MBP 2017

Applications

KEYSIGHT TECHNOLOGIES

Notices

Restricted Rights Legend

If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as "Commercial computer software" as defined in DFAR 252.227-7014 (June 1995), or as a "commercial item" as defined in FAR 2.101(a) or as "Restricted computer software" as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause.

Use, duplication or disclosure of Software is subject to Keysight Technologies' standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), as applicable in any technical data.

Portions of this software are licensed by third parties including open source terms and conditions. For detail information on third party licenses, see Notice.

Contents

Application Notes . . . 6
- Add Additional Instance Parameter . . . 6
  - Overview . . . 6
  - Steps . . . 6
- Advanced Graph Export . . . 8
  - Overview . . . 9
  - Basic Concepts . . . 9
  - Export Graphics . . . 10
- Binned Model Generation and Tweaking . . . 14
  - Overview . . . 14
  - Generate Binned Models . . . 14
  - Three Options Regarding Binned Model . . . 15
  - Tweak Binned Models . . . 17
- Call External Simulator . . . 20
  - Overview . . . 20
  - Call Local HSPICE . . . 21
  - Call Remote HSPICE . . . 21
- DC/CV MDM Loader Configuration . . . 22
  - Overview . . . 22
  - Circuit Standard . . . 23
  - List of Standard Node Names and Instance Names . . . 24
  - Mapping . . . 26
- Generate and Tweak Corner Model . . . 28
  - Overview . . . 28
  - Generate Corner Models . . . 28
  - Tweaking Corner Models . . . 31
- Implementing Verilog-A Model . . . 34
  - Overview . . . 34
  - Preparation . . . 35
  - Sample Models . . . 35
  - MBP Supported Functions and Keywords . . . 36
- Implementing MOSRA Models . . . 41
  - Overview . . . 41
  - Raw Data Format . . . 42
  - MOSRA Simulation and Parameter Extraction . . . 43
- Load Multi-Die Data . . . 47
  - Overview . . . 47
  - Script-based Environment Setup . . . 47
  - Customization . . . 49
- Mismatch Modeling . . . 49
  - Overview . . . 49
  - Data, Plot and MC Simulation . . . 50

Extraction Flow 52
Mismatch IMV 58
Multiple Simulations 60
Overview 61
Double Simulation 61
Multiple Models Comparison 63
Optimization Weight Setting 64
Overview 65
Device Weight Setting 65
Curve Weight Setting 68
Parameter Boundry 70
Overview 70
Double Simulation 71
Statistical Modeling 74
Overview 75
Data, Plot and Simulation 75
BPV Extraction Flow 80
Statistical IMV 86
Tmi Aging Model Application 88
TMI Aging Model Application 88
Overview 89
Preparation 89
Simulation and Parameter Extraction 90
Transient Simulation 93
Transient Simulation 93
PCA Model Extraction 95
PCA Model Extraction 95
Statistical Model Extraction Module 107
Introduction 107
Start the statistical model extraction module 108
Setup the statistical model extraction progress 109
SRAM Solution Operation Note 121
Statistical Model Extraction Module 133
General Introduction 133
Work Flow 135
Demo Examples 158
Q&A 178

- Application Notes

Application Notes

- Add Additional Instance Parameter
- Advanced Graph Export
- Binned Model Generation and Tweaking
- Call External Simulator
- DCorCV MDM Loader Configuration
- Generate and Tweak Corner Model
- Implementing Verilog-A Model
- Implementing MOSRA Models
- Load Multi-Die Data
- Mismatch Modeling
- Multiple Simulations
- Optimization Weight Setting
- Parameter Boundry
- Statistical Modeling
- Tmi Aging Model Application
- Transient Simulation
- PCA Model Extraction
- Statistical Model Extraction Module
- SRAM Solution Operation Note
- Statistical Solution Application

Add Additional Instance Parameter

This application note describes how to add additional instance parameters in Model Builder Program (MBP).

NOTE This document was originally released for MBP V2011.1.0 in August 2011.

Overview

MBP accepts user-defined instance parameters in both model file and measurement data. In this document, the steps required to add new instance parameters in MBP are covered. For more information go to http://www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at http://www.keysight.com/main/contactInformation.jspx?tmprop=TM&cc=SG&lc=eng.

Steps

You can follow the steps listed below to complete the process of adding additional instance parameters in MBP.

Model File

To begin, you must first edit the model file to which the new instance parameter will be added. For example, assume that you have specified an instance parameter *xxd*; (in this case, *xxd* has no real physical meaning) in one subcircuit model as follows:

```
.param

+xvth0 = 1e5

. subckt macromodel d g s b w=1e-6 l=1e-6 xxd=1e-6
sa=1e-6 sb=1e-6

.param

+dvth0 = _xxd*xvth0_

m1 d g s b nmos w=w l=l xxd=xxd sa=sa sb=sb

.model nmos nmos

*** Flag Parameter ***

+level = 49 version = 3.2 binunit = 2

+vth0 = _0.7+dvth0_;

......

.ends
```

In this example, the new instance *xxd*; is added and declared in the model file. The parameter *dvth0* is expressed as the product of *xxd* and *xvth0.*

Instance Configuration File

Open the file *inst.ini* under **`$MBP_HOME\etc\hspice\mosfet\bsim3v3\inst_core`**. Append the *xxd*, item to the file. Here, **`$MBP_HOME`** is the MBP installation path.

```
m1 d g s b nmos w=<1e-6,W,1e-6,0,um> ... ps=<0,PS,1,0,
m> xxd=<1e-6,xxd,1,0,m>
```

The file *inst.ini* claims the model name and instance parameter of the certain device. MBP creates a netlist according to this file. In this example, the instance named *xxd* is added with the default value of *0*, a unit scale of *1*, an offset value of *0*, and the unit name *m*.

Measurement File

Next, you must specify *xxd* in the measurement file. For example:

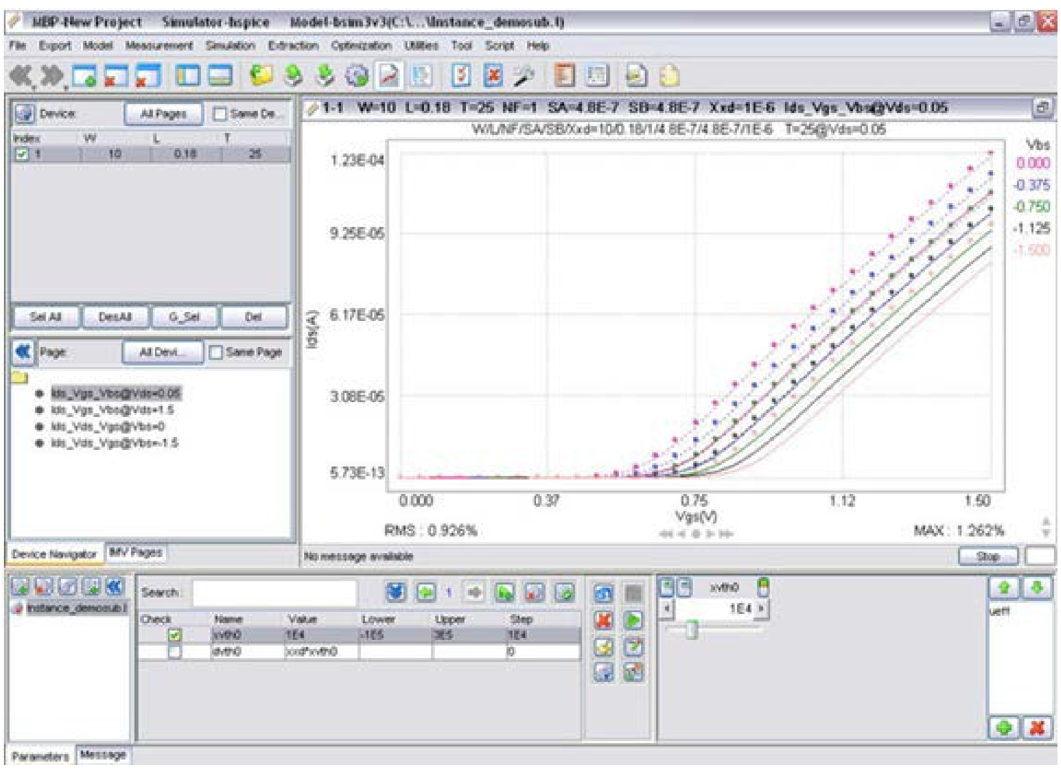
```
condition{mode=forward,type=nmos,modtype=DC}

Page(name=Ids_Vgs_Vbs,x=Vgs,p=Vbs,y=Ids) { Vds=0.05,W=10
.0,L=0.18,NF=1.0,SA=4.8E-7,SB=4.8E-7,T=25.0,xxd=1e-6}
```

In this example, the new instance *xxd*; is added with the value of *1e-6*;

Simulation

Finally, load the modified model into MBP. Once the user tunes *xvth0*, which is correlated to *xxd*, the change may be observed in the simulation curves, as shown in the following figure.

Add the xxd instance parameter



Advanced Graph Export

This application note describes how to customize a report to be exported in Model Builder Program (MBP).

NOTE This document was originally released for MBP V2011.1.0 in July 2011.

Overview

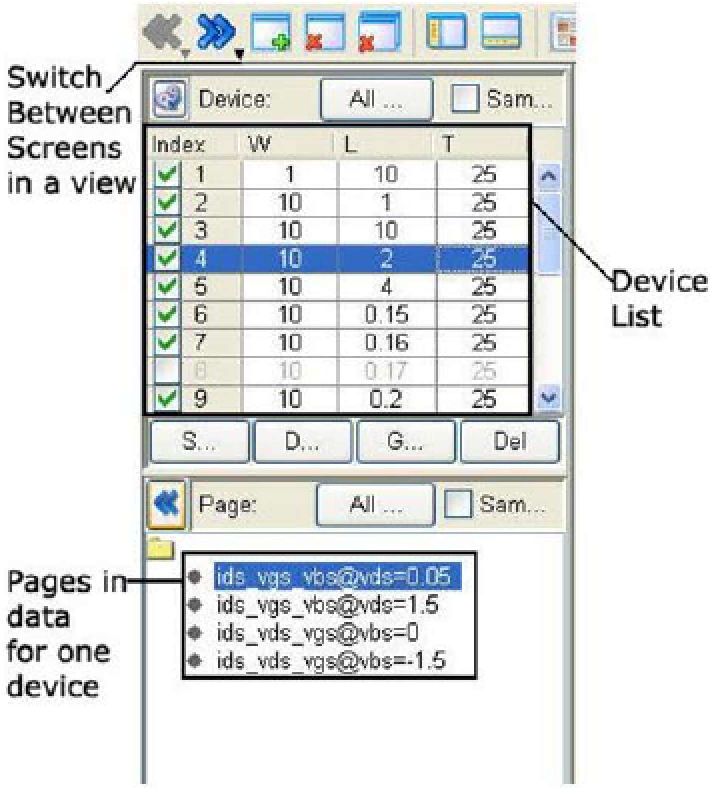
In addition to the basic graph export functions like *All Pages Current View* and *Screen In One* that MBP supports, the software also provides an advanced graph export function that allows you to customize reports according to their specific needs.This document introduces the options and steps for exporting graphs using this advanced mode. For more information go to www.keysight.com or contact your local Keysight office. The complete list is available at www.keysight.com

Basic Concepts

Before introducing MBP's advanced graph export function, a clear and solid understanding of the software's plot category methodology is necessary. To begin, the concepts of Device, Page and View in MBP are defined as follows

- Device: Some measurement data and model simulation results are included under each device. Different devices are included in the device array at the top of the device navigator.
- Page: Page means a set of curves under every device, like Id_Vg and Id_Vd. This information is shown in the page list at the bottom part of the device navigator.
- View : Another plot sequence named *view* is independent from the device array and page list. View means that all of the plots are contained in the current screen. However, because of space limitations in the display area, MBP cannot display all of the plots at the same time. Instead, MBP displays some plots according to the layout configuration and hides the rest. You can loop the plots in View using the blue arrow buttons in the tool bar. As shown in the figure device list, you can add plots to the current view by clicking the Add screen button beside the blue arrow. The math transform and the log /linear scale will also be memorized in a View. Pay attention to the difference between View and Screen. A View is a kind of extension of Screen and may have more than one screen.

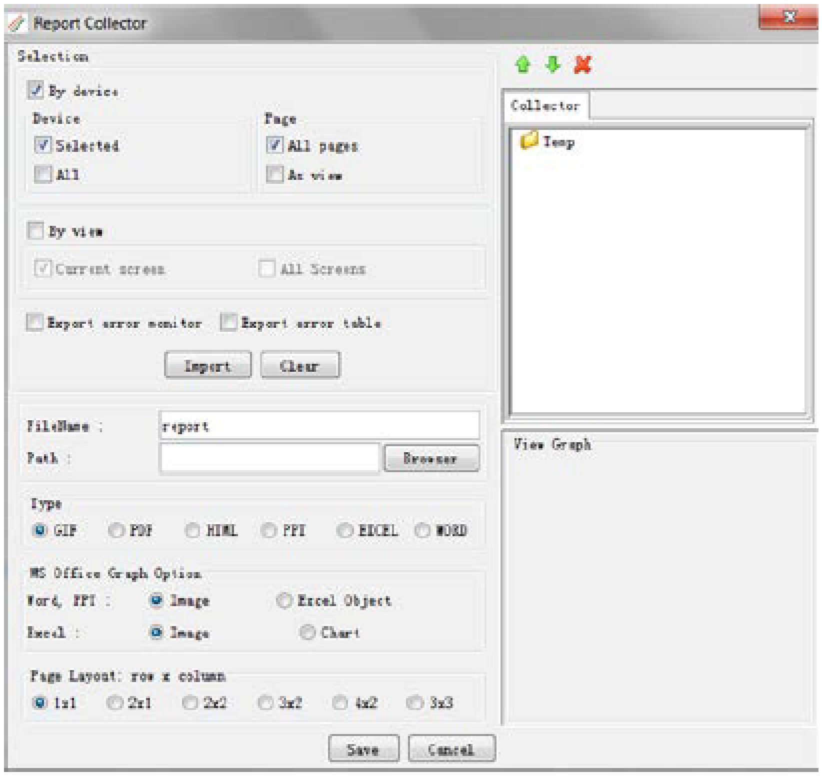
Device list



Export Graphics

MBP's advanced export function is based on the Device, Page and View grouping categories. To export graphics, you must first click Export > Export Graph > Advanced from the main menu. This pops up the window called Report Collector, as shown in the following figure:

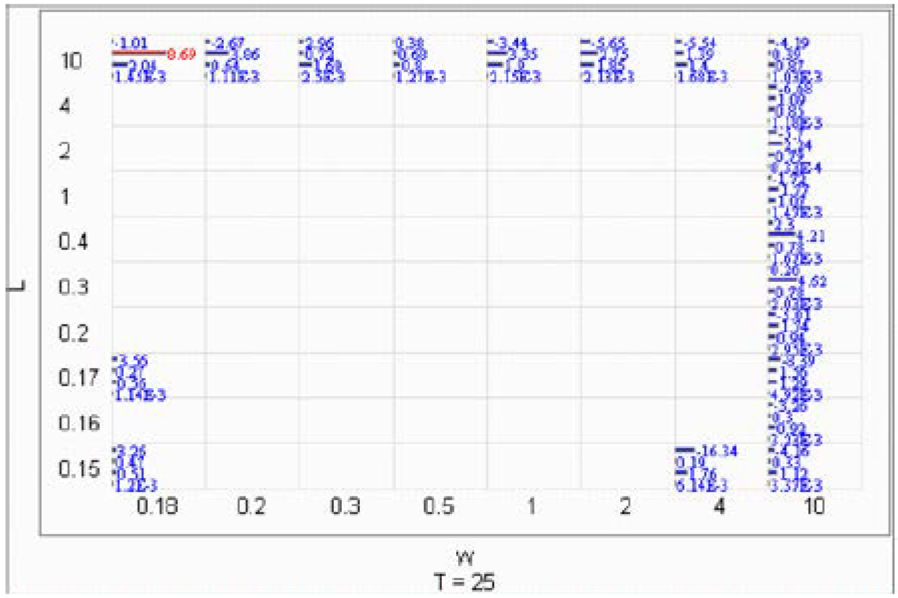
Report Collector



Choose exported graphs

You must select the graphs to be exported. MBP provides two options in this regard: to export By view or By device You can choose either option.If By view is checked, there are two options you can choose from. One is to export the graphs in the current screen by checking Current screen. The other option is to export the graphs in all screens by checking All screens .If By device is checked, you have more options. First, you must specify the devices to be exported, either choosing all devices by checking All or only the devices selected in device navigator by checking Selected . Then, define the page to be exported for each device. Here, there are two options. All pages exports all of the pages for the selected devices, while the As View option enables you to define a set of plots in a view and export the graphs of the selected devices according to the current view. If you check the Export error monitor option, the error monitor graph as shown in the figure below will also be exported.

Error Monitor graph



By checking the Export error table option, the error tables as shown in the following figure will be added to the report.

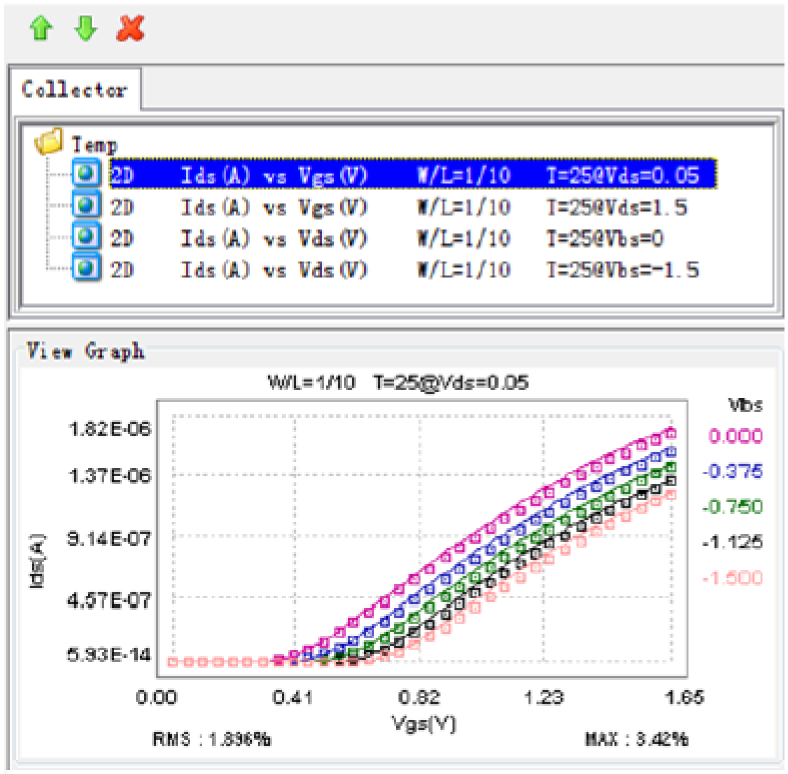
Error table

Page	W	L	T	Bias	Math	RMS	MAX
ids_vds_vgs	1	10	25	vbs=0	y	1.96%	4.19%
ids_vds_vgs	1	10	25	vbs=-1.5	y	2.91%	5.50%
ids_vds_vgs	10	1	25	vbs=0	y	1.58%	4.80%
ids_vds_vgs	10	1	25	vbs=-1.5	y	1.28%	3.15%
ids_vds_vgs	10	10	25	vbs=0	y	0.38%	1.73%
ids_vds_vgs	10	10	25	vbs=-1.5	y	1.60%	3.51%
ids_vds_vgs	10	2	25	vbs=0	y	1.22%	3.83%
ids_vds_vgs	10	2	25	vbs=-1.5	y	1.12%	2.67%
ids_vds_vgs	10	4	25	vbs=0	y	0.59%	2.44%
ids_vds_vgs	10	4	25	vbs=-1.5	y	1.52%	3.60%

Collector

The next step, is to click Import so that all of the graphs to be exported will be placed in the *Collector* section, as shown in the figure below. Here, you can adjust the graph sequence by clicking the Move up or Move down icons. Then, click Remove icon to remove the selected graph from the Collector . When a file is selected in the Collector , its corresponding graph will be plotted automatically in the View Graph section.

Collector section



Choose the Exported File Format

MBP allows multiple formats to be exported, including GIF (.gif), PDF (.pdf), HTML (. html), PPT (.ppt or
.pptx), EXCEL (.xls or .xlsx) and WORD (.doc or .docx). You may select any format desired in the Type section. If the exported file type is PPT, Excel or Word, you will have more options to choose from in the MS Office Graph Option section.

1. If the file type is PPT or Word, there are three options:
   - Choose Image in Word, PPT to export the graphs only in the generated PPT or Word file.
   - Choose Excel Object in Word, PPT and Image in Excel to export graphs (plotted by MBP) with data in Excel format in the generated PPT or Word file.
   - Choose Excel Object in Word, PPT and Chart in Excel to export graphs (plotted by Excel) with data in Excel format in the generated PPT or Word file.
2. If the file type is Excel, there are two options:
   - Choose Image in Excel to export the data and graphs (plotted by MBP) in the generated Excel file.
   - Choose Chart in Excel to export the data and graphs (plotted by Excel) in the generated Excel file.

Choose Exported File Format

After confirming the page layout (row x column), you can then click Save to generate the customized report.

Binned Model Generation and Tweaking

This application note describes how to generate and tweak binned models in Model Builder Program (MBP).

NOTE This document was originally released for MBP V2011.1.0 in July 2011.

