

NOTICE: This document contains references to Agilent Technologies. Agilent's former Test and Measurement business has become Keysight Technologies. For more information, go to **www.keysight.com**.





Agilent Technologies

**IC-CAP 2010.08
July 2010
Target Modeling**

This is the default Notice page

© **Agilent Technologies, Inc. 2000-2010**

3501 Stevens Creek Blvd., Santa Clara, CA 95052 USA

No part of this documentation 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.

Acknowledgments

UNIX ® is a registered trademark of the Open Group.

MS-DOS ®, **Windows** ®, and **MS Windows** ® are U.S. registered trademarks of Microsoft Corporation.

Pentium ® is a U.S. registered trademark of Intel Corporation.

PostScript® is a trademark of Adobe Systems Incorporated.

Java™ is a U.S. trademark of Sun Microsystems, Inc.

Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries.

Qt Version 4.5

Qt Notice

The Qt code was modified. Used by permission.

Qt Copyright

Qt Version 4.5, Copyright (c) 2009 by Nokia Corporation. All Rights Reserved.

Qt License Your use or distribution of Qt or any modified version of Qt implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included.

Qt Availability <http://www.qtsoftware.com/downloads>

Patches Applied to Qt can be found in the installation at:

\$HPEESOF_DIR/prod/licenses/thirdparty/qt/patches.

You may also contact Brian Buchanan at Agilent Inc. at brian_buchanan@agilent.com for more information. For details see:

http://bmaster.soco.agilent.com/mw/Qt_License_Information

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 documentation 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

IC-CAP 2011.01 - Target Modeling
(Technical Data - Commercial Items) and DFARS 227.7202-3 (Rights in Commercial
Computer Software or Computer Software Documentation).

IC-CAP 2011.01 - Target Modeling

Graphic User Interface for Target Modeling	3
Using the Target Modeling Tool	6
Options	20
Adding Your Own Simulation Templates	20

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.

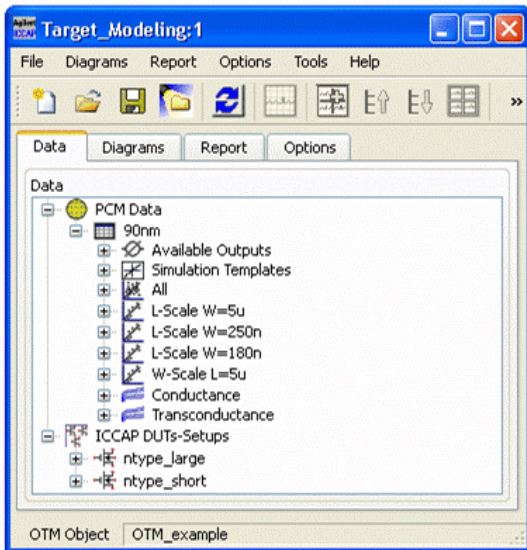
The control module is an IC-CAP *.mdl* file, which manages the OTM objects and handles templates necessary for PCM simulation, report generation, and so on.

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.

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.

- Back to *Target Modeling* (target)

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.

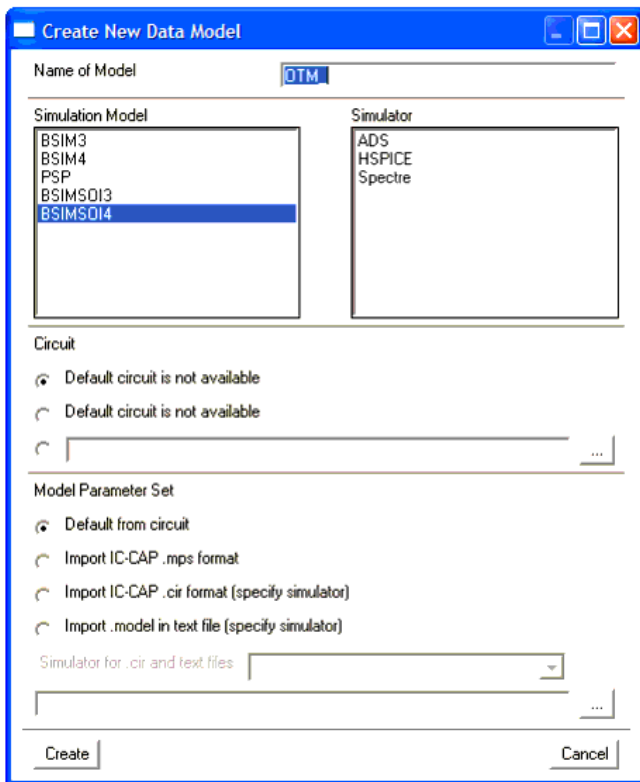
**Caution**

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).

