *Report > Generate Report*. The report process starts and the results are stored as *.HTML* files at the selected location. You can use any web browser to view the files.

Options

Use this tab to change which simulator is used with the project. The default simulator was selected when you created the data model.


To change which simulator or the circuit is used, choose *Options > Change Simulator and Circuit*. Select one of the available simulators as well as the circuit desired. After selecting a simulator or circuit, a window opens enabling you to check the model parameters used by the imported circuit.

Carefully check if the used circuit parameters match your needs. No checking routine is implemented since one of the goals of this toolkit is to provide flexibility to use other models or circuits without being bound to defaults.

If you would like to use a different simulator than those currently defined, edit the *Target_Modeling.mdl* file and setup the new simulator by adding it to the transform

Configuration/Options/Define Simulators. A ReadMe section at the top of the transform contains instructions on how to modify it and add your own simulator.

NOTE

When editing the data model file, use *File > RefreshTree* from the menu or the appropriate icon , for the changes to take place.

After you edit the Target Model file, close the GUI then reopen to get the changes into the GUI.

Adding Your Own Simulation Templates

How to implement a user defined simulation template is described in the “Simulation Templates for Target Modeling” support document. This procedure requires you to be very familiar with IC-CAP because model files have to be edited. There is no automatic error check available, since you have all the capability that IC-CAP provides.