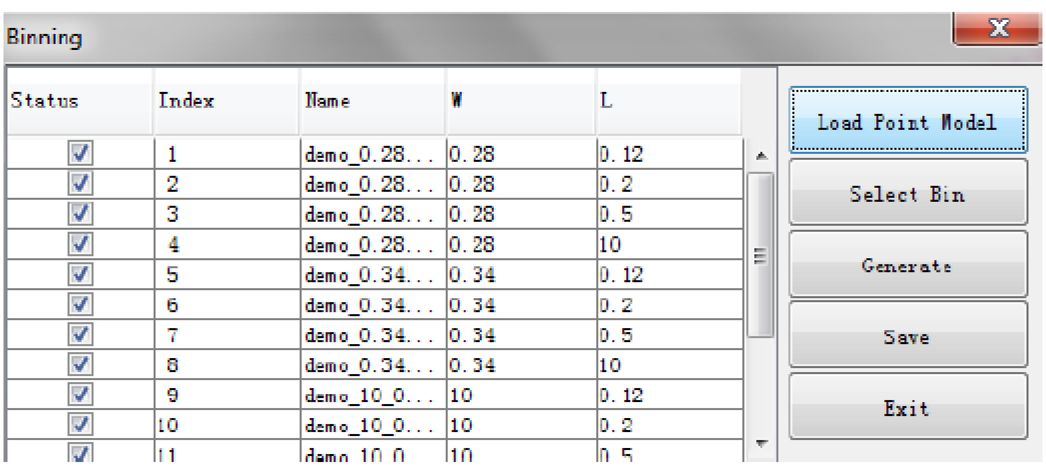
Overview

With process variation or when there is a requirement to fit a specific target, tweaking is generally used as a fast way to achieve the corresponding models. For a binned model, the modeling engineer should always pay attention to ensure continuity is kept from bin-to-bin. MBP features an integrated binning tweak capability to meet this need. In this document, we introduce the steps required to generate and tweak binned models. For more information go to www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at www.keysight.com/find/contactus

Generate Binned Models

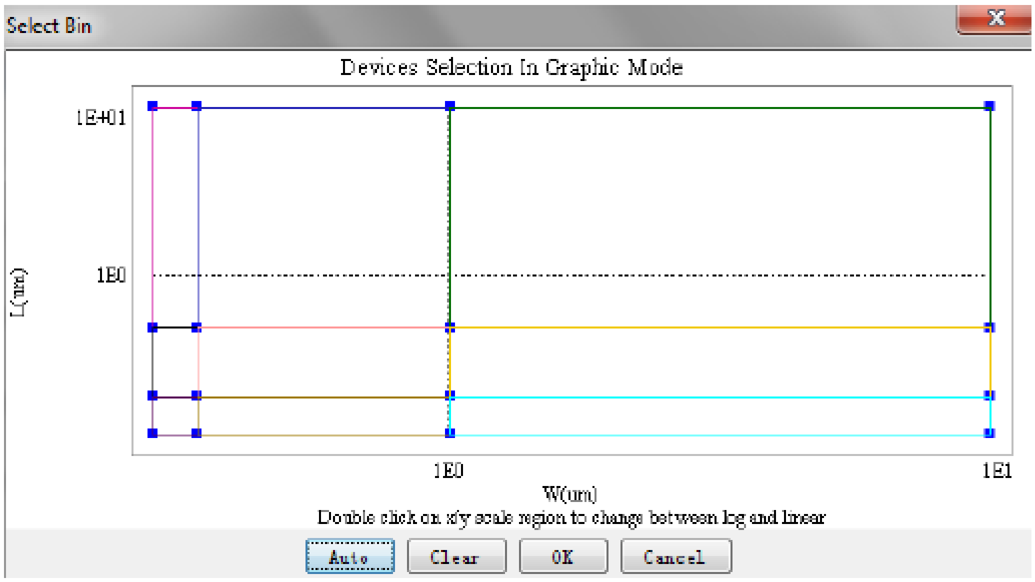
First we introduce how to generate binned models from point models. After all the point models are ready, you can generate the corresponding binned model. To begin, choose Utilities > Binning from the main menu, and then click the Load Point Model button to load the point models. As shown in the figure Load point model a total of 16 point models are loaded.

Load point model



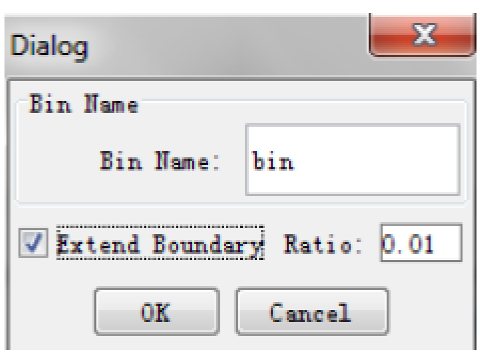
Binning region selection is done in a geometry plane. Click Select Bin and the graphical bin selection window will pop up. Press and hold Ctrl, then use the cursor to select the bin region. You can also click Auto to generate the bins automatically. In the example, the 16 devices are divided into nine bins, as shown in the following figure:

Select bin



After the bin selection is done, click Generate. The dialog window shown in the figure below will pop up where you must type in the bin model name. Then, you have an option to expand the bin boundary by checking the Extend Boundary box and inputting the ratio value. For example, after checking Extend Boundary and setting the ratio at 0.01, LMax will become 1.01E-5 from 1E-5. WMax also becomes 1.01E-5 from 1E-5.

Generate binned model

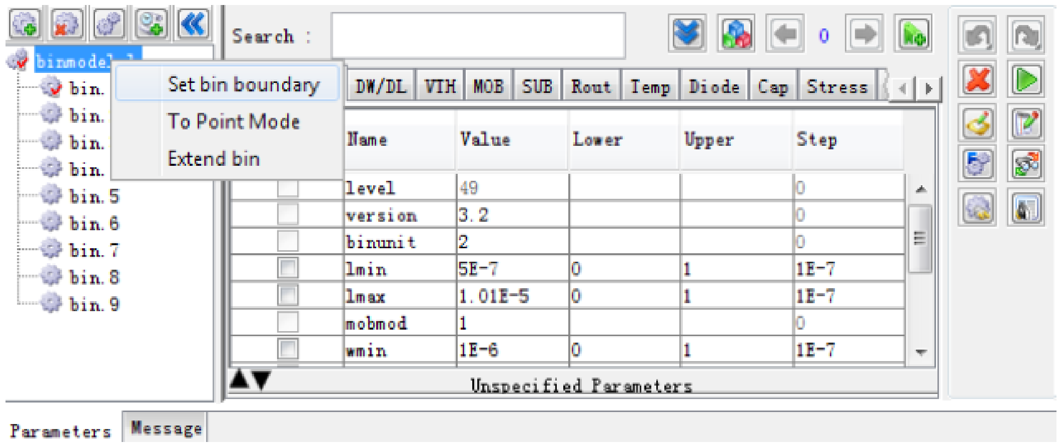


Press the Save button to store the generated binned model.

Three Options Regarding Binned Model

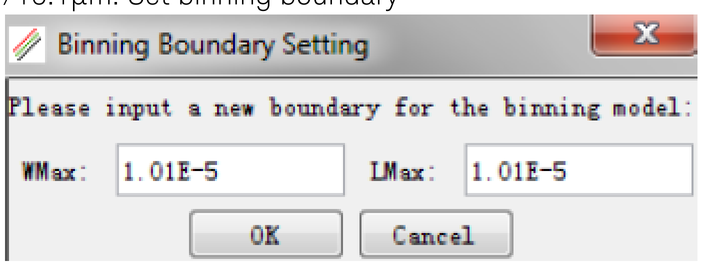
Load the binned model just created. Right-click to open a new window as shown in the following figure:

Three binned model options

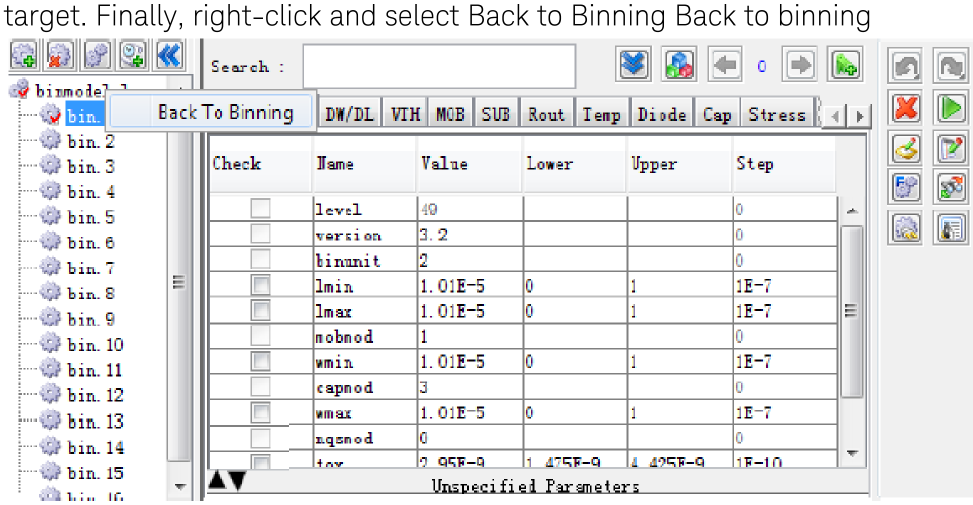


There are three options:

1. Set bin boundary
   For a binned model, the user often extends the maximum channel width and length a little to verify that the simulation of the WMax/LMax device is correct. For example, given the setting shown below in the figure, the model is extracted from WMax/ LMax=10µm/10µm device. However, in the final binned model, the user will obtain a bin boundary of WMax/LMax =10.1µm /10.1µm. Set binning boundary

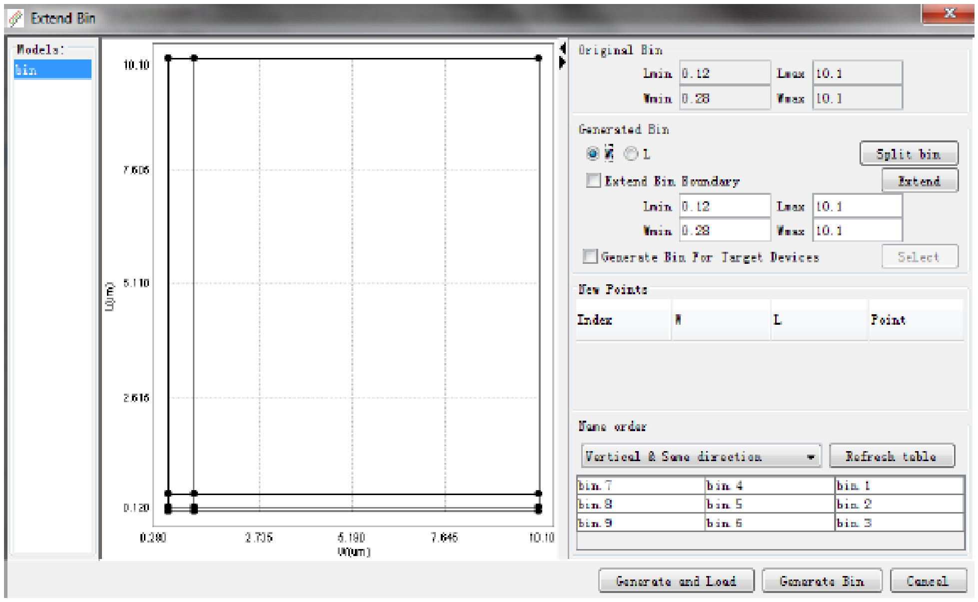


2. To point mode
   Convert the loaded bin models to point models. The newly generated point models are to the right of the original bin boundary points. After this operation, you can tune the bin boundary point models directly to meet the target. Finally, right-click and select Back to Binning Back to binning



3. Extend bin

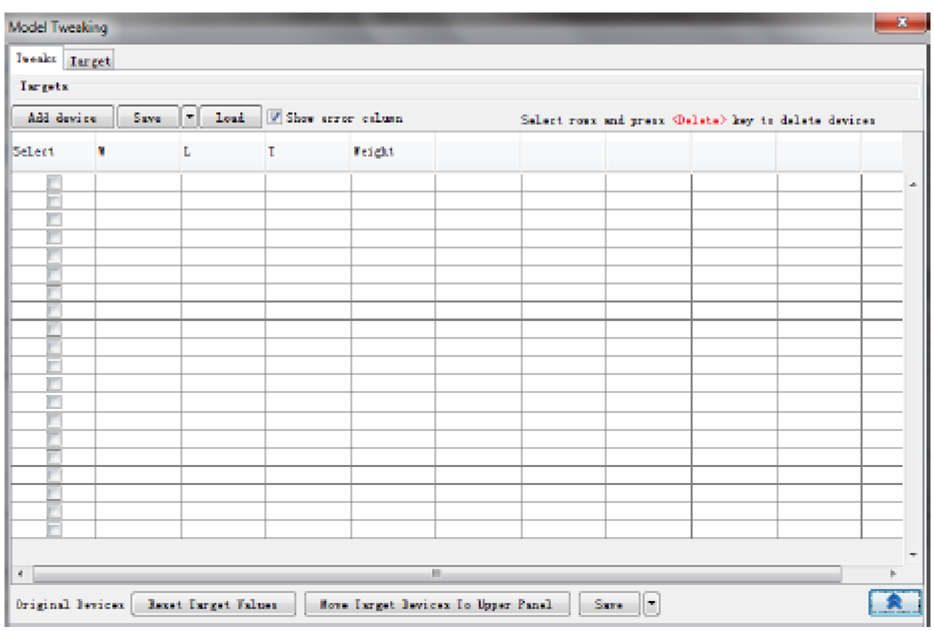
   In the Extend Bindialog window, shown in the figure extended bin, you can re-scale the bin region or generate a binned model that incorporates all of the target devices, including the new insertion points, bin boundary point devices. Extend bin



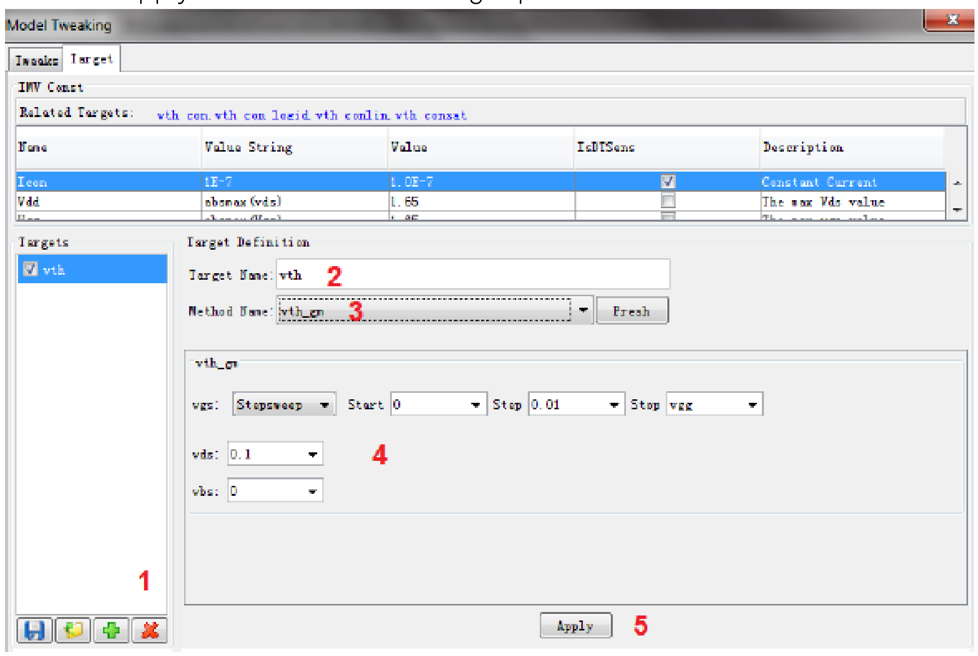
Tweak Binned Models

Besides the method of re-extracting the point models and re-generating the binned model, MBP allows you to directly tweak binned models. With this method, you can tweak binned models much like they would tune global models.To implement this model in MBP choose Extraction > Model Tweaking. The window shown in the following figure is displayed.

Model tweaking window



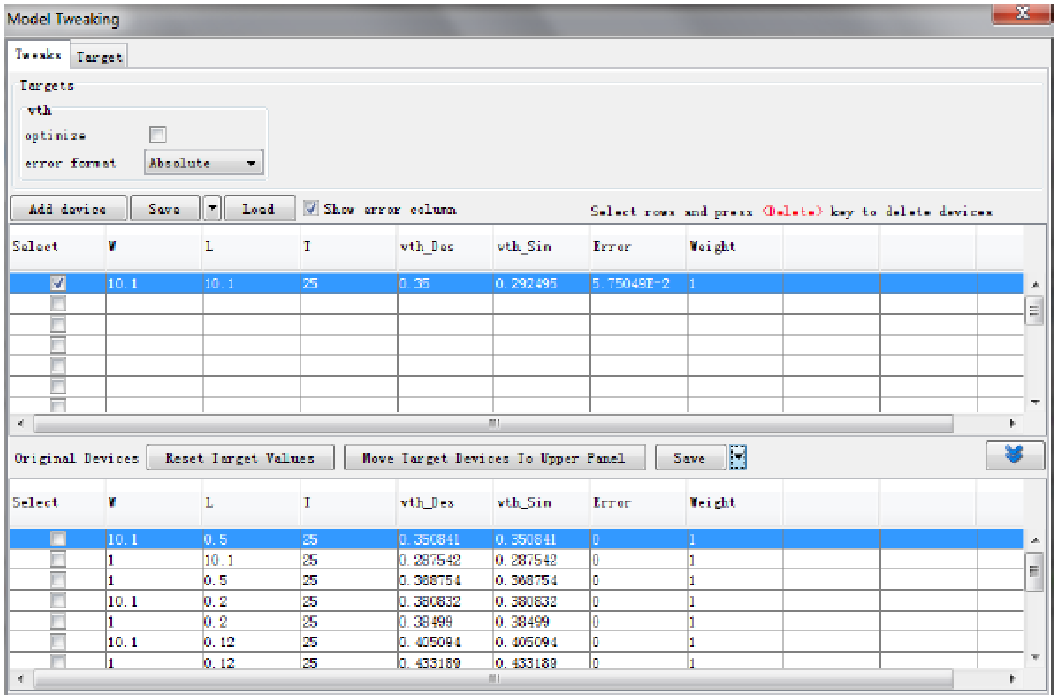
1. Click the Target tab in the window, as shown in figure target panel. Then:
2. Click  to add a tweaking target.
3. Assign a name to this target, vth for example.
4. Choose one built-in algorithm (to achieve the above target) from the drop-down list. Here vth_gm means to calculate the threshold voltage with the maximum transconductance method.
5. Input the bias conditions to perform the algorithm.
6. Click the Applybutton to confirm. Target panel



Next, you can return to the Tweaks tab. Click the  icon to expand the hidden Original Devices panel. All of the original devices used to generate the binned model are listed in the panel. Choose the desired target devices by checking the box, and then click the Move Target Devices to Upper Panel button. You can also add new target devices by clicking the Add device button.

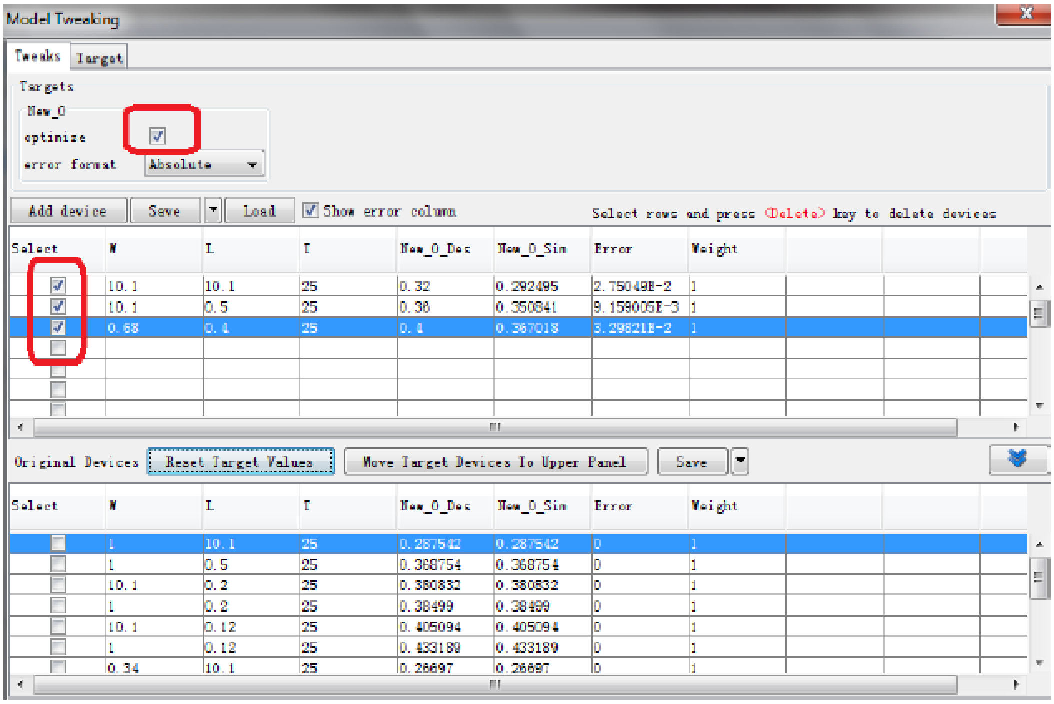
Edit the values of instance parameters (W, L, T, etc.) and set the design targets (e.g., vth_Des), as shown in the following figure:

Binned model tweaking



Next, determine the bins to be tweaked. Obtain the geometry of the target devices (the devices selected to be optimized) and check which bins contain these devices. These bins are the ones that will be tweaked later. According to the target values and the bins to be tweaked, you can select the parameters with which to tweak the binned model. In the process of selecting tweaking parameters, you can add, remove and parametrize the non-binable parameters (e.g., tox), bin-core parameters (e.g., vth0) and L/W/P parameters (e.g., lvth0, pvth0 and pvth0).