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.

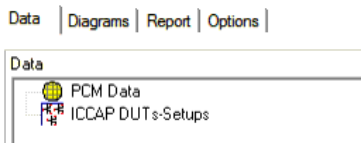
Next, select the model parameter set. You can use the default model parameter set included in the circuit or import one in IC-CAP *.mps* or *.cir* format. For example, import the example model parameter set from `$ICCAP_ROOT\examples\model_files\target_modeling\examples\BSIM4_NMOS_from_scratch\model_parameters`.

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.

- Back to *Target Modeling* (target)

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

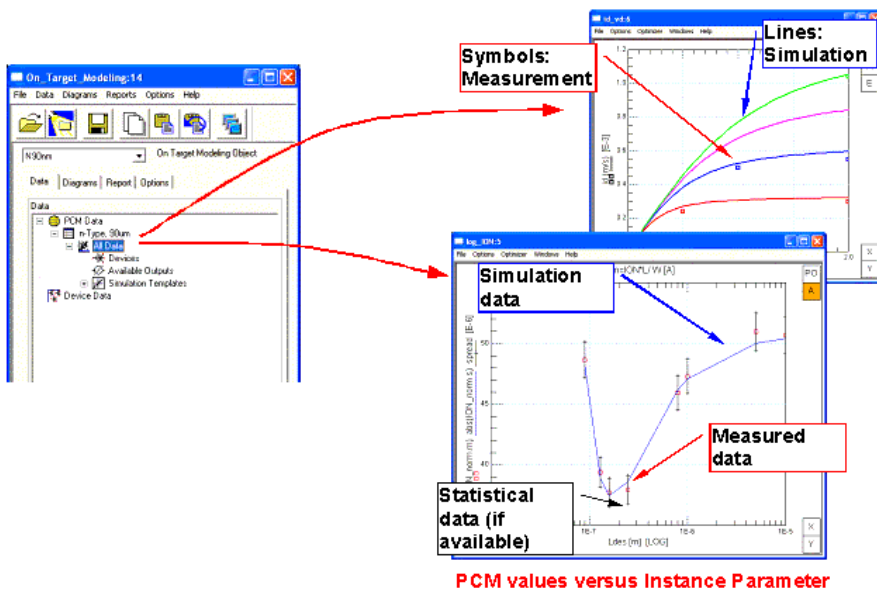
Those plots are generated by configuring Simulation Templates.

- 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.

Possible kinds of diagrams

I-V like diagrams



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

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

Two similar templates for the BSIMSOI3 and BSIMSOI4 models are implemented (for floating body SOI):

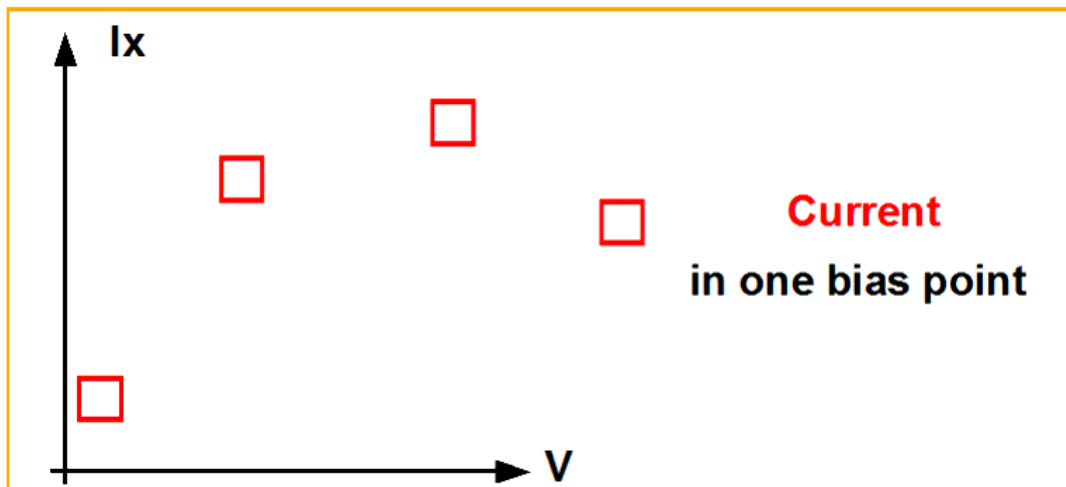
- Single Point current
- $V_{th}(I_{ref}, gm)$, $\max(gm)$, $gm(V_{th} + \text{offset})$