Automated optimization



After selecting the parameters, checking the optimize box and the Select items for the target devices, you can perform automated optimization. Click the optimize button  in the Optimization window and MBP will invoke the internal optimizer to proceed.

After optimization, you can check if the tweaking binned model meets their requirement. If Yes you can save the newly generated model directly. If No then, you can either manually tune the parameters or change the conditions and re-run the optimization.

Call External Simulator

This application note describes how to call an external simulator for simulation in Model Builder Program (MBP).

NOTE This document was originally released for MBP V2010.3.0.1 in May 2011.

Overview

MBP supports the ability to call external simulators, regardless of whether they are installed on a local computer or a remote host. From MBP v2007.2.6 on, all the configurations for calling external simulators can be set on the software's graphical user interface (GUI).

NOTE To avoid a long simulation time, it is recommended that the internal SPICE in MBP be used for simulation.

This section uses HSPICE as the example to introduce the steps required to call HSPICE in MBP. You can follow these steps to call HSPICE on a local computer or a remote host. For more information, go to www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at www.keysight.com/find/contactus

Call Local HSPICE

To call HSPICE on a local computer, first open a cmd window. Type *HSPICE* in the command line to confirm that HSPICE can be run directly on the local computer. If HSPICE is properly installed, but cannot run directly, right-click My Computer and open System Properties > Advanced > Environment Variables. Add `$HSPICE_HOME\bin` to the variable Path . Here, `$HSPICE_HOME` is the HSPICE installation path on the computer.

In the MBP main GUI, choose Simulation > Simulator > External Hspice from the main menu to select the simulator.

When doing the simulation, the HSPICE window will flash in the background.

NOTE You may not see the HSPICE window when calling HSPICE on a remote host.

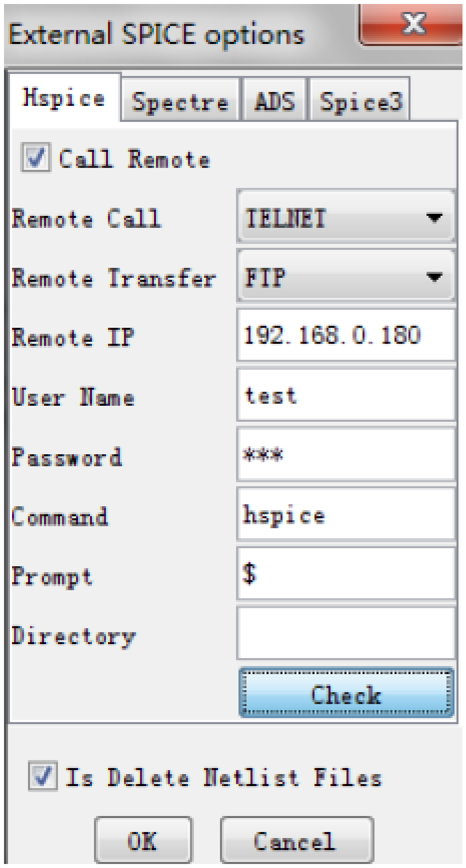
If the simulation results are not correct, choose Simulation > External SPICE from the main menu and deselect the Delete Netlist Files option in the popup window. Redo the simulation to see if the results are what is expected.

NOTE All netlist files can be found in the `$MBP_HOME\hspice` folder (`$MBP_HOME` is the MBP installation path on the computer). You can use these files when debugging.

Call Remote HSPICE

When calling HSPICE on a remote host, you must first confirm they have permission to do so. Telnet can be used to log in to the remote service to confirm the permission. In the MBP main GUI, choose Simulation > Simulator > External Hspice from the main menu to select the simulator. Then, choose Simulation > External SPICE from the main menu. The window will pop up as shown in the following figure.

External SPICE options

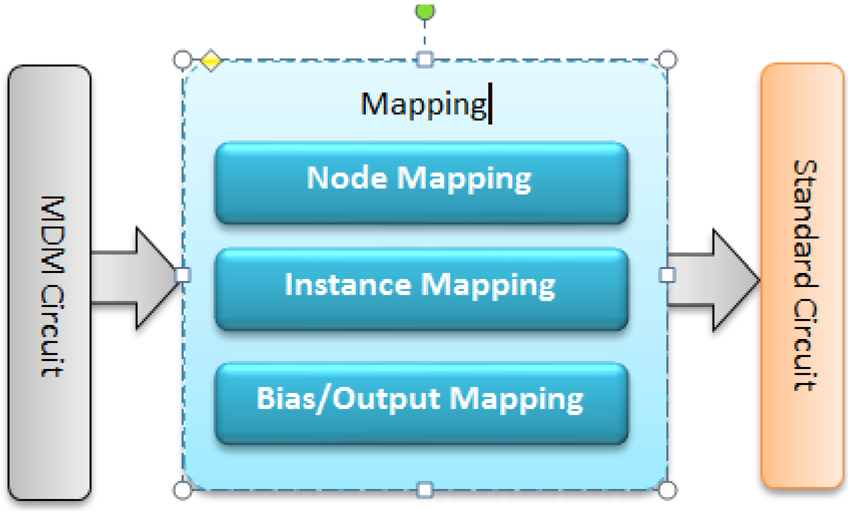


Next, enable the Call Remote option by clicking the box. Enter the necessary information such as Remote IP, User Name and Password. After filling out the settings, you can check whether HSPICE can be called successfully by clicking Check . Finally, click OK to close the window.

DC/CV MDM Loader Configuration

Overview

MDM Loader module is designed for MBP to load ICCAP's .MDM file, by default we have completed the configuration for most conditions so that user can load a MDM file without any modification to the loader, but if user want to load any special MDM files the customization of the LOADER need to be done by the user himself.
This material instruct on how to do the customization
The overall structure of MDM Loader:



By configure the MDM loader user configure the Mapping to change the circuit in MDM file to a MBP standard circuit, then MBP can parser the file and simulation correctly.

Circuit Standard

Bias/Output Name Standard

Bias/Output Name is composed with type, node1 and node2.

Type

Type can be one of characters below:

Name	Type
V	Voltage
I	Current
C	Capacitance
G	Conductance

Node

If node name contains '_', it is a composed node, for example:"d_s_b", this node is connected to 'd','s' and 'b'.

Bias/Output Name Format

Format 1
`<Type>_<Node1>#<Node2>`
The first character is type, and followed by by node1, '#' and node2.

Example

Vd_s_b#g	Voltage between d_s_b and g
Vg#d_s_b	Voltage between g and d_s_b

Format 2

`<Type><Node1><Node2>`

If each node contains no more than 1 character, '#' should be omitted.

Example

Vgs	Voltage between g and s
Vbs	Voltage between b and s

Format 3

`<Type>_<Node1>#`

or

`<Type>_<Node1>`

If the node2 is "ground", it should be omitted.

Example

Vd_s_b#	Voltage between d_s_b and ground
Vg	Voltage between g and ground
Vb	Voltage between b and ground

Analysis Name Standard

Format

`<Output><Sweep1><Sweep2>@<Bias1>,<Bias2>`

1. Analysis name is composed with output and bias names.
2. If there is '_' in output/bias name, it will be replaced with '~'

Example

Analysis Name	Output	Biases
Ids_vgs_vbs@vds,vs	Ids	Vgs,vbs,vds,vs
Cgc_vb_vdg_vds@vg	Cgc	Vb,vdg,vds,vg
Cg#d~s_vb_vdg_vds@vg	Cg#d_s	Vb,vdg,vds,vg

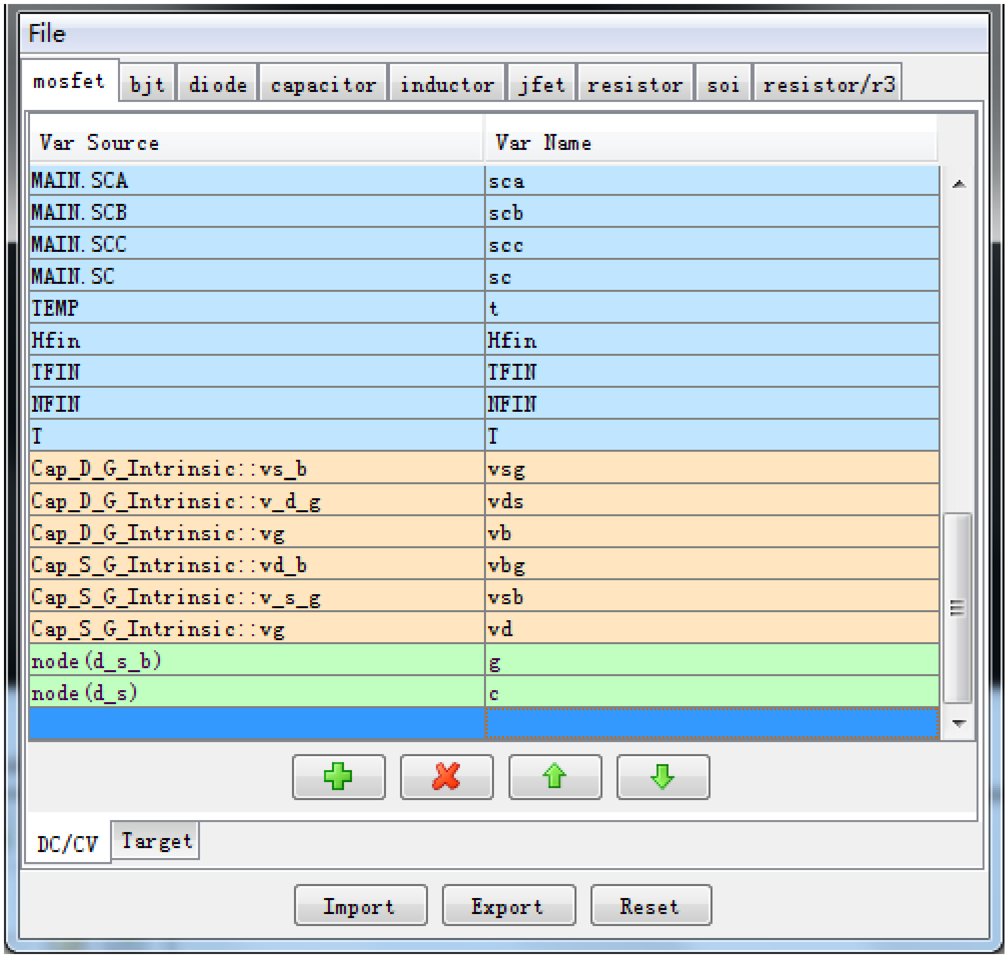
List of Standard Node Names and Instance Names

Model	Nodes	Instances
/bjt/gp	c,b,e,s	area,areab,areac,temp,l,w
/bjt/hicum	c,b,e,s	area,temp,l,w
/bjt/mextram	c,b,e,s	area,temp
/bjt/vbic	c,b,e,s	area,m,temp,l,w
/capacitor /mimcap	p,n	w,l,temp
/capacitor /mosvar	g,bi,b	w,l,temp,m,M_SEG,NGCON
/diode/diode	p,n	area,pj,temp,l,w,lm,wm,lp,wp,w
/diode/diode3	p,n	area,pj,temp,l,w,lm,wm,lp,wp,w
/diode/juncap2	p,n	area,pj,pgate,temp
/jfet/jfet	d,g,s	w,l,temp
/mosfet/mos2	d,g,s,b	w,l,temp,nf,m,sa,sb,sd,sca,scb,scc,sc,ad,pd,as,ps,nrs,nrd
/mosfet/mos3	d,g,s,b	
/mosfet/mos66	d,g,s,b	
/mosfet /bsim3v3	d,g,s,b	
/mosfet/bsim4	d,g,s,b	w,l,temp,nf,m,sa,sb,sd,sca,scb,scc,sc,ad,pd,as,ps,nrs,nrd,rsc,rdc
/mosfet/bsim6	d,g,s,b	l,w,nf,temp,nrs,nrd,m,rgatemod,rbodymod,geomod,rgeomod,sa, sb,sd,sca,scb,scc,sc
/mosfet /bsimcmg	d,g,s,b	l,d,tfin,fpitch,nf,nfin,ngcon,temp,nrs,nrd,lrsd
/mosfet /bsimimg	d,g,s,b	l,w,nf,temp,nrs,nrd

Model	Nodes	Instances
/mosfet/hisim2	d,g,s,b	w,l,temp,nf,m,sa,sb,sd,ad,pd,as,ps,nrs,nrd,sca,scb,scc
/mosfet /hisim_hv	d,g,s,b	w,l,temp,nf,m,sa,sb,sd,ad,pd,as,ps,nrs,nrd
/mosfet /LayoutConfig	d,g,s,b	w,l,temp,nf,m,sa,sb,sd,sca,scb,scc,sc,ad,pd,as,ps
/mosfet/psp/102 /local	d,g,s,b	
/mosfet/psp/102 /global	d,g,s,b	
/mosfet/psp/103 /local	d,g,s,b	
/mosfet/psp/103 /global	d,g,s,b	w,l,temp,nf,m,sa,sb,sd,sca,scb,scc,sc,ad,pd,as,ps,nrs,nrd
/mosfet /RingOscillator	avin,roa,avdd,qvin, roq,qvdd,s	wp,lp,wn,ln,outtype
/resistor/poly-resistor	p,n	w,l,temp
/resistor/r3	n1,nc,n2	m,l,w,temp,swnoise,sw_et,wd,a1,p1,a2,p2,c1,c2,sw_mman, nsmm_rsh,nsmm_w,nsmm_l
/resistor/resistor	p,n	w,l,temp
/soi/b3soi	d,g,s,e,p	w,l,temp,nrd,nrs,nf,m,sa,sb,sd,ad,pd,as,ps
/soi/b4soi	d,g,s,e,p	w,l,temp,nrd,nrs,nf,m,ad,as,pd,ps,sa,sb,sd
/tmi/bsim4 /default	d,g,s,b	w,l,temp,nf,m,sa,sb,sd,sca,scb,scc,sc,ad,pd,as,ps,nrs,nrd

Mapping

Click File -> Data -> Data Loader -> Config on MBP to start the configuration GUI.



There are 3 kinds of Mapping:

1. Instance Mapping(Change the instance name to standard name)

	Var Source	Var Name
Describe	Name in mdm file	Standard Name
Sample	MAIN.SCA	SCA

2. Node Mapping(Change the node to standard name)

	Var Source	Var Name
Describe	node(nodeName)	Standard Name
Sample	node(d_s)	c

This mapping is only for current, capacitor and conductance:
For example:cg#d_s -> cgc
For voltage source, it doesn't care this mapping:
For example:vg#d_s -> vgd, vds=0

3. Bias/Output Mapping(Change the bias/output to standard name for specified setup)

	Var Source	**Var Name**
Describe	SetupName::VarName	Standard Name
Sample	Cap_S_G_Intrinisic::vg	vgs

Generate and Tweak Corner Model

This application note describes how to generate and tweak corner models in Model Builder Program (MBP).

NOTE This document was originally released for MBP V2011.1.0 in July 2011.

Overview

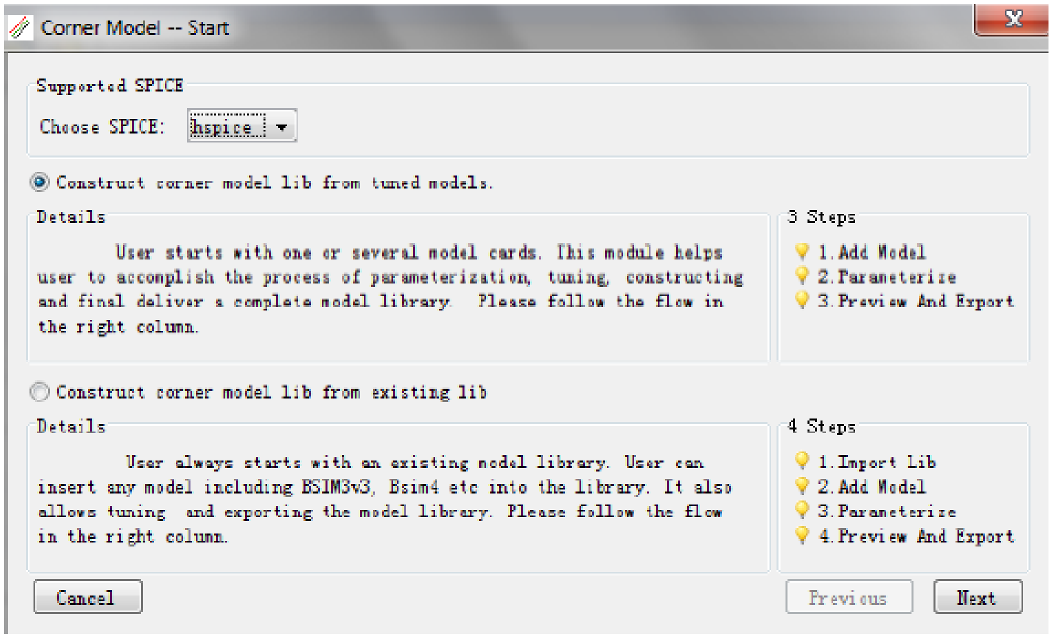
Device model is relevant to the actual fabrication process. Even a well developed process may have some variations. And these variations are likely to affect the device characteristics and circuit behavior. To account for the variations in semiconductor process, based on an initial set of typical and comer device models, the process dependent model parameters needed to be tweaked. The smart model tweaking module integrated with MBP enables easy model retargeting, adjustment of global or binning models according to new specification. Both model cards and model libraries are well supported.

In this article, we will use global model as an example to introduce the steps to generate and tweak corner models, respectively.

Generate Corner Models

To generate corner models, choose Utilities > Corner Model from the main menu. The window shown in the following figure:

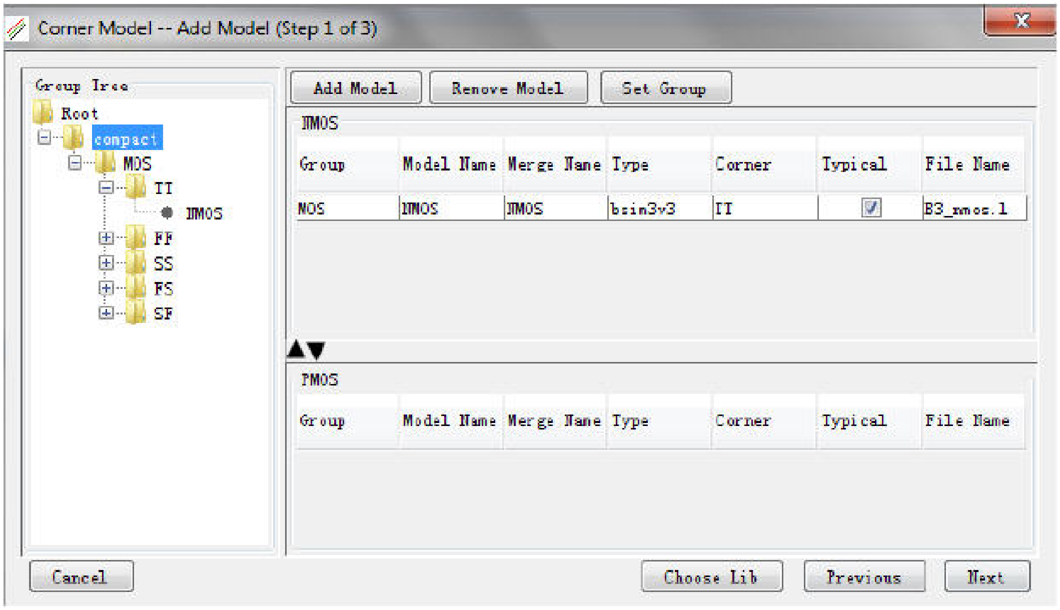
Corner model generation--Start



MBP currently supports two external simulators: HSPICE and Spectre. Choose either one as the SPICE simulator. You have an option to construct the corner model library from either tuned models or an existing library. In this document, we introduce the procedures to Construct corner model lib from tuned modes. The process of Construct corner model lib from existing lib is almost the same, except that it requires the additional step of importing a library.

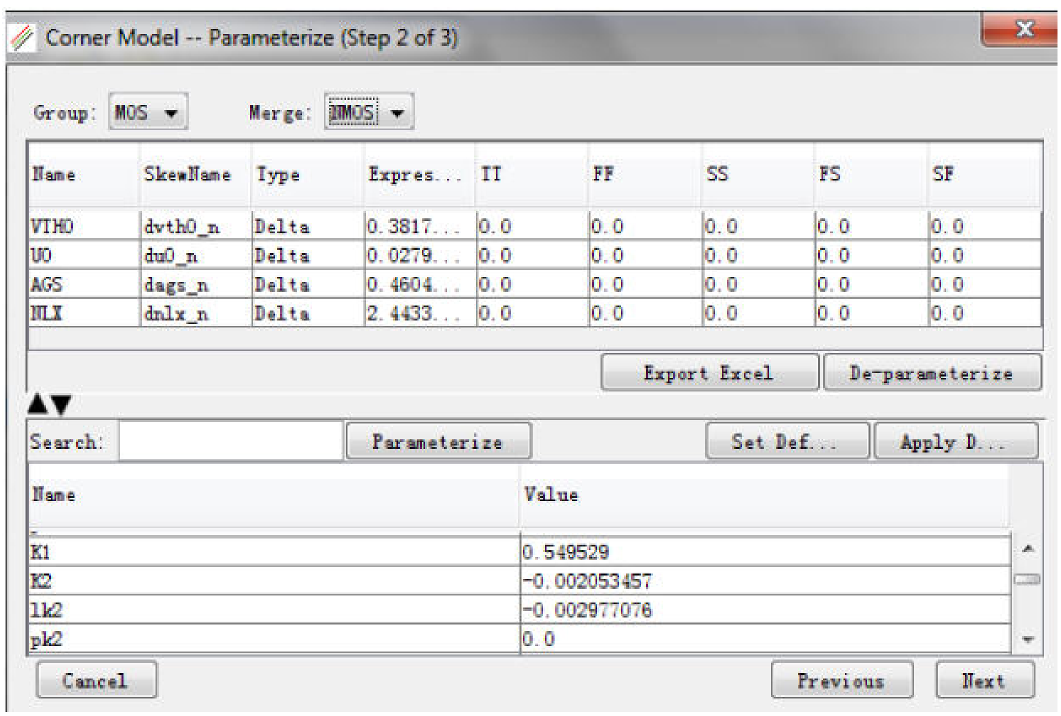
- Click Next.
- Select Add Model to load the model card.
- Click Add default in the popup Set Group window.
- Select OK.
- Check the Typical box and click Next , as shown in following figure.