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

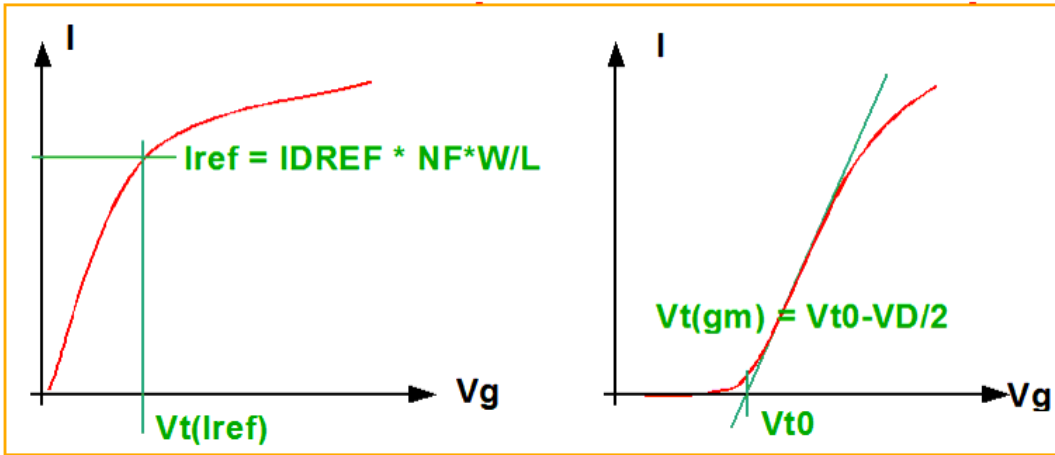
A support document named "Simulation Templates for Target Modeling" describes how to implement a user defined model.

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

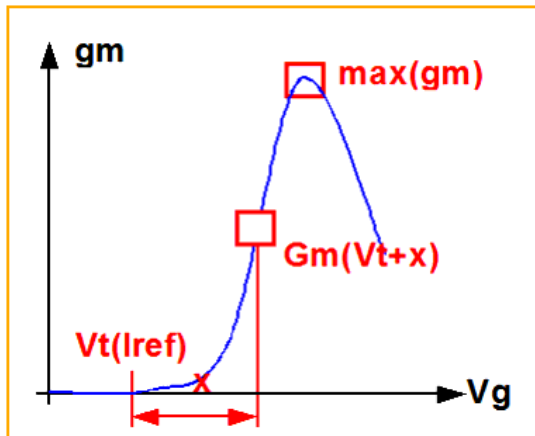
MOS single point simulations for I/V-diagrams



MOS threshold voltage related simulation V_{th}



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 cannot 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.

Simulation Assignment

Simulation Assignment - Sim_PCM_90nm_MOS_Single_Point

Instance Parameters

Parameters	PCM	Default
L	Length	
W	Width	
NF	NF	
AD	AD	
AS	AS	
PD	PD	
PS	PS	
SA	SA	
SB	SB	
SC	.	1.000u
SD	.	0.000
M	.	1.000

Single Point Current

Inputs				Outputs			
VD	VG	VS	VB	ID	IG	IS	IB
☐ 1.8	1.8	0	0	ION	.	.	.
☐ 0.1	1.8	0	0	ILIN	.	.	.
☐ 1.8	0	0	0	IOFF	.	.	.
☐				.	.	.	.
☐				.	.	.	.
☐				.	.	.	.
☐				.	.	.	.

Export Import Add Delete OK Cancel

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.

Single Point Current

Inputs				Outputs			
VD	VG	VS	VB	ID	IG	IS	IB
☐ 1.8	1.8	0	0	ION	.	.	.
☐ 0.1	1.8	0	0	ILIN	.	.	.
☐ 1.8	0	0	0	IOFF	.	.	.

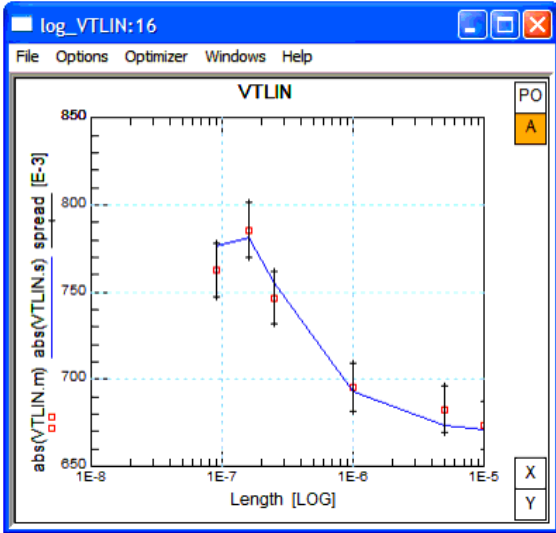
You can add more points (lines) or you can delete points by choosing the appropriate button at the bottom of the Simulation Assignment window. Mark the lines you would like to delete by selecting the check box in front of the line(s).

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

Vth(Iref,gm), max(gm), gm(Vth+offset)											
Inputs						Outputs				Parameter	
VGstart	VGstop	VGstep	VD	VB	VS	Vt(Iref)	Vt(gm)	Gm(Vt+x)	max(gm)	IDREF	x
0	1.8	0.05	0.1	0	0	VTLIN	-	-	GMMAX	100n	
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 IDREF column.

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

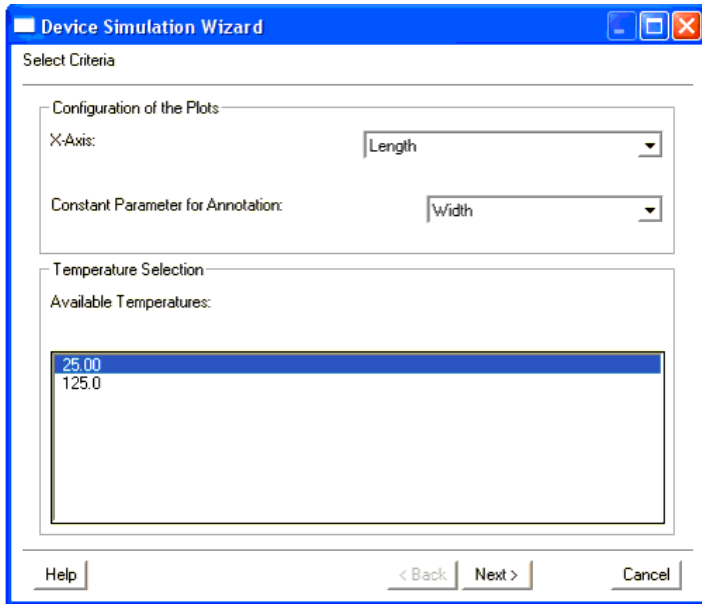
From the directory

\$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.

Selections for a subset



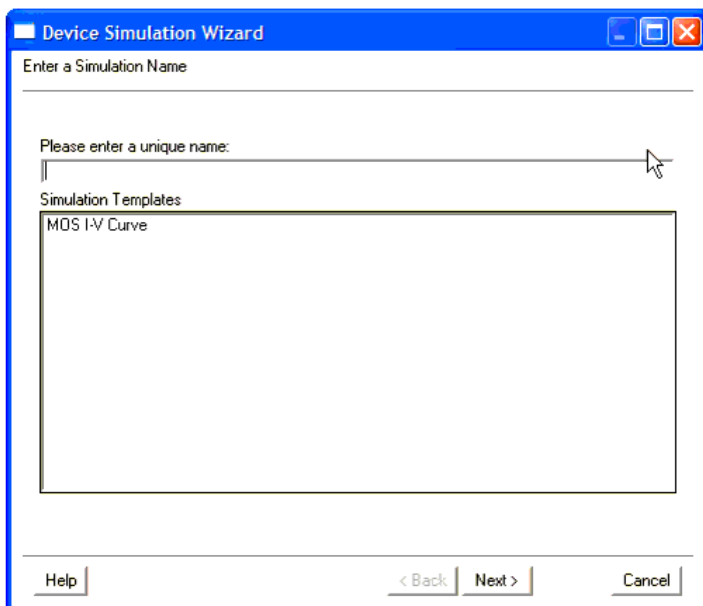
You can configure plots for the new subset based on different selectable X-Axis parameters as well as annotation parameters and temperatures, if available.