Corner model generation-Step 1



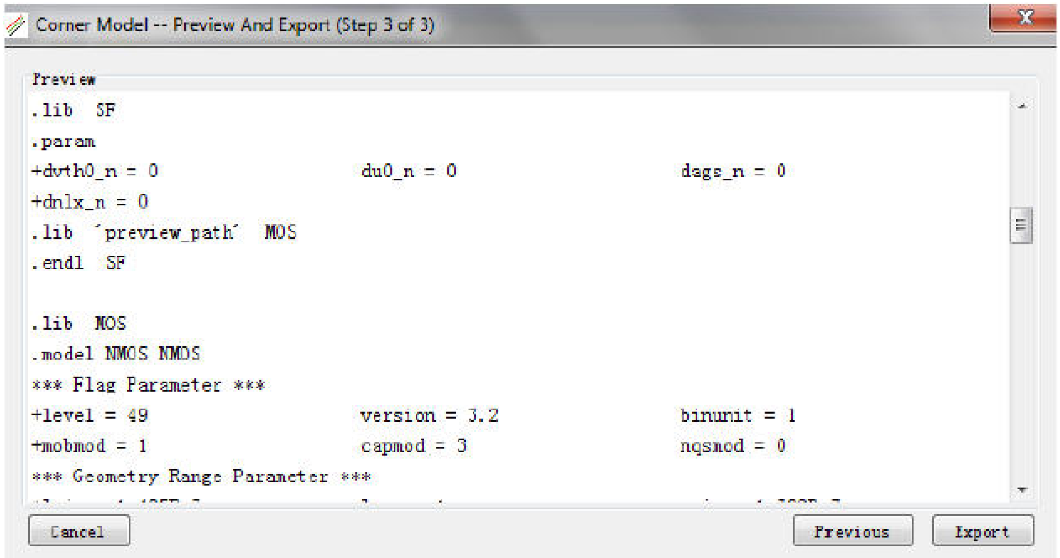
In the popup window shown in following figure, choose the parameters (e.g., vth0, u0 and ags) and click Next.

Corner model generation-Step 2



The corresponding corner model structure will now be built, as shown in following figure. Click Export to save the library.

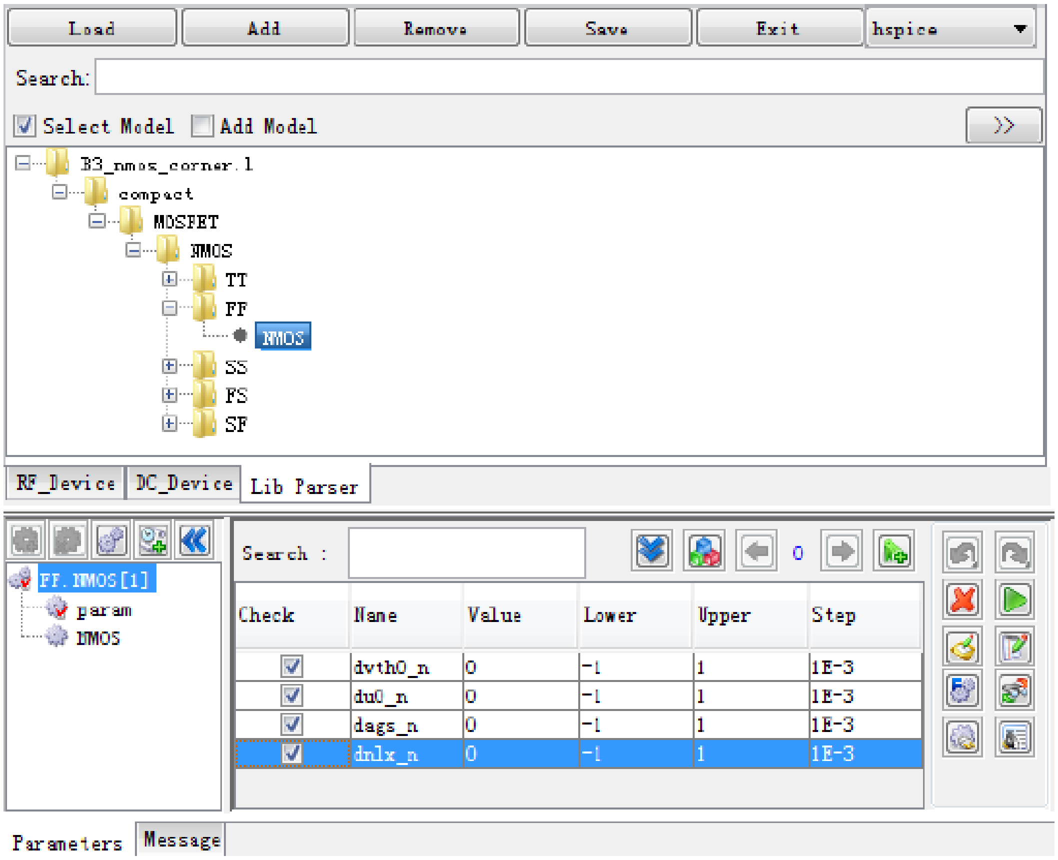
Corner model generation-Step 3



Tweaking Corner Models

Choose Utilities > Lib Parser from the main menu. A warning message window will pop up. Click Yes to continue. In the Lib Parser tab, click Load to load the library just created. The tree structure of the library and the model parameters to be tweaked are listed as shown in following figure.

Corner library and model parameters

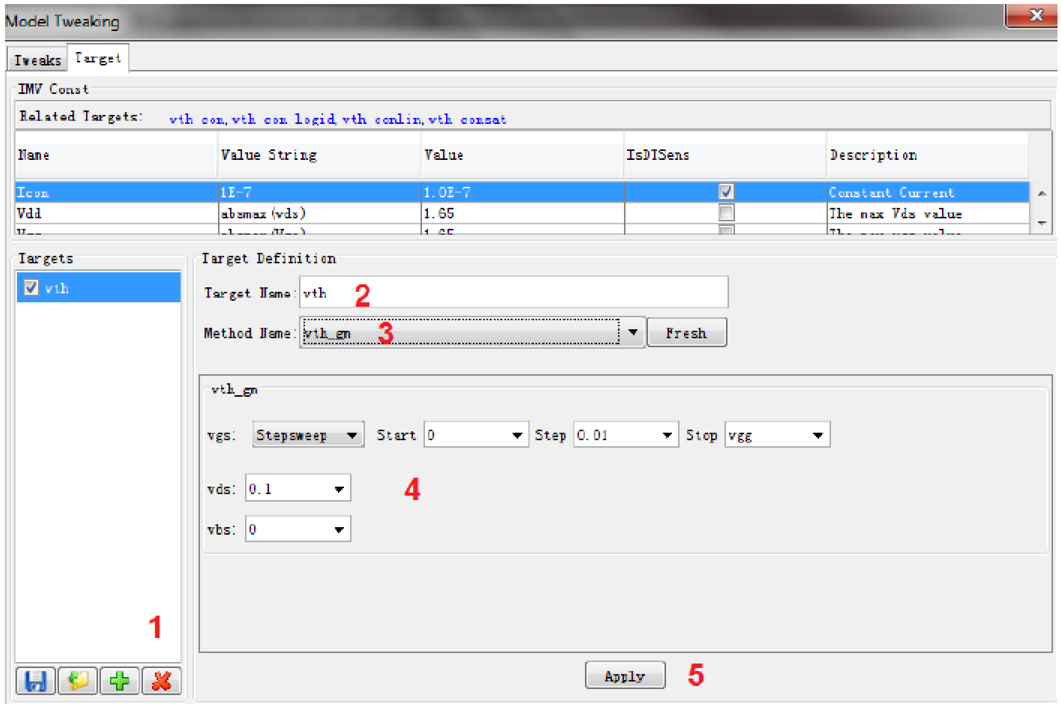


Define Tweaking Target

To define the tweaking target choose Extraction > Model Tweaking from the main menu and switch to the Target panel. The window shown in Figure: Target panel will appear. In this window:

1. Click  to add a tweaking target.
2. Assign a name to this target, for example vth.
3. Choose one built-in algorithm (to achieve the above target) from the drop-down list. Here, vth_gm means to calculate the threshold voltage with the maximum transconductance method.
4. Input the bias conditions to perform the algorithm.
5. Click the Apply button to confirm.
6. Repeat steps 1 to 5 to add Idsat as another tweaking target.

Target panel

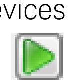


Define Target Devices

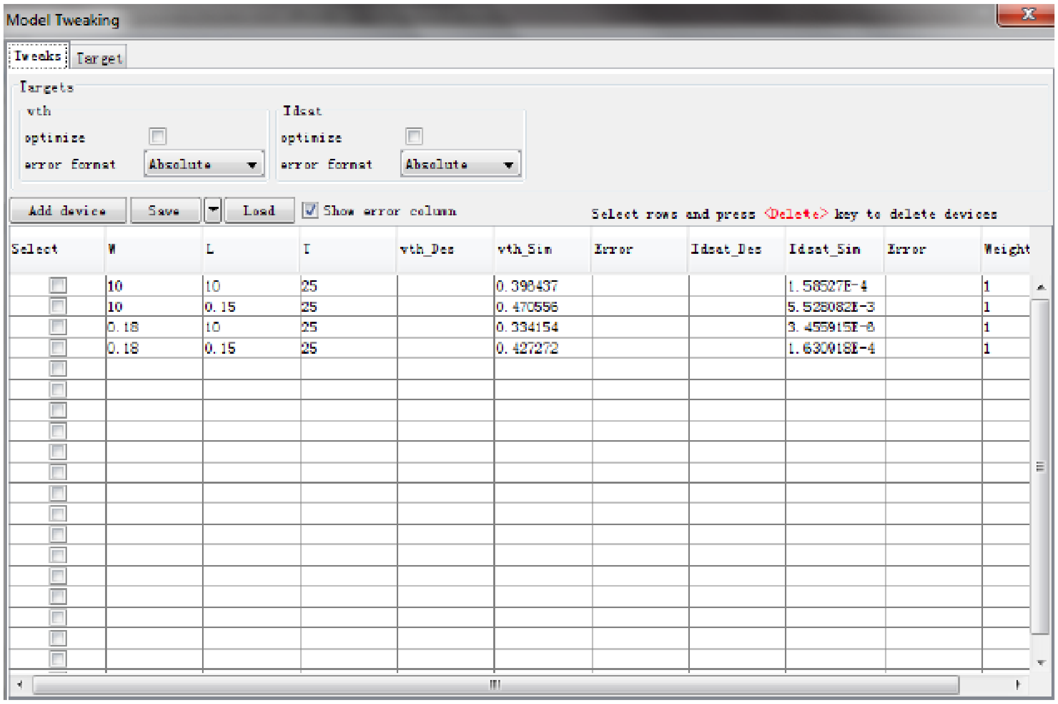
After defining the tweaking targets, switch to the Tweaks panel. Press the Add device button to add target devices to the list. You can edit the instances such as W , L and T, in this case. Right-clicking on the table will pop up the instance parameter list. MBP allows you to save or load device information, as shown in Figure: Tweaks panel.

Following the instances, the design target (suffixed by Des, e.g., vth_Des), simulated value (suffixed by Sim, e.g., vth_Sim), and their differences (e.g., Error) are listed. You can choose the error format as either Absolute or Relative. Type in the value of the design target. The simulated value and Error will respond instantly once you tune the model parameters.

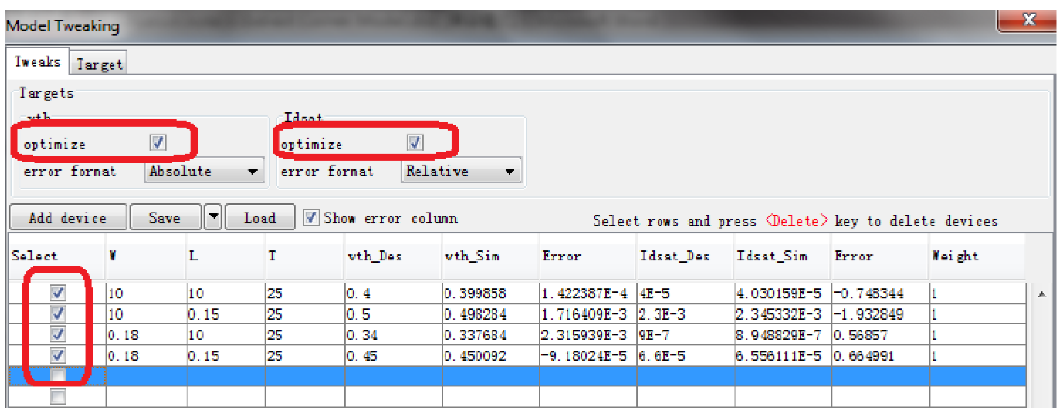
Model Tweaking

Select the proper parameters from the Optimization/Parameter_ window. You can then start tweaking the model. MBP provides both manual and automated tuning. For the latter, make sure that the design targets are set, parameters are selected, the optimize items for tweaking target are checked, and the Select items for the target devices are checked (Figure: Automated optimization). Then, click the Optimize  button in the Optimization window. MBP will invoke its internal optimizer to proceed.

Tweaks panel



Automated optimization



If more than one target device is selected for automated tuning, you can differentiate the importance between devices by specifying different Weight values. The device with the higher weight value normally generates higher accuracy.

Implementing Verilog-A Model

This application note describes how to implement Verilog-A models in Model Builder Program (MBP).

NOTE This document was originally released for MBP V2009.1.0 in July 2011.

Overview

Verilog-A is an industry standard modeling language for analog circuits. MBP initiated support of Verilog-A models with MBP v2009.1.0.

This application note describes how to implement Verilog-A models in MBP. For more information, go to www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at: www.keysight.com/find/contactus

Preparation

To implement Verilog-A models, you must first ensure that MBP v2009.1.0 or a later version has been properly installed on the computer. Also, the Verilog-A license feature is needed.
Windows users must add `$MBP_HOME\win32\mingw\bin` to the environment variable Path. Here, `$MBP_HOME` stands for the directory where MBP is installed, for example `C:\Keysight\modelbuilder`. Then, reboot the computer.

For Linux users, run which gcc and which g++ in the command line to make sure both gcc and g++ have been installed properly on the machine. Otherwise, contact your IT administrator.

Sample Models

There must be a subcircuit model to define which Verilog-A model is to be called and which parameters are to be tweaked. MBP allows you to load this subcircuit model and tweak the parameters in the same way as any other model parameters in MBP.

Sample models are listed below for the HSPICE and SPECTRE simulators.

HSPICE

The following model, ekv.l, is an example of a model that is simulated by HSPICE:

```
hdl ekv.va // Define Verilog-A model to use: ekv.va.
model verilog1 ekv // Define new model named verilog1.
Use Verilog-A odel ekmv from ekv.va.
+VTO=0.5 // Define model parameters to be tweaked in
MBP.
+GAMMA=1
+PHI=0.5
.subckt rf_nch d g s b W=10E-6 L=10E-6
x3 d g s b verilog1 L=L W=W // The user could use the
new model named verilog1,
x2 d g s b ekv L=5E-6 W=10E-6 // or use the original
model named ekv.
.ends
```

SPECTRE

The following model, ekv.l, is an example of a model that is simulated by SPECTRE:

```
simulator lang=spectre
ahdl_include ekv.va // Define Verilog-A model to use:
ekv.va.
model verilog1 ekv // Define new model named verilog1.
Use Verilog-A model ekv from ekv.va.
+VTO=0.5 // Define model parameters to be tweaked
in BP. M
+GAMMA=1
+PHI=0.5
subckt rf_nch (d g s b)
parameters W=10E-6 L=10E-6
x3 (d g s b) verilog1 l=l w=w // The user could use the
new model named *verilog1*,
x2 (d g s b) ekv l=5e-6 w=10e-6 // or use the original
model named *ekv*.
ends rf_nch
```

NOTE The element using Verilog-A must start with **x** even in SPECTRE. Only the parameters declared in **verilog1** , such as VTO, GAMMA and PHI, can be tweaked in MBP. The original Verilog-A model (e.g., ekv) can be called and simulated in the subcircuit. However, the parameters in ekv not declared in verilog1 cannot be tweaked in MBP.

MBP Supported Functions and Keywords

MBP supports most of the common functions and keywords defined in Verilog-A, including:

- Basic operation: supports most of the basic operation in Verilog-A.
- Syntax: supports if/else, for loop, while loop, etc. Does not support repeat.
- Simulation system function: supports $stop, $temperature, $vt, $vt(temp), and strobe(express).
- Function: supports user-defined function.

For additional details, refer to the following table:

Support Table

Category	Type	Item	Example	Support Status	Description
Basic Operator	Mathematic	/		Supported	Attention: res = 1/5; //The resulte of this integer division is zero,res = 0.

Category	Type	Item	Example	Support Status	Description
Basic Operator	Mathematic	+		Supported	
Basic Operator	Mathematic	-		Supported	
Basic Operator	Mathematic	*		Supported	
Basic Operator	Mathematic	sqrt	sqrt(x)	Supported	
Basic Operator	Mathematic	ln	ln(x)	Supported	
Basic Operator	Mathematic	log	log(x)	Supported	
Basic Operator	Mathematic	abs	abs(x)	Supported	
Basic Operator	Mathematic	pow	pow(x,y)	Supported	
Basic Operator	Mathematic	min	min(x,y)	Supported	
Basic Operator	Mathematic	max	max(x,y)	Supported	
Basic Operator	Relational Operators	<	a>b	Supported	
Basic Operator	Relational Operators	>	a<b	Supported	
Basic Operator	Relational Operators	<=	a<=b	Supported	
Basic Operator	Relational Operators	>=	a>=b	Supported	
Basic Operator	Logical operators	!=		Supported	
Basic Operator	Logical operators	==		Supported	
Basic Operator	Logical operators	&&		Supported	
Basic Operator	Logical operators	||		Supported	
Basic Operator	Conditional Operator	?:	(a<b)?a:b	Supported	

Category	Type	Item	Example	Support Status	Description
Basic Operator	access	I( )	I(branch) I(node1,n ode2) I(node1)	Supported	Attention : I(node1) means the current from the node1 to the ground
Basic Operator	access	V( )	V(node1,n ode2) V(node1)	Supported	
Basic Operator	contribution	I(a,b)<+V(c,d)		Supported	
Basic Operator	contribution	V(c,d)<+variable or constant	V(in,mid)<+0.5;V(in, mid)<+x;	Supported	
Basic Operator	contribution	I(c,d)<+variable or constant		Supported	
Basic Operator	contribution	V(c,d)<+I(a,b)		Supported	
Basic Operator	contribution	I(a,b)<+I(c,d)		Not Supported	
Basic Operator	ddx	Y=ddx(z,x);		Supported	
Basic Operator	ddx	Y=ddx(z,x);		Supported	Attention : Y= k*ddx(z,x); is
Basic Operator	ddx	Y=k* Y;			unacceptable
Basic Operator	ddx	Y= ddx(func(x), x);	Fun(x) is the function of x. And fun is maybe	Supported	
Basic Operator	ddx	Y=ddx(Y,x);	Y is a var	Not Supported	Attention : But z=ddx(Y , x); Y=z; that is right

Category	Type	Item	Example	Support Status	Description
Basic Operator	ddx	ddx (Userdefined function,x)		Supported	
Basic Operator	ddx	I()<+ddx(y,x)		Not Supported	Attention : But ,z=ddx(y, x) I()<+z Is right
Basic Operator	ddx	More than 2th derivate	ddx(ddx(a,b),b)	Not Supported	
Basic Operator	idt			Not Supported	
Basic Operator	assignment	y=V(p,n);		Supported	
Basic Operator		z=I(p,n);		Not Supported	
Syntax		If else			
Syntax		Forloop		Supported	
Syntax		case		Supported	
Syntax		whileloop		Supported	
Syntax		repeat		Not Supported	
Syntax	Defining Macros	`define		Supported	
Syntax	Conditional Compilation	`ifdef `else `endif		Supported	
syntax	including	`include	`include "discipline s.vams"	Supported	
	Simulation control	$stop		Supported	

Category	Type	Item	Example	Support Status	Description
Simulation System Function					
Simulation System Function	Simulation control	$finish		Supported	
Simulation System Function	Environment Parameter Functions	$realtime	Current simulation time in seconds.	Not Supported	
Simulation System Function	Environment Parameter Functions	$temperature	Ambient temperature in kelvin.	Supported	
Simulation System Function	Environment Parameter Functions	$vt	Thermal voltage ( ).	Supported	
Simulation System Function	Environment Parameter Functions	$vt(temp)	Thermal voltage at given temperature	Supported	
Simulation System Function	Environment Parameter Functions	$abstime	Returns the simulation time, in seconds	Not Supported	
Simulation System Function	Input/output	$fopen		Not Supported	
Simulation System Function	Input/output	$fclose		Not Supported	
Simulation System Function	Input/output	$fwrite		Not Supported	
Simulation System Function	Input/output	$strobe (“express”)		Supported	

Category	Type	Item	Example	Support Status	Description
Function	Userdefined Function			Not Supported	
Analog	Initial Step	@(init_s)		Not Supported	
Events		tep( ))			
	Final Step	@(final_step( ))		Not Supported	
	Cross	cross()		Not Supported	
	Timer	timer()		Not Supported	
Hierarchical	Basic_ hierarchical			Not Supported	
	Port_connect			Not Supported	

Reference: Accellera Verilog-AMS Language Reference Manual, Analog & Mixed-Signal Extensions to Verilog HDL, Version 2.3.1

Implementing MOSRA Models

This application note describes how to implement a MOSRA model in Model Builder Program (MBP).