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

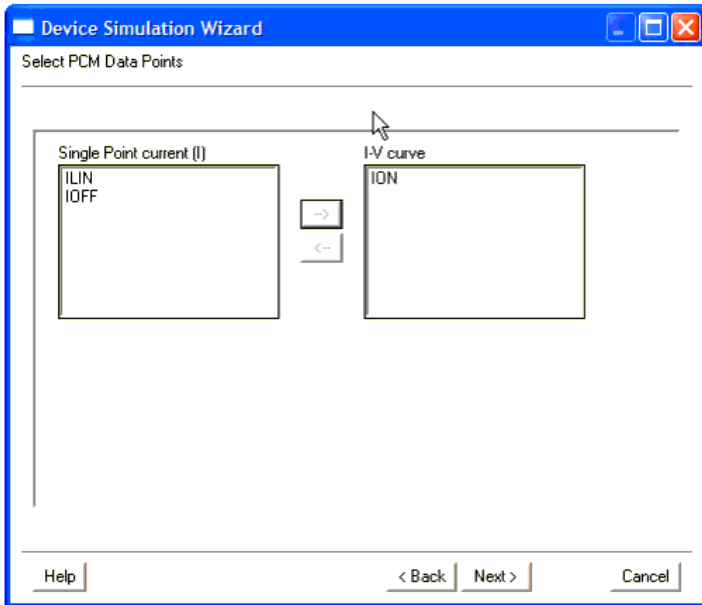
To derive device data, right click a PCM table or a subset then choose *Derive Device Data*. The Device Simulation Wizard window opens. Enter a name for the simulation and choose an available simulation template from the list, then choose *Next*.

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.

Assignment of predefined simulations

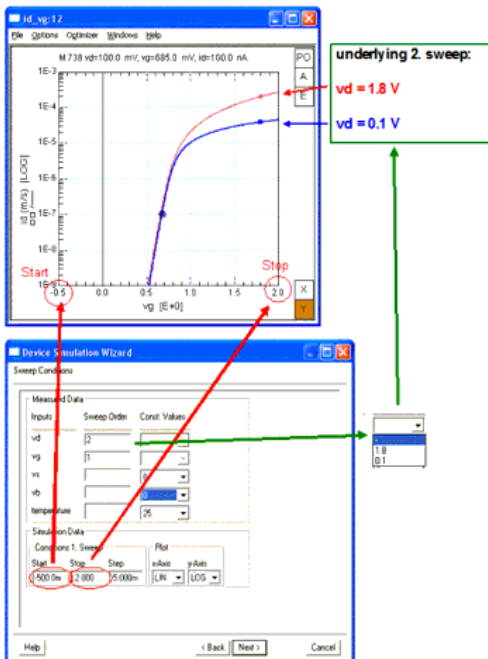


See [MOS single point simulations for I/V-diagrams](#) and [MOS threshold voltage related simulation Vth](#) for a graphical explanation of Single Point or Vth related items.

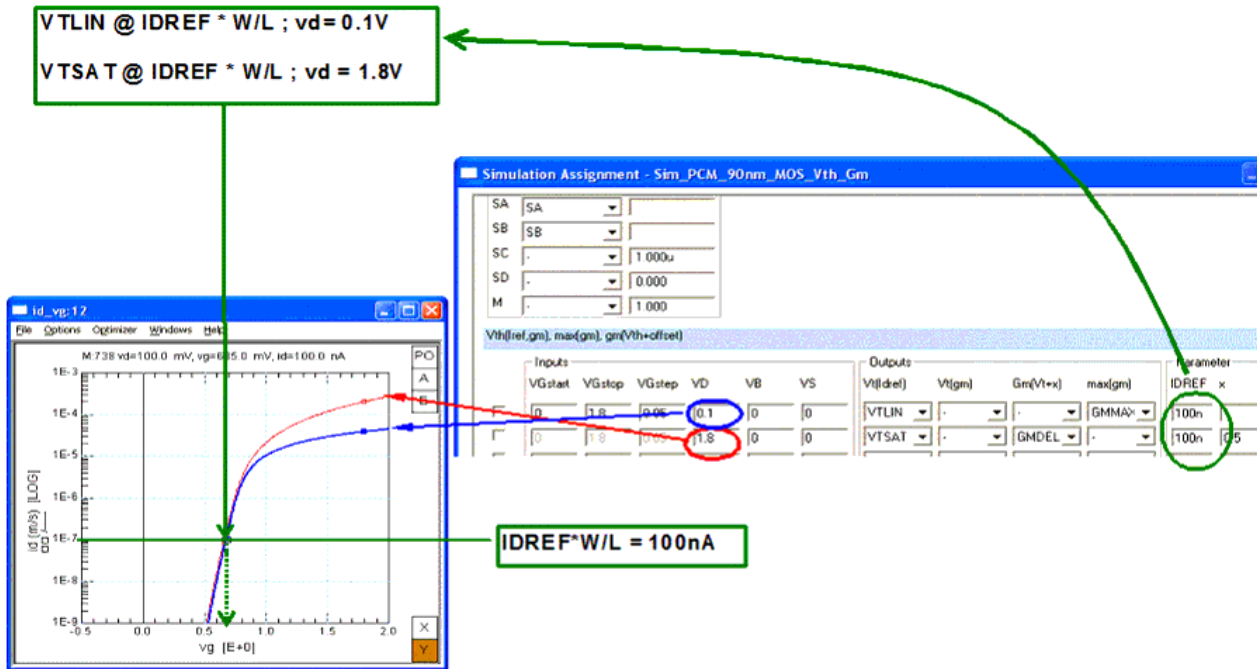
When you're done assigning data points, choose Next to enter simulation conditions. See [Device Simulation Wizard](#).

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 cannot be changed in this window.

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



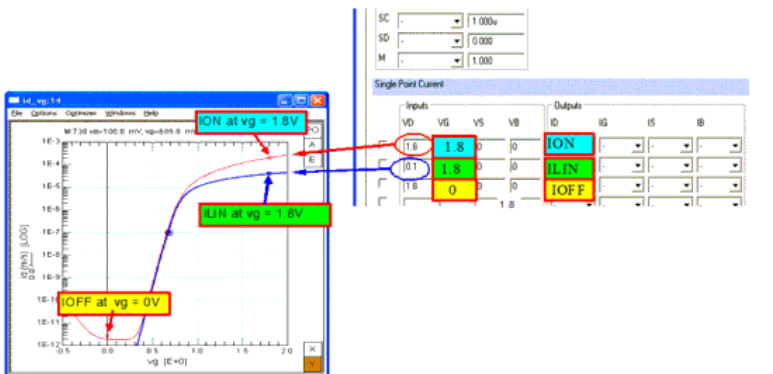
Simulation assignment and related outputs at a predefined parameter value, IDREF



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

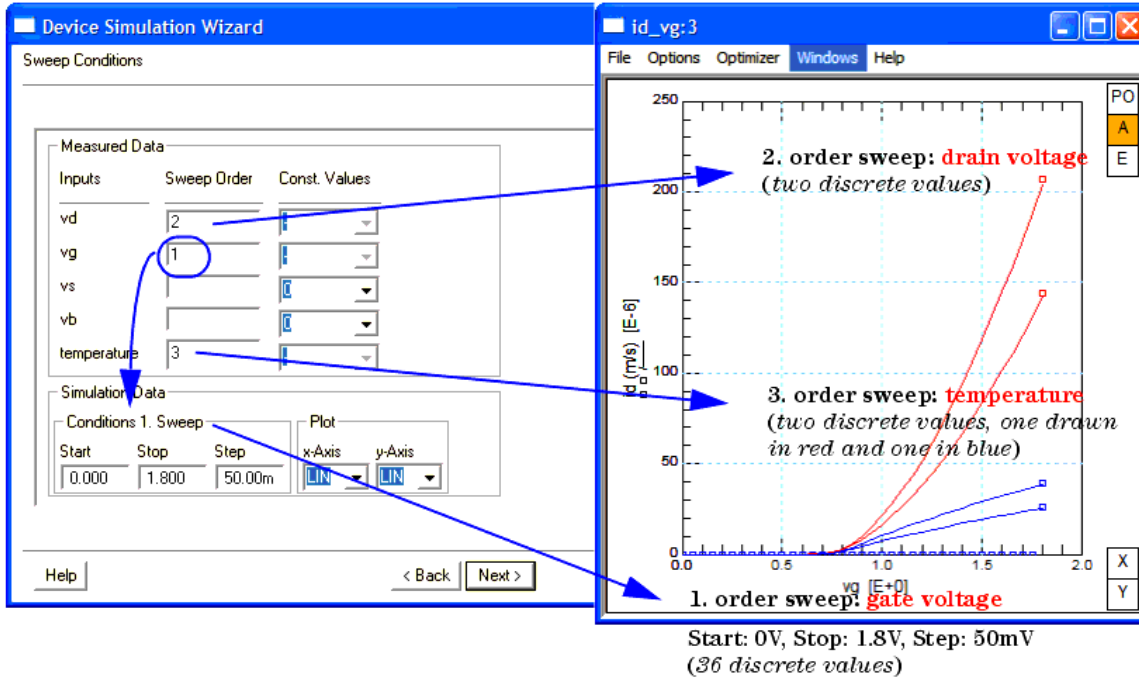
Note
The value of $IDREF$ is multiplied by W/L for each transistor.