NOTE This document was originally released for MBP V201.1.0 in July 2011.

Overview

Device models are relevant to the actual fabrication process. MBP offers an environment for MOS reliability analysis in general and MOSRA in particular. With this environment, you can measure device performance degradation over time and evaluate stress effects.

This section provides information on the raw data format and MOSRA simulation and parameter extraction. For more information go to www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at www.keysight.com/find/contactus.

Raw Data Format

The data format for MOSRA analysis is similar to that of general measurement data in MBP. A sample of MOSRA data is as follows:

```
condition {corner = tt,date = oct_20_02,instrument=
(hp4145, probe_station),
mode=forward, datatype= mosra, version=1.0, type=nmos}
Page (name=ids_vgs_vbs,x=vgs,p=vbs,y=ids) { vds=0.05, w=
10.0, L=0.13, T=25.0}
stress ( time=0.0, vds=2.0, vgs=1.0, vbs=0.0)
curve { 0.0}
0.0 5.000E-14
0.05 5.002E-14
0.1 5.010E-14
0.15 5.055E-14
0.2 5.315E-14
```

Here, the keyword datatype should be specified as mosra and version=1.0 corresponds to the MOSRA level. The keyword stress defines the bias condition and the duration of the aging test.

MBP also supports another kind of MOSRA data, which allows you to take the aging span as the variable. For example:

```
condition{corner = tt,date = oct_20_02,instrument=
(hp4145, probe_station),mode=forward} Datatype
{S_target} Version{2.1}
type{nmos} Delimiter{,}
Instance{L, W, T}
Strss_Condtion{S_vgs=1, S_vds=2, S_vbs=0, S_time}
Input{Vgg=2, Vdd=2, Vbb=-1,Vdlin=0.05}
Data{ w, l, t, S_vgs, S_vds,S_vbs, S_time, vth_lin,
vthsat, Idlin, Idsat, Ioff, gm }
10, 2, 125, 1, 2, 0, 0, 0.728628, 0.697769, 7.348435E-5,
8.848617E-4, 2.029162E-12,
4.2957E-4
10, 2, 125, 1, 2, 0, 1e5, 0.728985, 0.698127, 7.343337E-
5, 8.8407E-4, 2.028872E-12,
4.2903E-4
```

Here, Datatype{S_target} means the data type is DP data.

NOTE

All of the variables in this kind of MOSRA data need to start with S. For example, the gate-to-source voltage(vgs) during an aging test should be named as **S_vgs** .

All of the data, including instance parameters, bias conditions, timing nodes, and physical quantities, is stored in the Data session.

As shown in figure Pre-defined IMV pages for stress data, several IMV pages have been pre-defined in MBP to help you to better understand stress data trends.

Pre-defined IMV pages for stress data

MOSRA Simulation and Parameter Extraction

MBP invokes the external simulator (Synopsys HSPICE) for MOSRA model simulation.

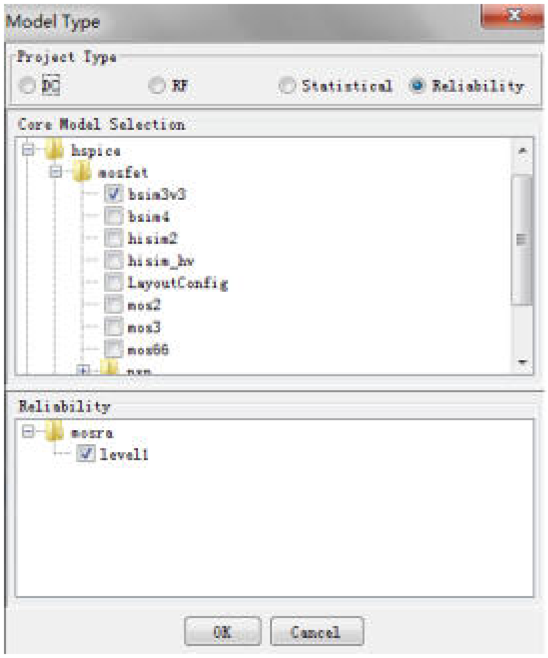
NOTE

Ensure that HSPICE has been installed properly before the simulation.

Choose Model Type

Choose Model > Select Model from the main menu and select Reliability in the pop up window. Then, select one core model in the upper Core Model Selection section and MOSRA in the lower Reliability section.

Selecting MOSRA analysis



Model Parameters Panel

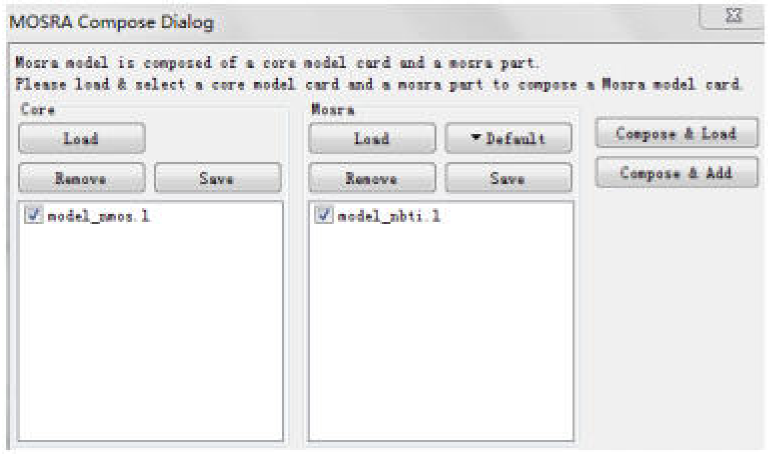
After setting up the model type, MBP merges the selected core model and MOSRA model. For example, as shown in the Parameters panel of following figure, the upper model (mosra model) is a MOSRA level 1 model and the lower one (nmos) is a BSIM3V3 core model. By clicking any model, the corresponding parameters will show on the right-side of the window.

Model viewer for MOSRA

Load Model

In the main menu, choose File > Model > Load to load the model. A window named MOSRA Compose Dialog is displayed.

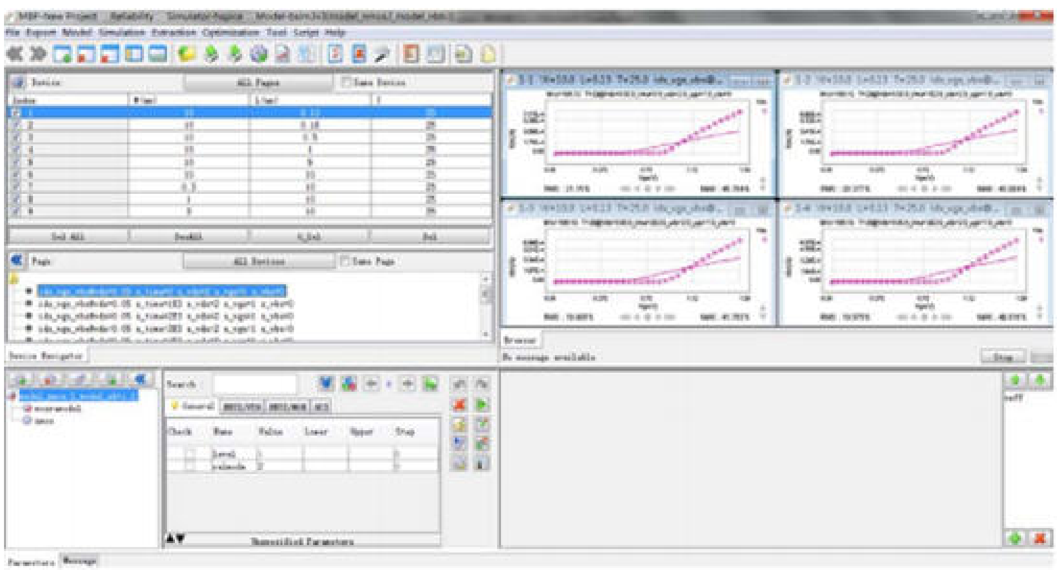
MOSRA compose dialog window



A complete MOSRA model consists of two parts: the core model and the MOSRA model. You can deal with these two parts separately. Click Remove to delete the existing models and click Load to load other models. After loading the models, click Compose & Load to replace the current MOSRA model.

Then, choose File > Data > Load from the main menu to load the data file. The window with the MOSRA model and data.

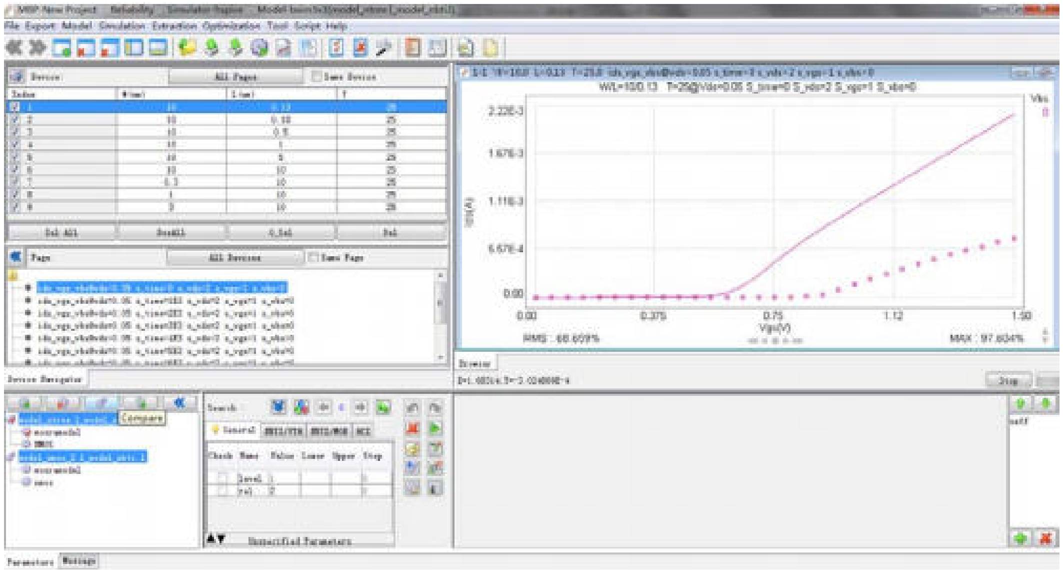
Load data



Now, you can select model parameters and adjust them to fit the measurement data.

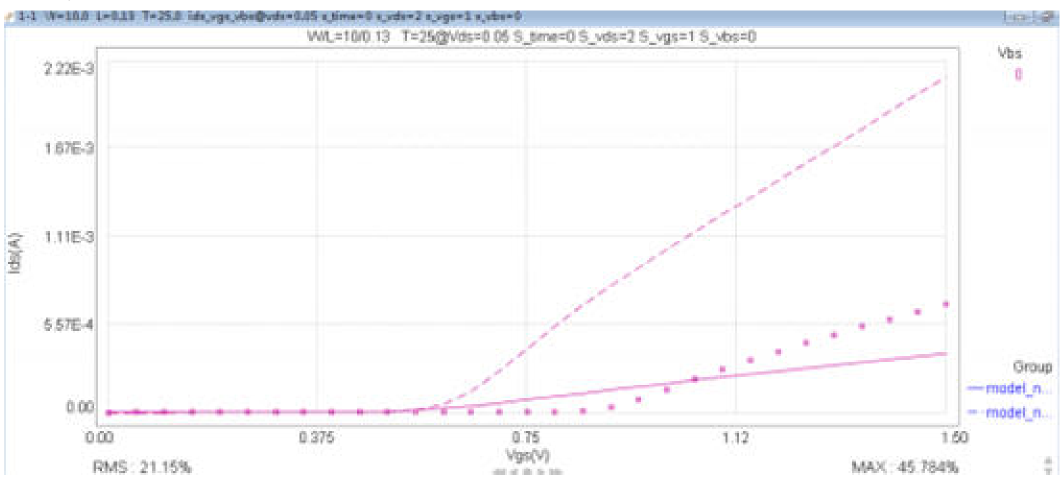
MBP allows you to compare two MOSRA models. Simply, click Compose & Add to append a MOSRA model for comparison. Select the two models and click the Compare icon to compare them.

Compare two models



The result is shown in the following figure.

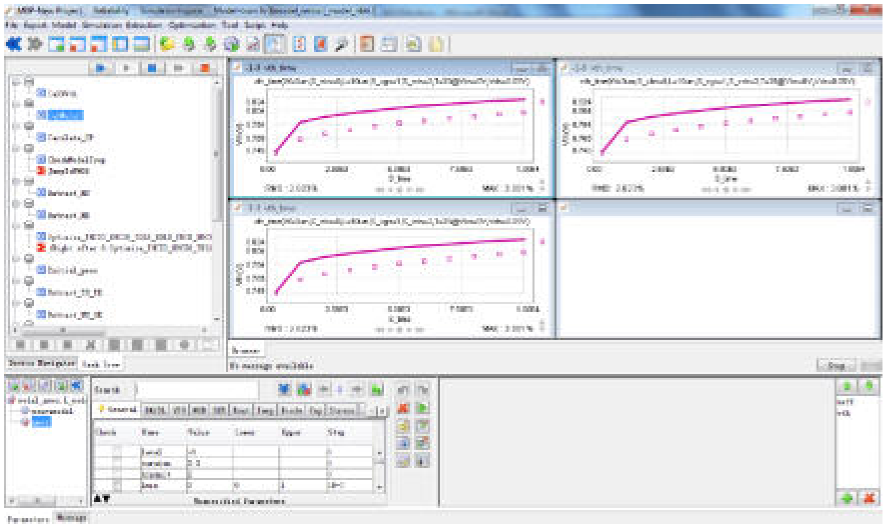
Comparison result



Run Task Tree

MBP also provides a built-in automatic extraction flow (task tree) for the MOSRA model. Task tree can be enabled by choosing Extraction > Task Tree from the main menu. After loading the task tree, you could run the flow automatically, or step by step. Task tree will select devices, region and parameters for optimization automatically. The task tree optimization window is shown in the following figure:

Task tree of a MOSRA model



Load Multi-Die Data

This application note describes how to set up the script-based environment so that MBP can load and utilize ET data.

NOTE This document was originally released for MBP V2011.1.0 and above in October 2011.

Overview

Besides the single sweep data (.mea) and single point data (.dp), there is also a multi-die data structure called ET data. This kind of multi-die data is important for retargeting the final SPICE model and monitoring final corner models.

MBP has the ability to deal with this type of ET data. In this document, we describe how to set up the script-based environment so that MBP can load ET data and use it to run simulation. For more information go to www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at: www.keysight.com/find/contactus.

Script-based Environment Setup

You can complete the script-based environment setup by following these steps:

Step 1. Un-Zip

The default.7z is actually the sample script code package for loading ET data. It can be un-zipped in any appropriate path. It contains the folders and files shown in the following figure.

Script code package

? Unknown Attachment

script_code.gif

Step 2. Start a New Project

Choose File > Project > New from the main menu. Doing so will start a new project.

Step 3. Load the Script Project File

You must choose Script > Script Project from the main menu to pop up the MBP Script window. Click the Open icon. Then, load the file jt.prj in the folder where the the project is just un-zipped.

MBP Script window

? Unknown Attachment

script_window.jpg

Step 4. Run the Script Project

After loading, choose the *load_etdata* item by clicking Project > default > sys > gui > menu > load_etdata.

load_etdata

? Unknown Attachment

load_etdata.jpg

Step 5. Load ET Data

A new menu ERData > Load should now be added under Script from the main menu. Choose to load the ET data (multi-die data), as shown in following figure.

Load ETData

? Unknown Attachment

load_ETData.gif

Step 6. Plot ET Data

Go to the IMV page and refresh the IMV tree. Here, you will find the Idsat_l_et example, as shown in the following figure:

IMV tree

? Unknown Attachment

imv_tree.jpg

Click this IMV item. The multi-die plot will appear as shown in the following figure:

Multi-die plot

? Unknown Attachment

multidie_plot.jpg

Customization

You can also modify the source script to hide/show the mean (or median) value of the multi-die data. Let's use the above Idsat_l_et plot as an example. Choose Script > Script Project from the main menu to pop up the MBP Script window. In the Project tab, click default > imv >; imv > idsat > idsat_l_et . Double click the idsat_l_et item to display the code window.

Script for the Plot

? Unknown Attachment

script_plot.jpg

In the script, there are variables to show/hide the mean and median value: isshow_mean and isshow_median. Set it to true (or false) to show (or hide) the curve.

Mismatch Modeling

This application note introduces the basic components of Model Builder Program's (MBP's) mismatch module. The steps to run the built-in extraction flow and how to configure and plot an IMV graph in MBP are also demonstrated.

NOTE This document was originally released for MBP V2011.1.2 in December 2011.

Overview

Two devices in design that are the same (e.g., exactly the same property, geometry, etc.) may show different electrical behavior on the Silicon due to mismatch. The main reason for the difference is the local process variance across the wafer. Mismatch affects the yield and reliability of the final products. An accurate mismatch model is therefore, necessary to ensure the robust design of many analog and digital circuits.

MBP supports mismatch modeling and simulation for all major semiconductor devices such as MOSFETs, bipolar transistors, resistors, and capacitors. In this document, we first introduce the data format supported, plot configuration and Monte Carlo (MC) simulation in MBP. Examples of running the built-in extraction flow, and configuring and plotting an IMV graph are also demonstrated. For more information go to www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at: www.keysight.com/find/contactus.

Data Format

For mismatch, MBP supports two kinds of data formats. The first one is based on the actual measurement data, while the other allows you to input the mean and sigma value of the target.

Data Format I

Here is an example of the first data format supported in MBP:

```
miscondition {date=,type=NMOS}
Page (name=vth_gm,target=vth_gm,scale=1.0,p=(L,W))
{vds=0.1,Vgs=1,Vbs=0,icon=1E-7,T=25}
{0.18,2.0}
0.002604200182343308 0.015163779612074824 -0.00163863082
04789923
0.0012339958926110839 0.008895426625025848 -0.0048731228
537369775
..........................
{0.18,10.0}
0.0011114374468496058 0.007055480752867993 -7.
450025621094092E-4
4.969414898894353E-4 0.003966458228031433 -0.00242138927
38844667
..........................
```

The first line of the data file begins with the keyword miscondition and contains information like date and device type. The second line defines all page-related information. The information within the round bracket ( ) contains Page name, target, scale, and P variable. The information within the brace {} declares Page constants, including the bias/current condition and temperature.

The latter part is the data block information. Every curve block always begins with {L W} . All data information is then listed behind it. In this example, the data information is the threshold voltage difference (Δvth_gm) between two adjacent devices with the same geometry.

Data Format II

You can also choose the other format. As an example:

```
condition {corner = tt,date = oct_20_02} Datatype
{mismatch}
Version{1.0}
type{nmos} Delimiter{,} Instance{L, W, T}
Input{vds=0.05, Vgs=1, Vbs=-1,icon=1e-7} Targets{Ids}
Data{ L, W, T, vds, vgs, vbs, ids}
40, 5, 25, 0.05, 3, 0, 0, 6.448e-4
40, 2, 25, 0.05, 3, 0, 0, 9.836e-4
```

```
30, 2, 25, 0.05, 3, 0, 0, 9.6235e-4
20, 2, 25, 0.05, 3, 0, 0, 1.28e-3
20, 1, 25, 0.05, 3, 0, 0, 1.889e-3
20, 0.5, 25, 0.05, 3, 0, 0, 2.448e-3
6, 1, 25, 0.05, 3, 0, 0, 3.4727e-3
10, 0.5, 25, 0.05, 3, 0, 0, 3.503e-3
6, 0.5, 25, 0.05, 3, 0, 0, 4.1108e-3
...............................
```

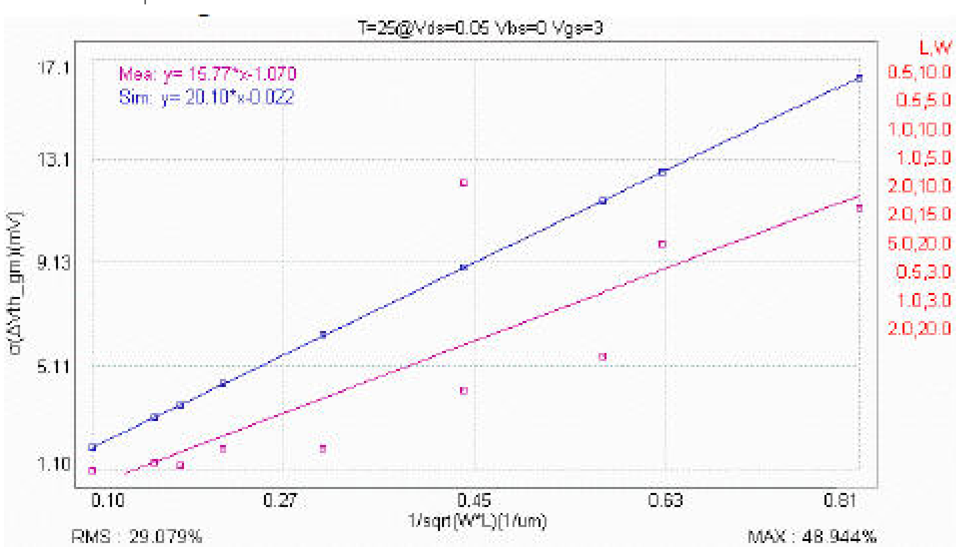
In this format, the first part of the data file contains general information such as corner type, date, data type, device type, instance, bias condition, and target. The second part of the data file includes the data block information. The first line begins with the keyword Data following the variables. All data is then listed from the second line. Note that the last two values in every line correspond to the mean and sigma of the target.

For example:
40,5,25,0.05,3,0,0,6.448e-4 means L=40um,W=5um, T=25C, Vds=0.05V, Vgs=3V, Vbs=0, ΔIds(mean) =0, ΔIds(sigma)=6.448e-4A.

Plot

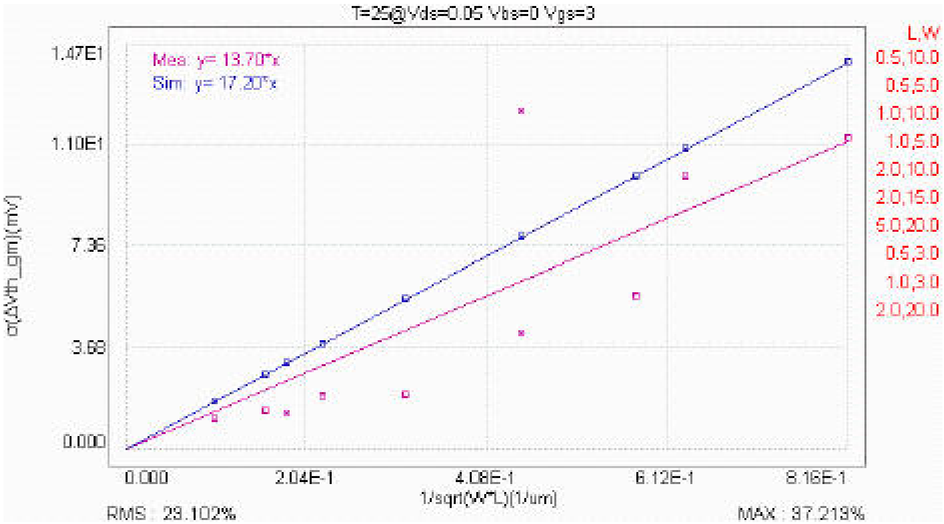
As shown in the following figure, the plot shows the value of **σ** (ΔVth_gm) versus *1 /sqrt (W*L), and the trend slope lines. The simulation point and trend line are plotted in blue and the measurement data and trend line are plotted in purple.

Mismatch plot



By clicking on the legend L, W you can disable/enable the geometries to be plotted. Right click on the plot and check the item Fit Line through Origin from the popup menu. MBP then forces the trend lines through the origin, as shown in the following figure:

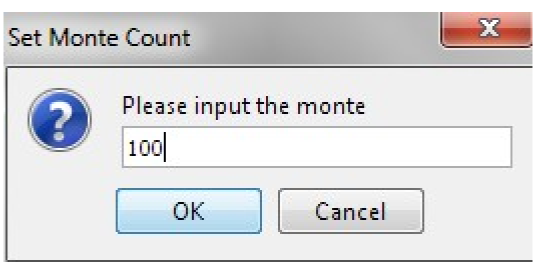
Fit Line through Origin



MC Simulation

MBP's internal engine supports Monte Carlo analysis of mismatch models. You can right-click on the plot and select Set Monte Count , as shown in the following figure. Then, set an appropriate number. A large number may lead to a more accurate result, but it can also cause a longer simulation time.

Set Monte Count



You have an option to execute a fast MC simulation by choosing Simulation > Fast-MC from the main menu.

Extraction Flow

We can use a demo to describe the steps required to run a mismatch model extraction through the built-in flow.

Demo Files

The demo folder is `$MBPHOME\demo\Mismatch\mosfet` Here, `$MBPHOME` is the MBP installation path. There are a total of three files in the folder:

- demo_model: the initial model card.
- param.txt: the parameter list used in the extraction flow.
- mis_data.mea: the demo data.

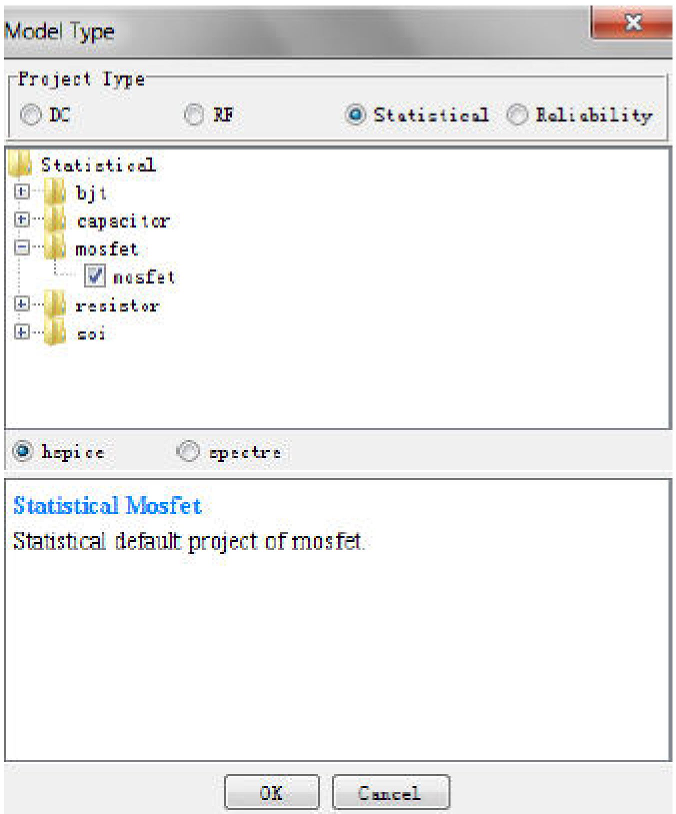
You can follow the steps below to complete the whole process.

Set Model Type

First, set the mode type. Choose Model > Select Model from the main menu. In the popup Model Type window, choose Statistical as the Project Type. Then choose mosfet as the device type, as shown in Figure: Model Type.

Click the OK button to close the window.

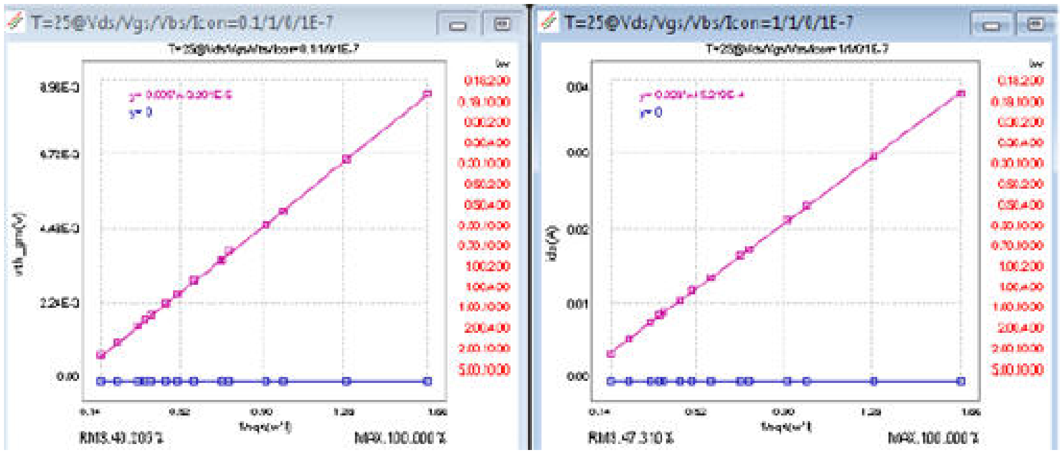
Model type



Load Data and Model

Choose File > Data > Load from the main menu and load the data file mis_data.mea as shown in the following figure:

Data VS initial model

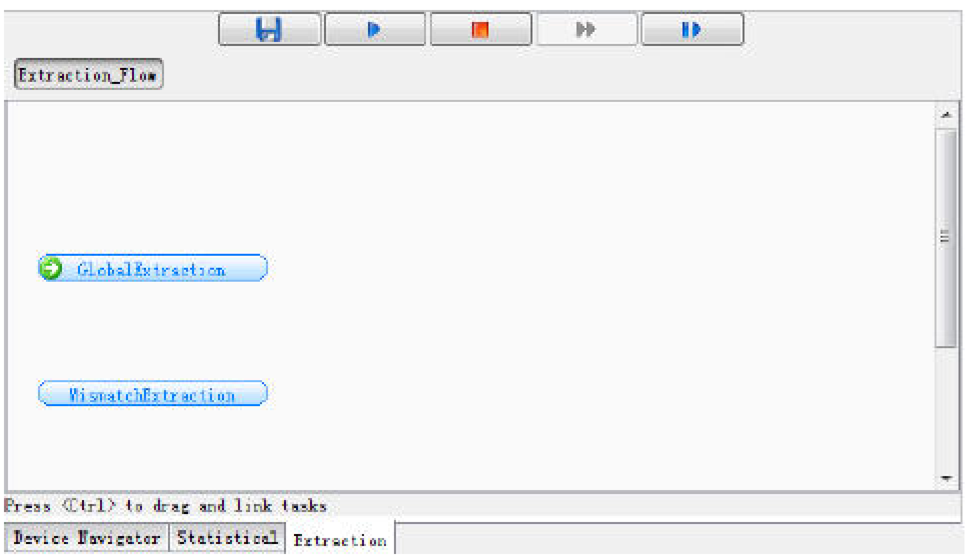


Then, choose File > Model > Load from the main menu and load the model file model_nmos.l. Here, MBP supports the ability to load the model with or without mismatch information.

Extraction Flow

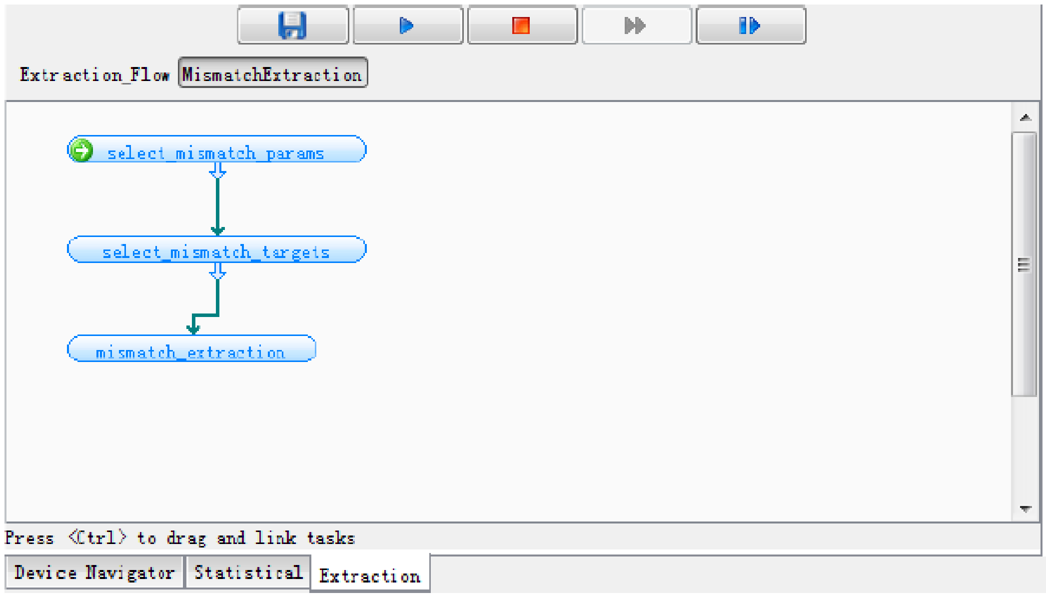
Choose Extraction > Extraction Flow from the main menu. The extraction panel is shown in the following figure:

Extraction Flow panel



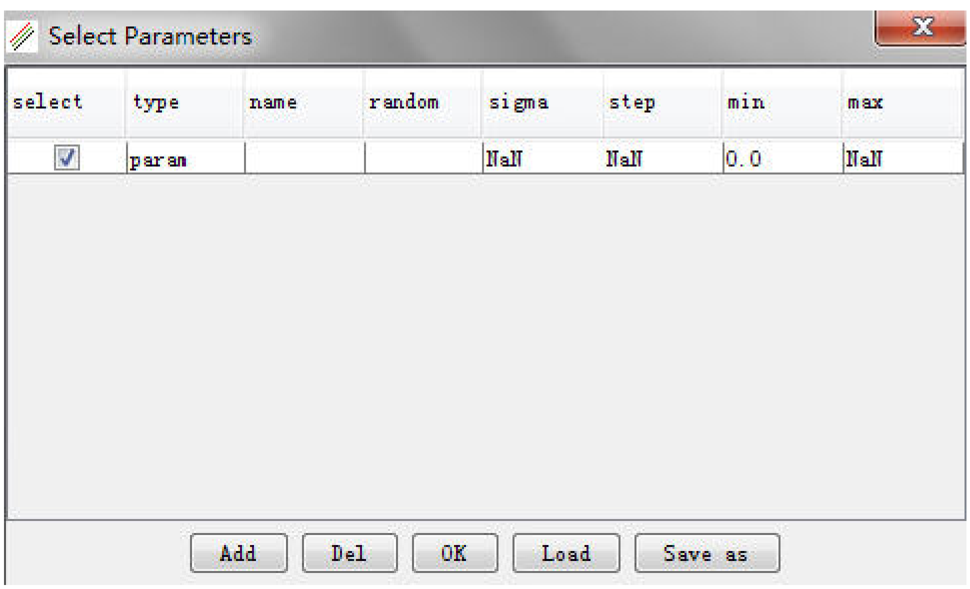
Double-click the Mismatch Extraction button to expand the flow, as shown in the following figure:

Global Extraction flow



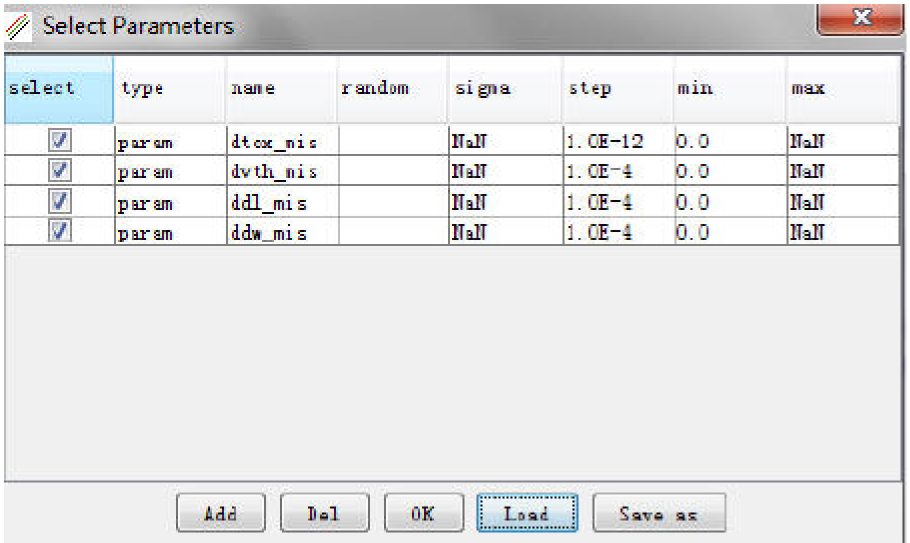
There are three steps in the flow: select_mismatch_params, select_mismatch_targets and, mismatch_extraction. Click the run icon to run the mismatch extraction flow. The Select Parameters window pops up as shown in the following figure.

Select Parameters window



Click the Load button to load the parameter list file param.txt. The parameters used for mismatch extraction are shown in the following figure.

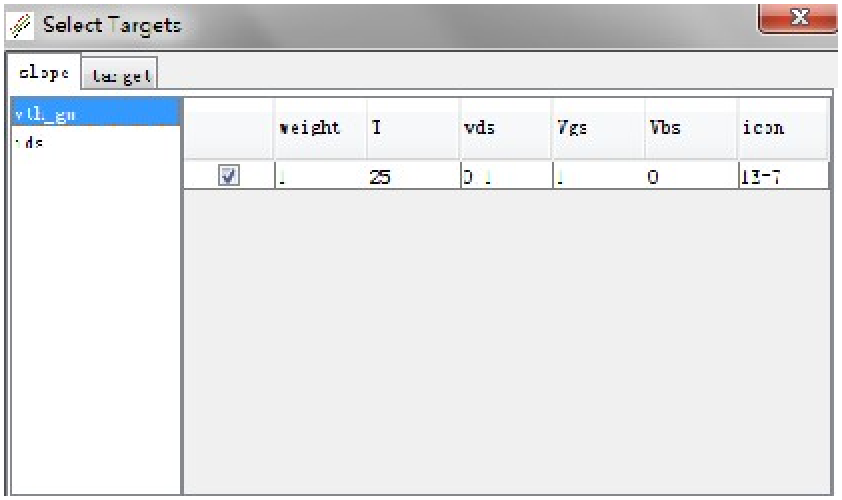
Load Parameter list



Some comments on the column names in the figure above are as follows:

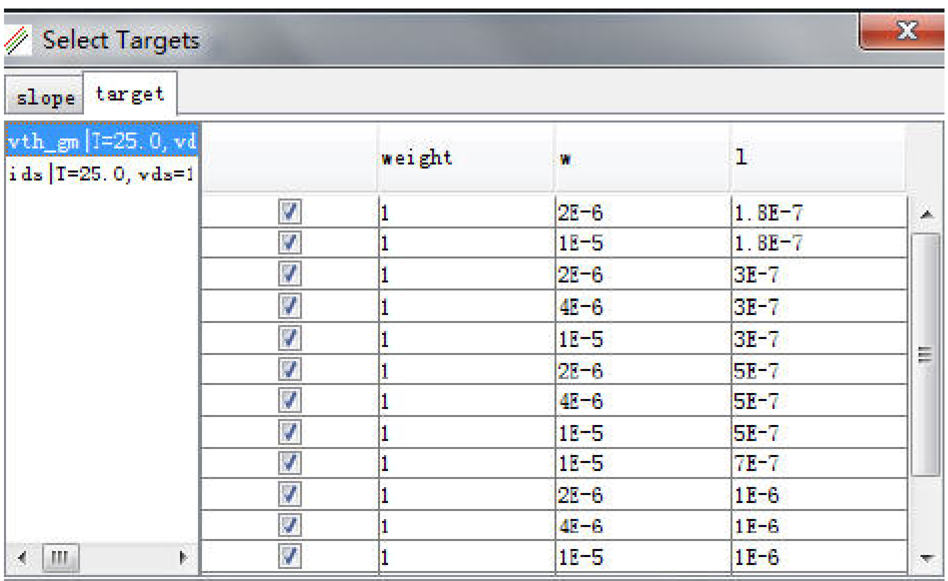
- select: when it is checked, the parameter gets re-extracted. If it is unchecked, then the parameter depends on the sigma value. If the sigma value is given, the final value of the parameter is the sigma value. If sigma value is not given, the current value of the parameter remains as the final one.
- name: mismatch parameter name. When *select* is unchecked, the parameter name can be blank. At the same time, both random and sigma must have correct values.
- random: random variable name. If in the current model file there is a random variable for the parameter, then put the name of the random variable here. If in the current model file there is no random variable for the parameter, you can either input a new name here, or keep it blank. For the latter case, a new name is created automatically. Note that the names of random variables cannot be repeated.
- sigma: sigma of the mismatch parameter. You can input the value here. The extraction flow then bypasses this parameter and uses the predefined value instead.
- step: the step for BPV calculation. It is used to calculate the sensitivity of the parameter to the target. Click the OK button to continue. The Select Targets window pops up as shown in the following figure.

Select targets - slope



In this window, you can select the data group for the following extraction and the corresponding weight. Click the target tab in the window as shown in the following figure.

Select targets - target



All the specific targets corresponding to the data group in as shown in Figure: Select targets-slope are listed here. You can also change the weight values, which affects the final value of the slope.

Close the window to continue. In the last step, the Save dialog window pops up. Input a file name to save the extracted model file. Then, the following mismatch parameters that have been extracted are found:

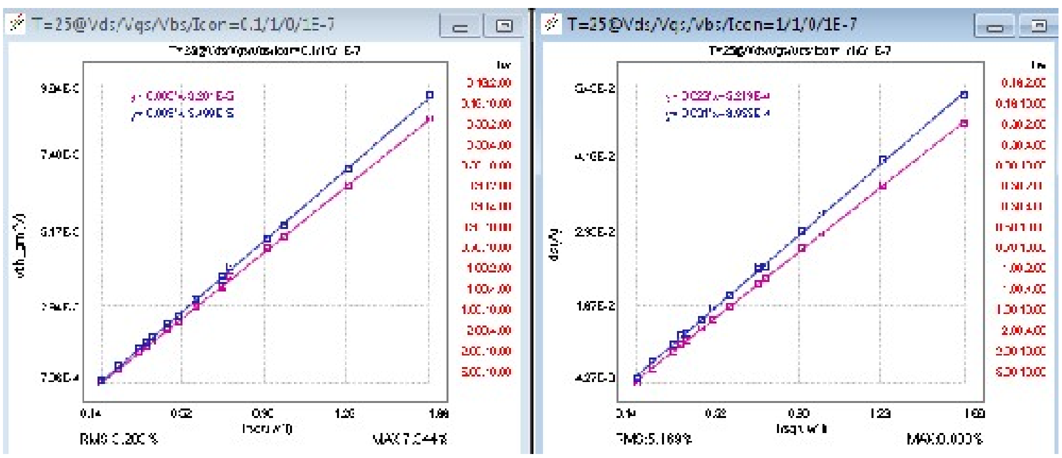
```
.param
+s_dtox_mis = 8.320052E-10 s_dvth_mis = 6.310913E-4
s_ddl_mis = 9.80127E-2
```

```
+s_ddw_mis = 6.175251E-3
.param
+random5 = agauss(0.0,1.0, 1)
+random6 = agauss(0.0,1.0, 1)
+random7 = agauss(0.0,1.0, 1)
+random8 = agauss(0.0,1.0, 1)
.param
+dtox_mis = &acirc;0.0+s_dtox_mis*random5'
+dvth_mis = &acirc;0.0+s_dvth_mis*random6'
+ddl_mis = &acirc;0.0+s_ddl_mis*random7'
+ddw_mis = &acirc;0.0+s_ddw_mis*random8'
```

The fitting result is shown in following figure.