Single Point Currents



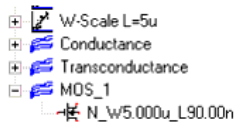
Relationship between inputs and outputs and their representation in the form of single measurements

Device Simulation Wizard



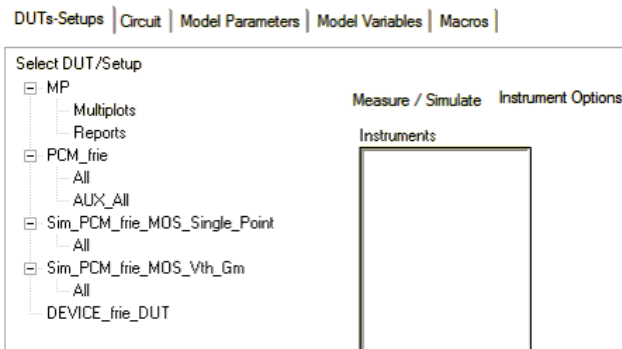
Entering Simulation Conditions (left) and resulting ID-VG diagram (right, shown for two different temperatures). The squares show actually measured points.

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.



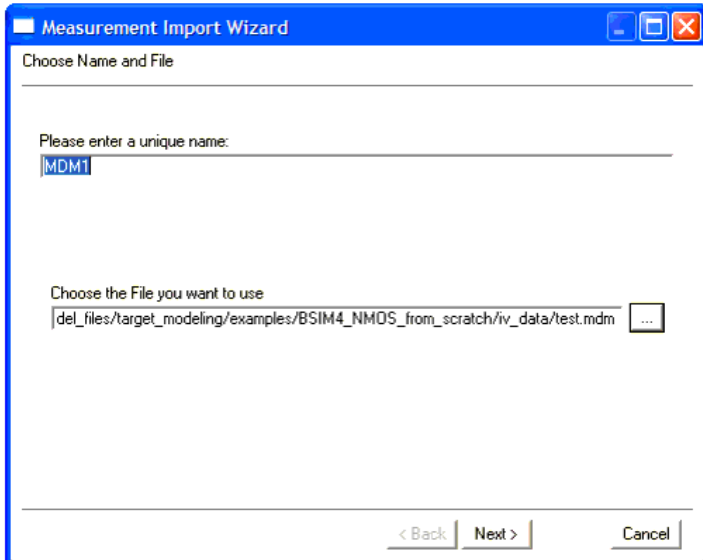
ICCAP DUTs-Setups

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 for an example .mdm file provided for import.

Measurement Import Wizard

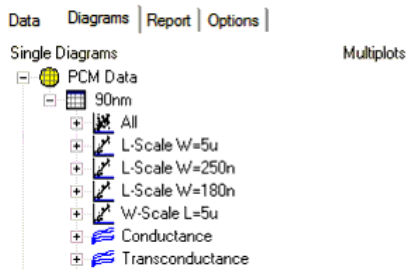


After entering the information, choose Next and you get a warning to check nodes, dimensions, and simulation settings. Choose Finish to import data into the setup and open the IC-CAP model file for editing.

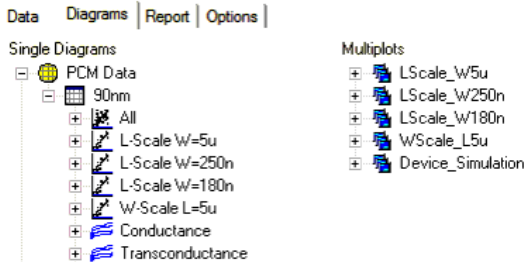
- Back to *Target Modeling* (target)

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.



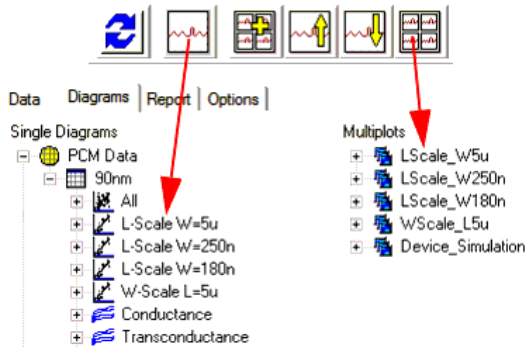
Another method to configure a multiplot is to right click a plot on the left and choose Copy, then right click on the desired subfolder on the right and choose Paste.

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.