You can continue to fine tune the parameters manually. They can also modify some settings and rerun the flow until a satisfying result is obtained.

Fitting result

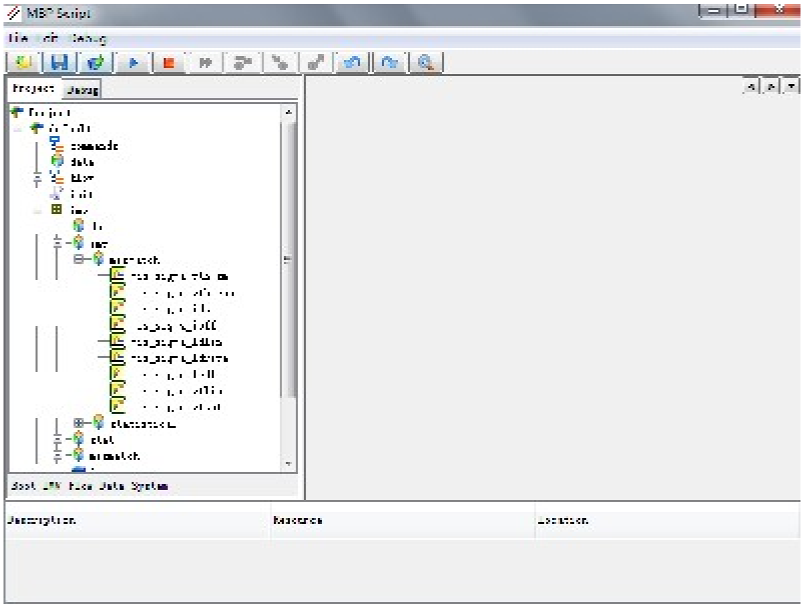


Mismatch IMV

Lastly, we use a demo case to illustrate how to customize mismatch IMV and plot it in MBP. After loading the data and model, choose Script > Script Project from the main menu to pop up the MBP Script interface.

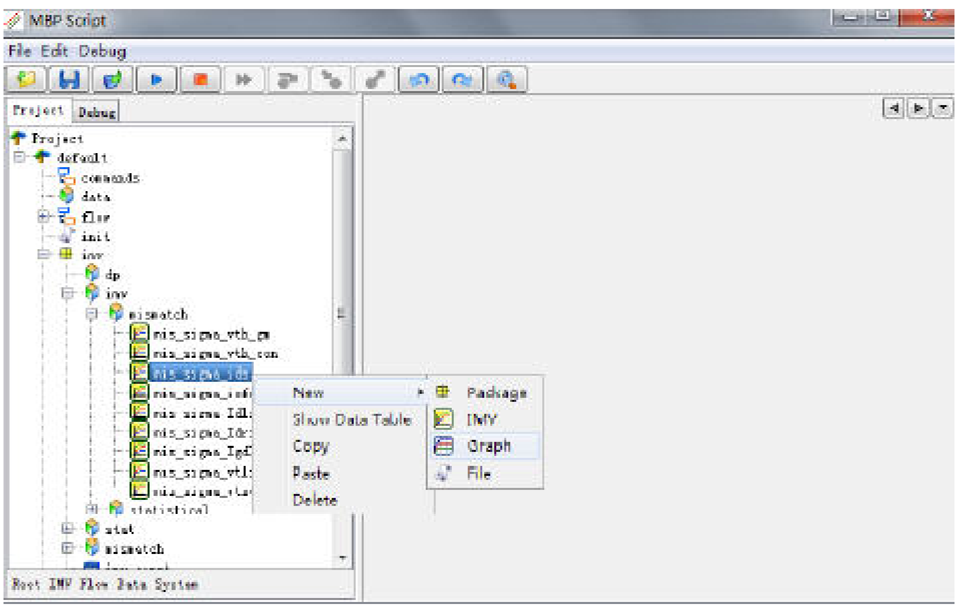
In the left Project tab window, click default > imv > imv > mismatch to expand the folder, as shown in the following figure:

MBP script



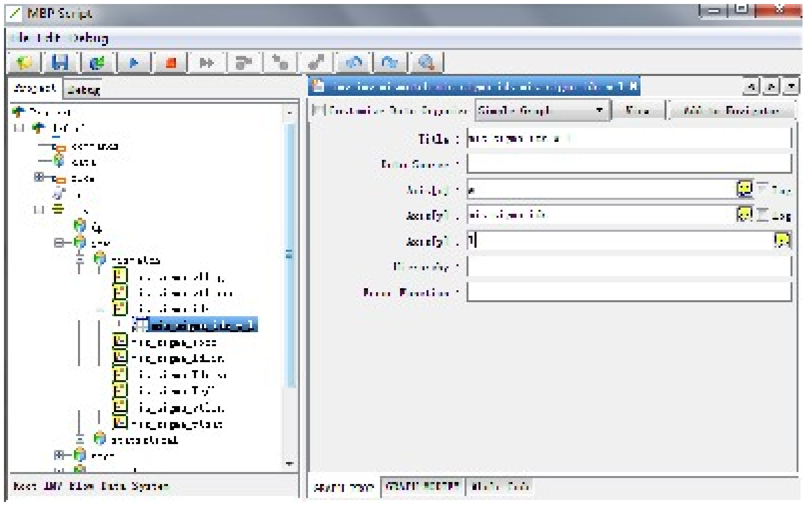
Right-click the IMV mis_sigma_ids and choose New > Graph (see following figure).

Create new graph



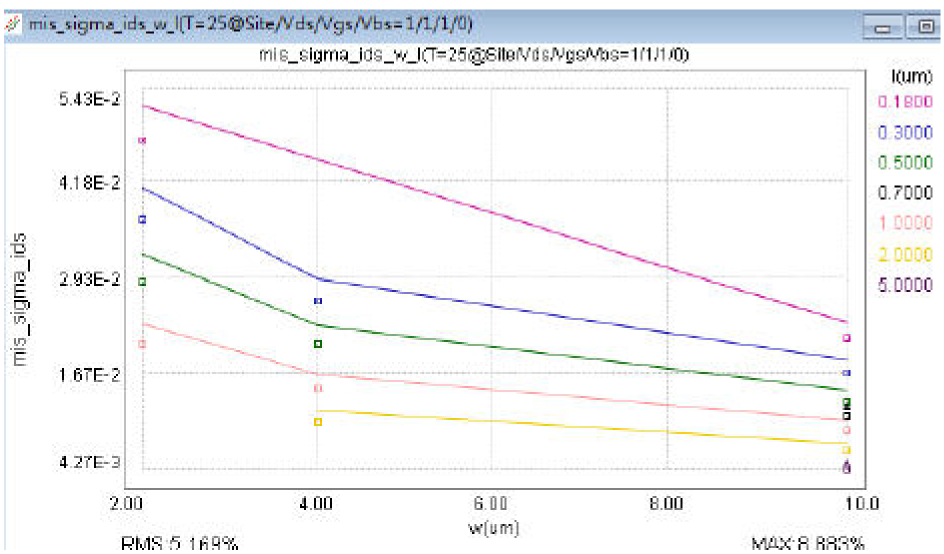
Input mis_sigma_ids_w_l as the code name. In the GRAPH_PROP tab, choose w as Axis[x], mis_sigma_ids as Axis[x] and l as Axis[x] as shown in the following figure.

Configure graph



Click the icon to save the current code. In the main menu of MBP, choose Extraction > IMV > IMV Pages to open the IMV page. Click the icon to refresh. You can then view the customized IMV page (mis_sigma_ids_w_l) as shown in the following figure:

IMV pages



Multiple Simulations

This application note describes how to compare two or more different models in Model Builder Program (MBP).

NOTE This document was originally released for MBP V2011.1.0 in July 2011.

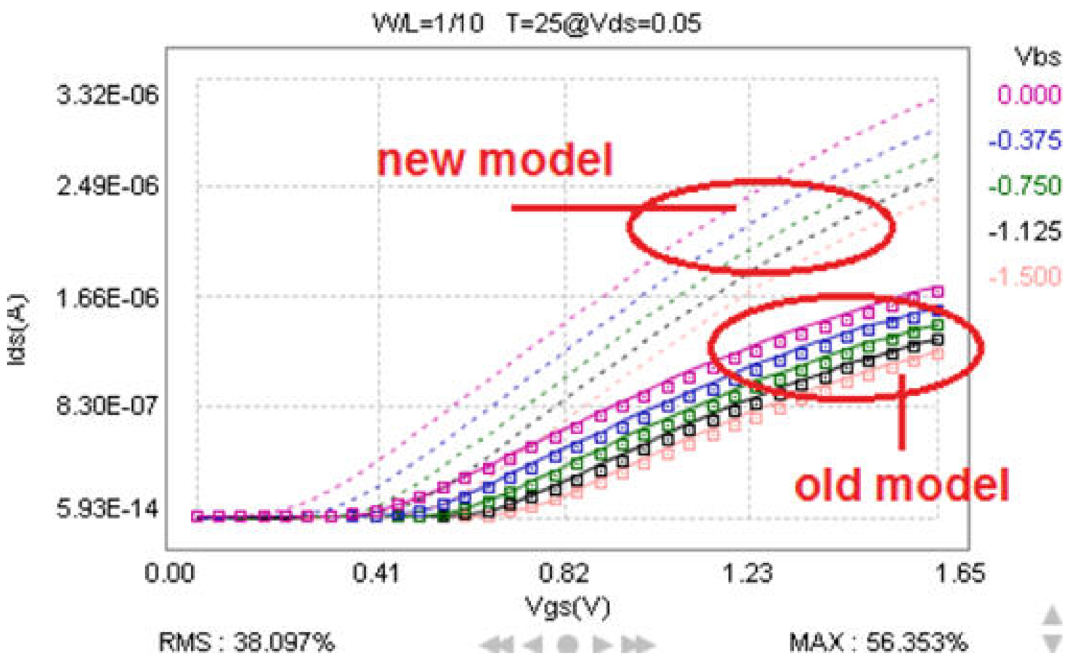
Overview

MBP provides an option to plot two or more models (including binning models) on screen at the same time. This feature is specifically designed for modeling engineers tuning or comparing different models.
This document introduces the steps and options to running a multiple simulation. For more information go to www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at: www.keysight.com/find/contactus.

Double Simulation

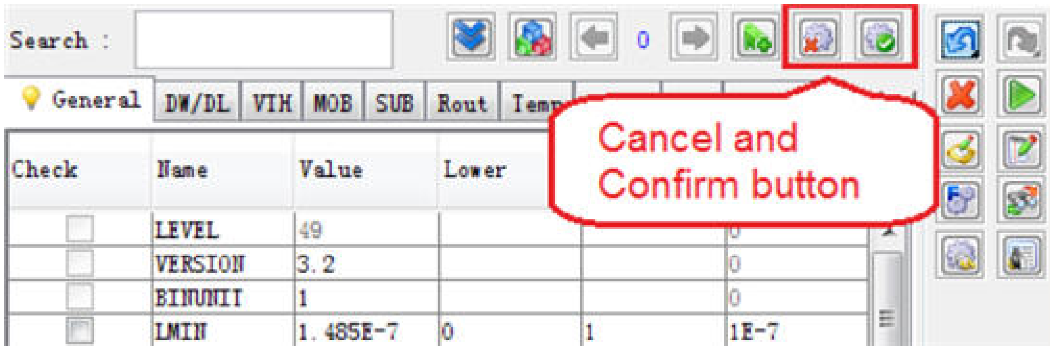
MBP features a double simulation function that can be used to compare two models— the models before and after tuning— by plotting their simulation results on screen at the same time.

Double simulation result



This function is enabled by clicking Simulation > Double Sim from the main menu or pressing the Ctrl and R buttons on the keyboard at the same time. After that, two new icons (Cancel and Confirm) are added to the optimization pane, as shown in the figure cancel and confirm icons. You can then easily compare the results of the two models.

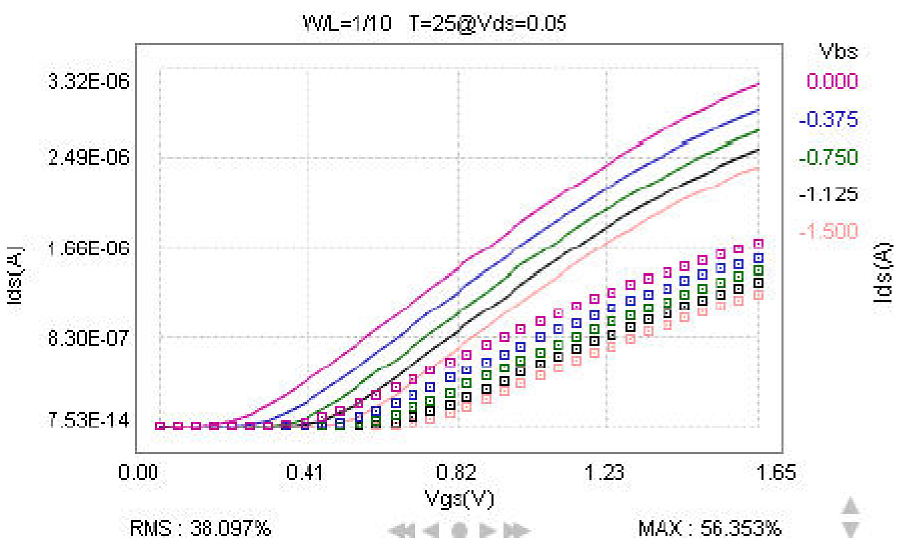
Cancel and confirm icons



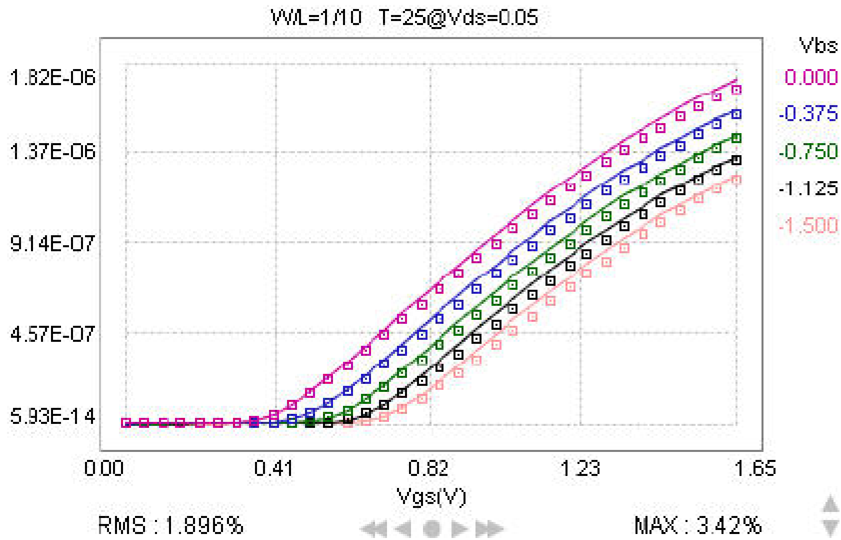
The old (original) model is represented by the solid line and cannot be modified. When one tunes a parameter, the new (current) model, which is represented by the dotted lines, becomes active and changes with the parameter.

On clicking the Confirm icon , the new model is kept. MBP then updates the old model using the parameters of the new model, as shown in the Figure: Confirm to use the new model. On clicking the Cancel icon , MBP reverts back to the old model as the current model, as shown in Figure: Cancel to use the old model as the current.

Confirm to use the new model



Cancel to use the old model as the current

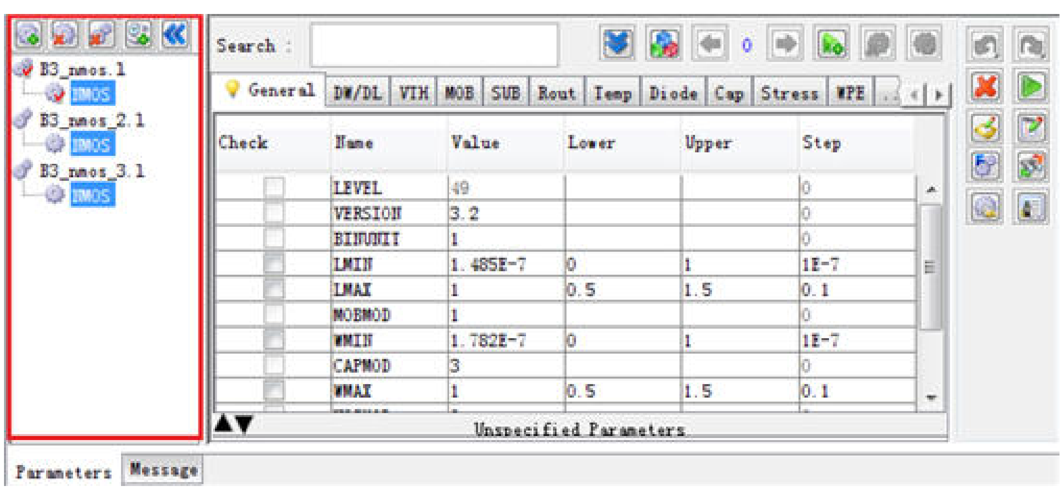


When the double simulation functionality is in use and you want to save the model, only the new model gets saved. In case you quit the double simulation after some adjustment, MBP deletes the old model (solid one) and uses the new one (dotted one) as the current model.

Multiple Models Comparison

In addition to performing double simulation, MBP also allows you to compare multiple models. The model manager interface is located at the bottom-left corner of the main graphical user interface (GUI), as shown in figure Model manager interface. There are five icons at the top of this model manager. From left to right, they are: Add Model,Remove Model,Compare,Save Version,and Hide. A model list is located under these icons. All models added to MBP can be found here. Model cards can be loaded into the model manager. In case you want to load a model library, it can be done by choosing Utilities > Lib Parser from the main menu.

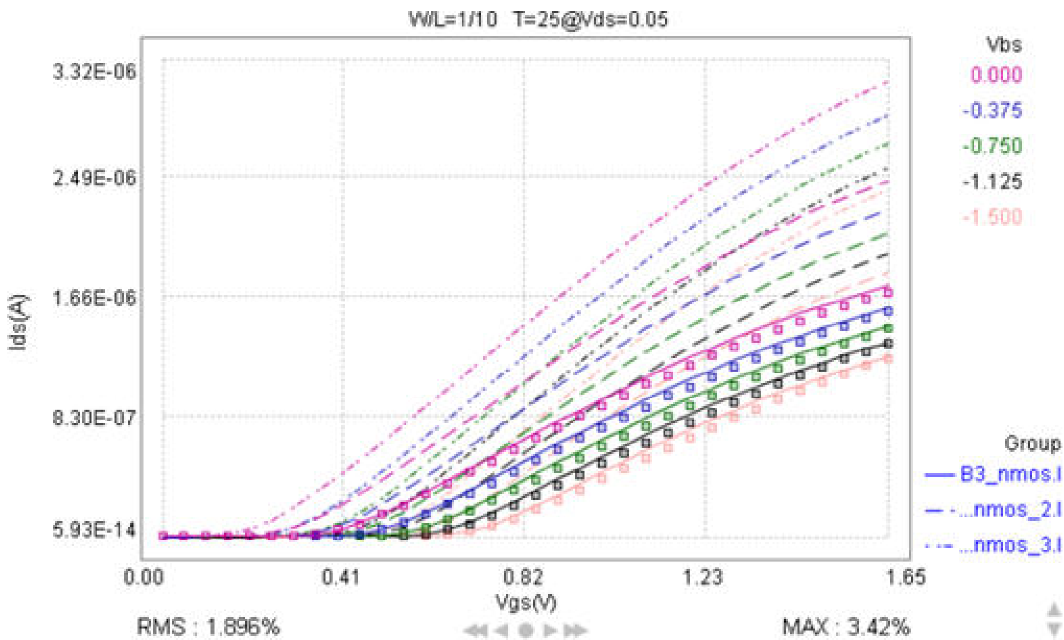
Figure 4. Model manager interface



The Hide icon  , as the name indicates, is used to minimize the model manager interface. The Add Model  and Remove Model  icons are for loading/removing models into/from the model manager. The Compare icon  is used to compare models in the model manager.

You can select multiple models (at least two) by clicking the models while pressing the Ctrl button on the keyboard. Next, click the Compare icon. The results can be viewed on the plot panel as shown in following figure.

Multiple models comparison



You can then observe the simulation results of three different models. The legend showing line symbols and the corresponding model names can be found at the bottom right corner of the screen.

NOTE

During the comparison the **Compare** icon  becomes the **Drop Compare** icon  .

Optimization Weight Setting

This application note describes how to set weight in Model Builder Program (MBP).

NOTE This document was originally released for MBP V2011.1.0 in August 2011.