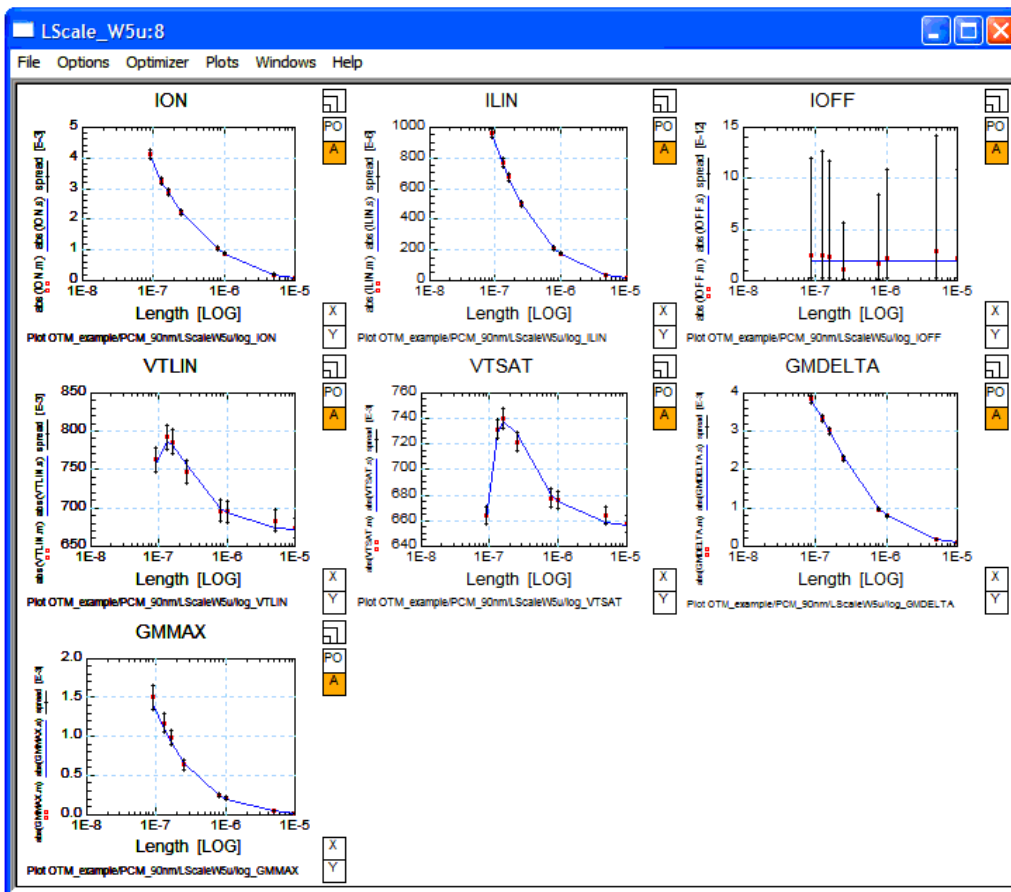
You can open multiplots by double clicking on the multiplot or select the multiplot then choose Diagrams > Display Multiplots or choose the *Display Plots in Multiplots* icon.

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

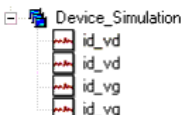


Click the + sign in front of one of the PCM data entries to open the tree of single diagrams. You can then open the underlying plots by double clicking on them. However in the example above, if you open the Conductance or Transconductance diagram tree, the plots below that folder are separated by devices.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

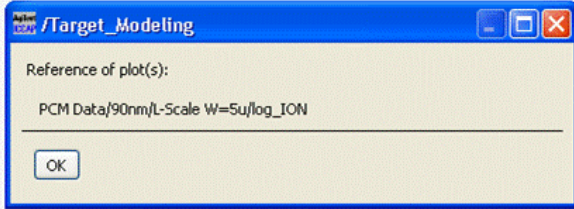
Multiplot for Lscale_W5u for a Target modeling project



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.

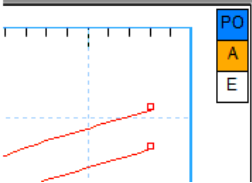


To get information regarding the origin of a plot, right click on the plot then choose Show Origin. A dialog box similar to the following appears.



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.



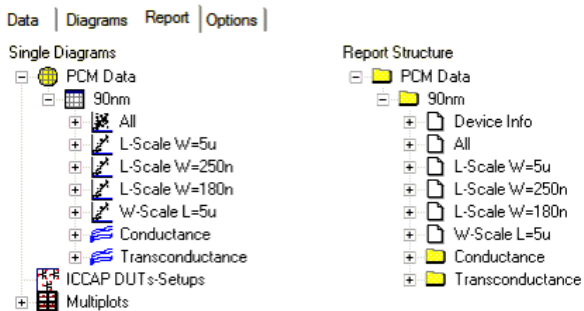
- Back to *Target Modeling* (target)

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.

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.

- Back to *Target Modeling* (target)

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 being 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.

- Back to *Target Modeling* (target)