Overview

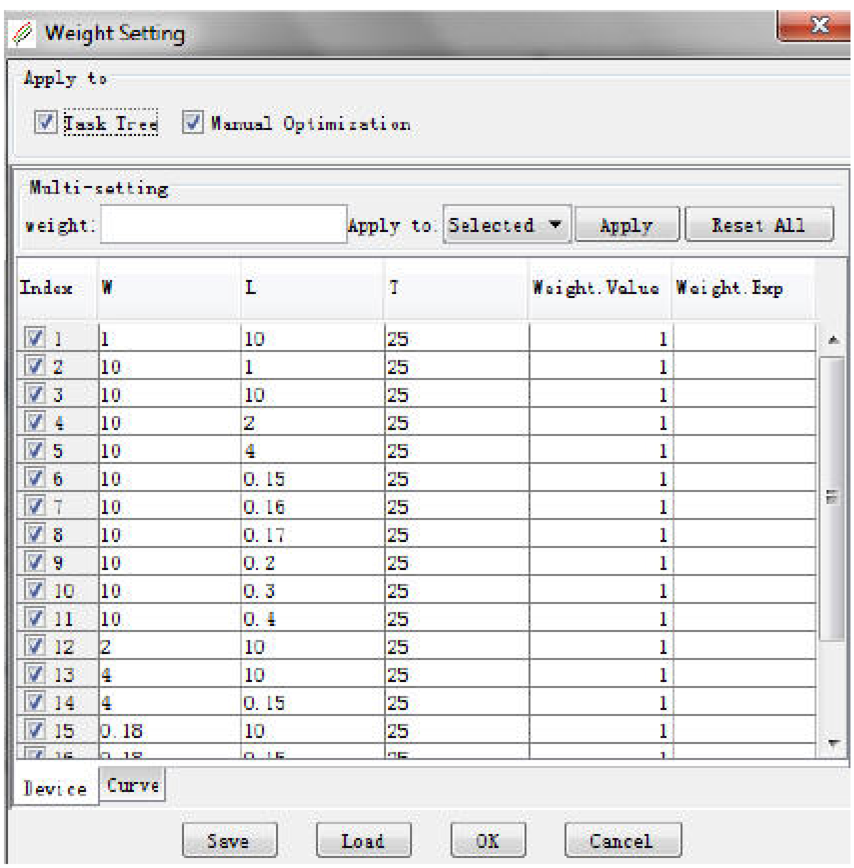
In MBP, weight can be separately assigned to device and curve. The weight setting takes effect when calculating root-mean-square (RMS) and thus, affects the final optimization result. When weight is set on one curve, all points on that curve inherit the weight value. In MBP, the default value of weight is always "1"; In this document, we introduce the steps to set weight for device and curve, respectively. For more information go to www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at: www.keysight.com/find/contactus.

Device Weight Setting

MBP allows you to set different weights for different devices when running the optimization. Here, weight means the part's weight in the whole integration. For example, the default weight for every device in MBP is 1, so the RMS values of all devices are multiplied by 1 (e.g., they remain unchanged). Since the built-in optimizer implements optimization according to the RMS value, it treats all devices with the same importance. However, if you set the weight value of one device as 2, then the RMS value of this device will be multiplied by 2 and the optimizer will treat it with much more importance than ordinary devices with a weight of 1.

To enable this feature, choose Extraction > Weight Setting from the main menu. The weight setting dialog will pop up as shown in the following figure:

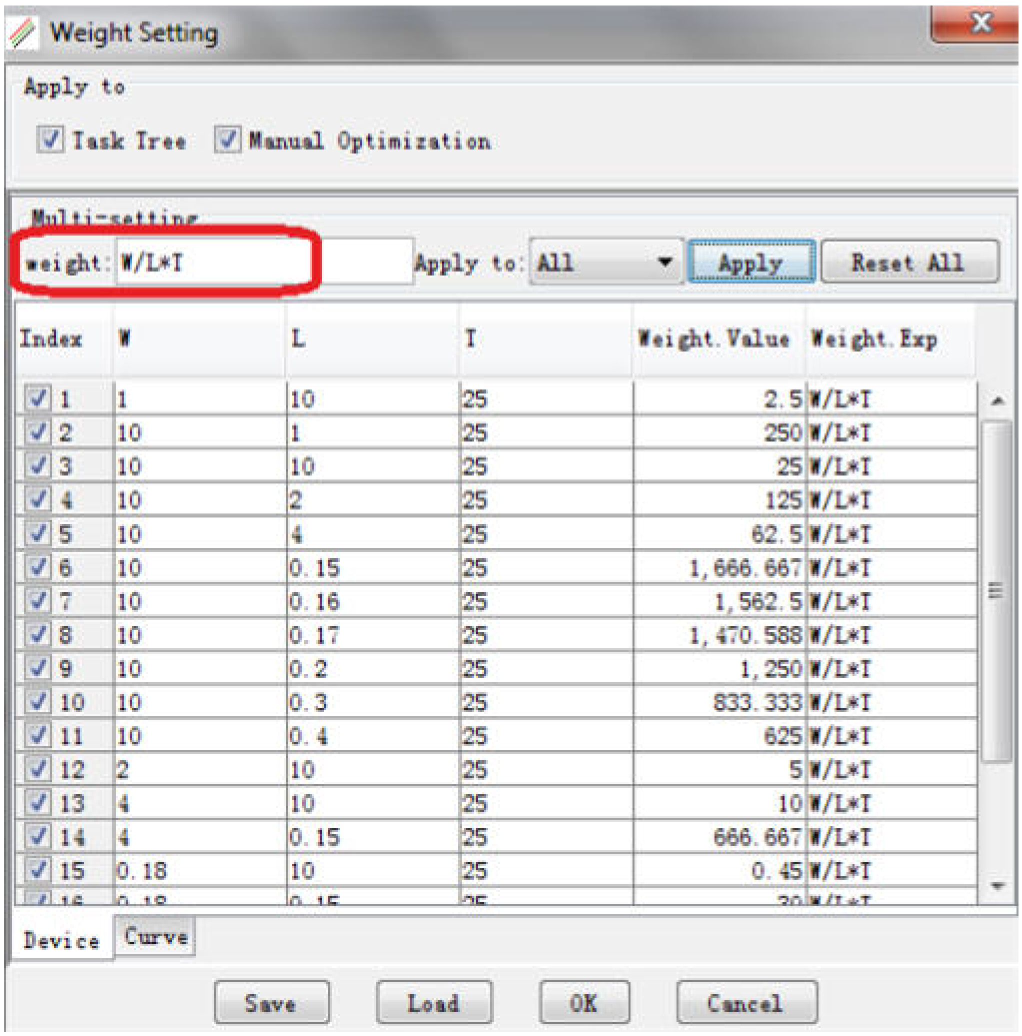
Weight setting for device



At the top of this dialog window there are two check boxes that allow you to apply the weight settings to either Task Tree or Manual Optimization. You can also check both options. In the Multi-setting panel, you can chose to apply the weight setting to the Selected devices or All devices. As for weight, you can either directly input the value or use the expression.

For example, as shown in figure Weight expression, weight is set as an expression given by W/L*T. After clicking the Apply button, the values and expressions in the column Weight.Value and Weight.Exp will be updated. Device instance parameters (such as W, L and T) may be employed in the expression.

Weight expression



MBP supported operator and functions are listed in Tables 1 and 2, respectively.

Support operator

Support Operator	Symbol
Power	^

Support Operator	Symbol
Boolean Not	!
Unary Plus, Unary Minus	+x, -x
Modulus	%
Division	/
Multiplication	*
Addition, Subtraction	+,-
Less or Equal, More or Equal	<=, >=
Less Than, Greater Than	<, >
Not Equal, Equal	!=, ==
Boolean And	&&
Boolean Or	||

Support function

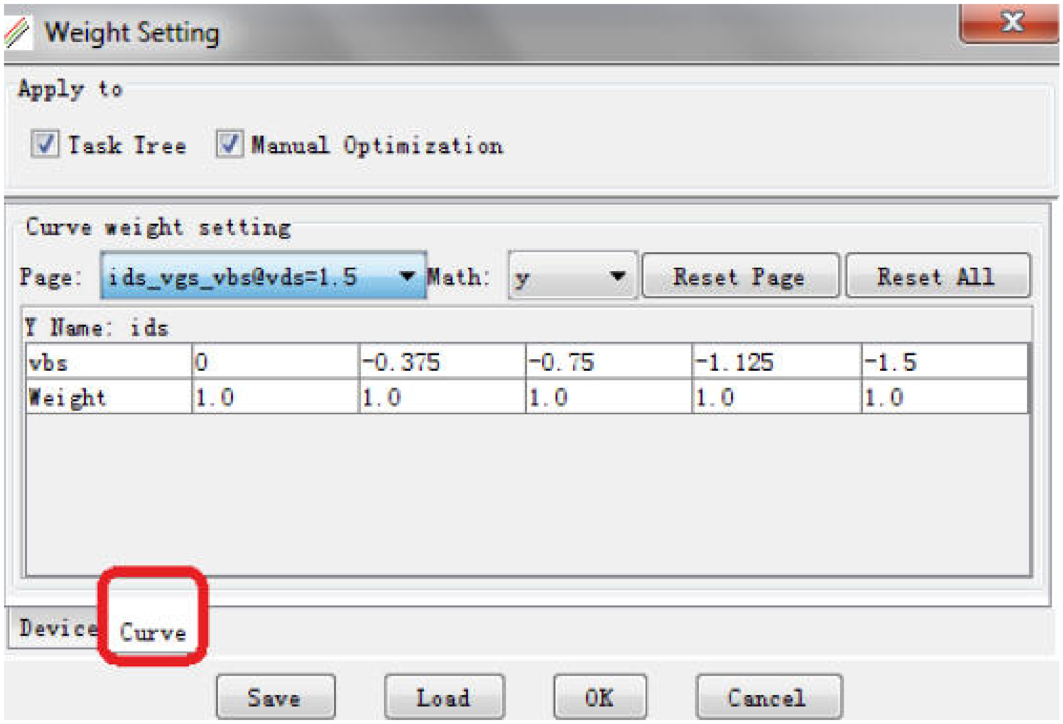
Support Function	Symbol
Sine	sin()
Cosine	cos()
Tangent	tan()
Arc Sine	asin()
Arc Cosine	acos()
Arc Tangent	atan()
Hyperbolic Sine	sinh()
Hyperbolic Cosine	cosh()

Support Function	Symbol
Hyperbolic Tangent	tanh()
Inverse Hyperbolic Sine	asinh()
Inverse Hyperbolic Cosine	acosh()
Inverse Hyperbolic Tangent	atanh()
Natural Algorithm	ln()
Algorithm base 10	log()
Angle	angle()
Absolute Value / Magnitude	abs()
Random number (between 0 and 1)	rand()
Modulus	mod()
Square Root	sqrt()
Sum	sum()

Curve Weight Setting

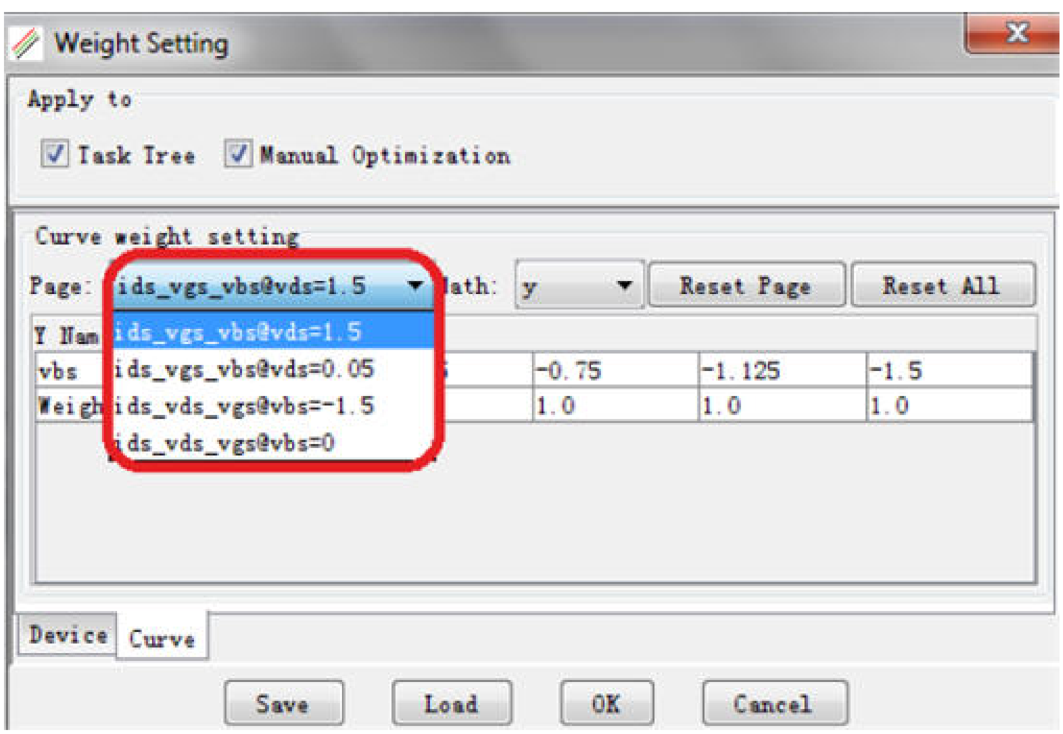
The weight can also be set in curve level. To do so, click the Curve tab to switch to the interface as shown in the following figure:

Weight setting for curve



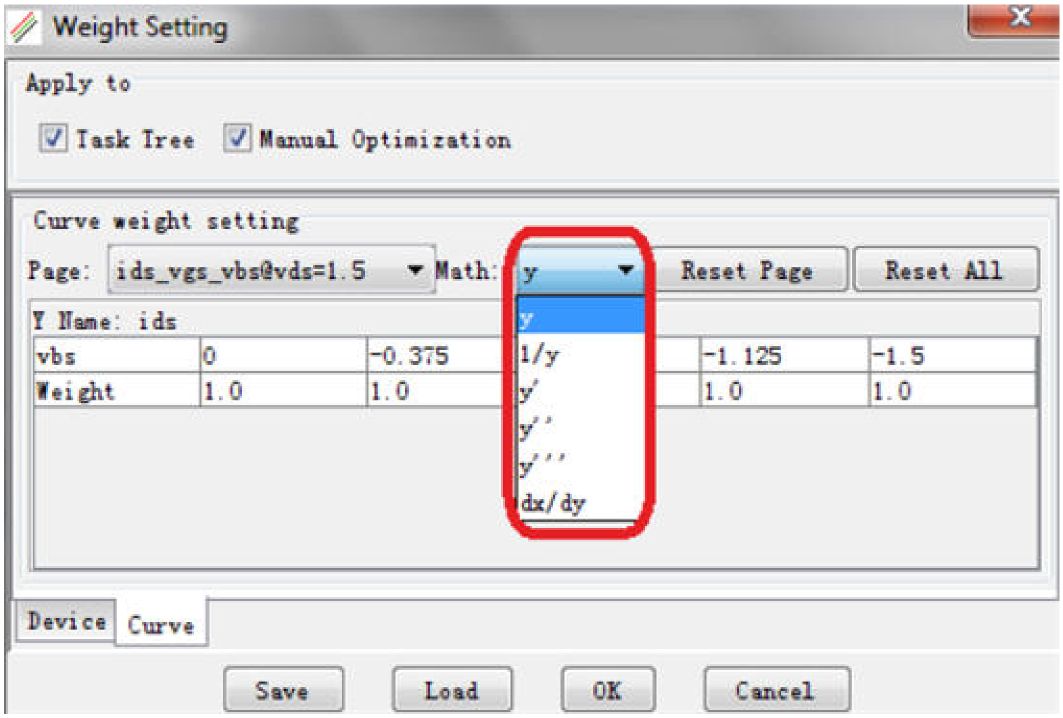
First, you need to define a plot. As shown in following figure, this is done by choosing one page from the drop-down list, for example, ids_vgs_vbs@vds=1.5

Set page



Next, select a math transform from the drop-down list as shown in figure Set Math. This is important because the user may be interested in Id_Vg_Vb or Gm_Vg_Vb (the derivative of Id_Vg) during the optimization. Choosing different math transformations can distinguish between these two plots.

Set math



The weight setting can also be accomplished by editing the value directly in the table. Note, however, that the final weight value of one curve needs to be multiplied by the weight of its device. So, the total weight of one curve is equal to `device_weight*curve_weight`.

After all settings are done, click the OK button to save and close the window. All of the selected settings will be immediately activated.

Parameter Boundry

This application note describes how to compare two or more different models in Model Builder Program (MBP).

NOTE This document was originally released for MBP V2011.1.0 in July 2011.

Overview

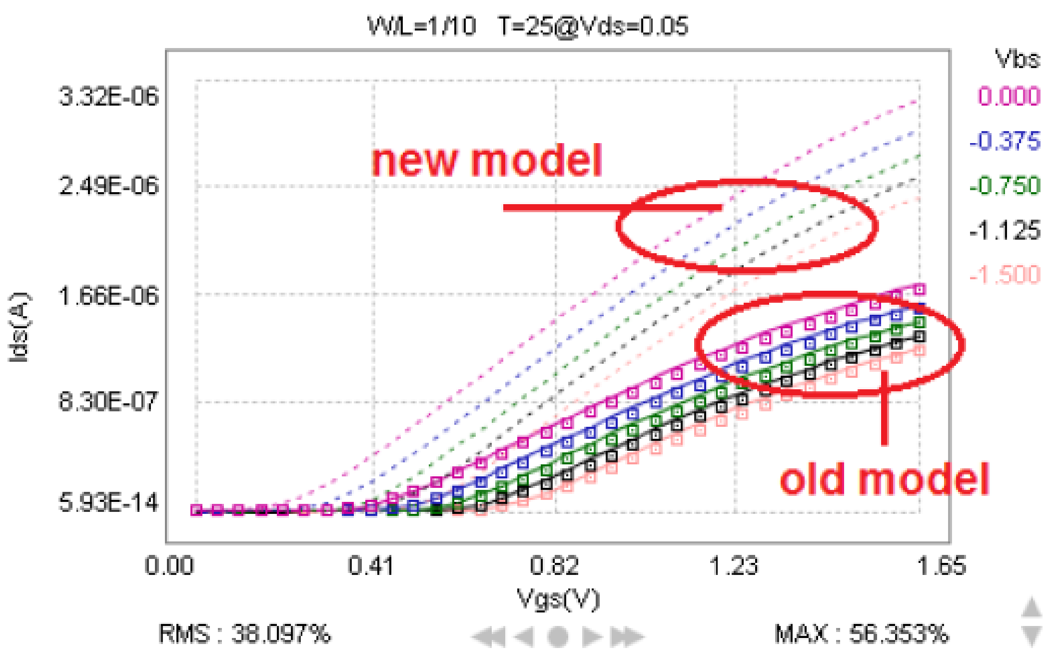
MBP provides an option to plot two or more models (including binning models) on screen at the same time. This feature is specifically designed for modeling engineers tuning or comparing different models.

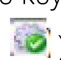
This document introduces the steps and options to running a multiple simulation. For more information go to http://www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at http://www.keysight.com/find/contactus.

Double Simulation

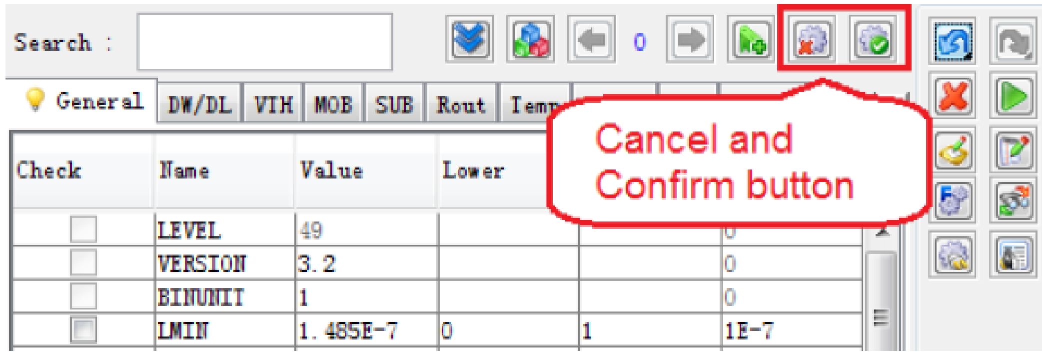
MBP features a double simulation function that can be used to compare two models' the models before and after tuning - by plotting their simulation results on screen at the same time (see following figure).

Double simulation result



This function is enabled by clicking Simulation > Double Sim from the main menu or pressing the Ctrl and R buttons on the keyboard at the same time. After that, two new icons (Cancel  and Confirm ) are added to the optimization pane, as shown in following figure. You can then easily compare the results of the two models.

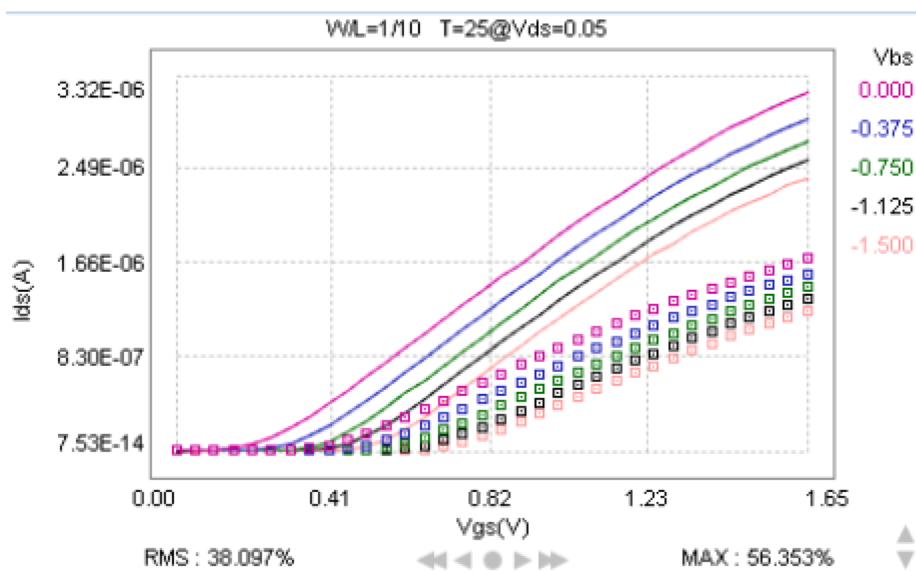
Cancel and confirm icons



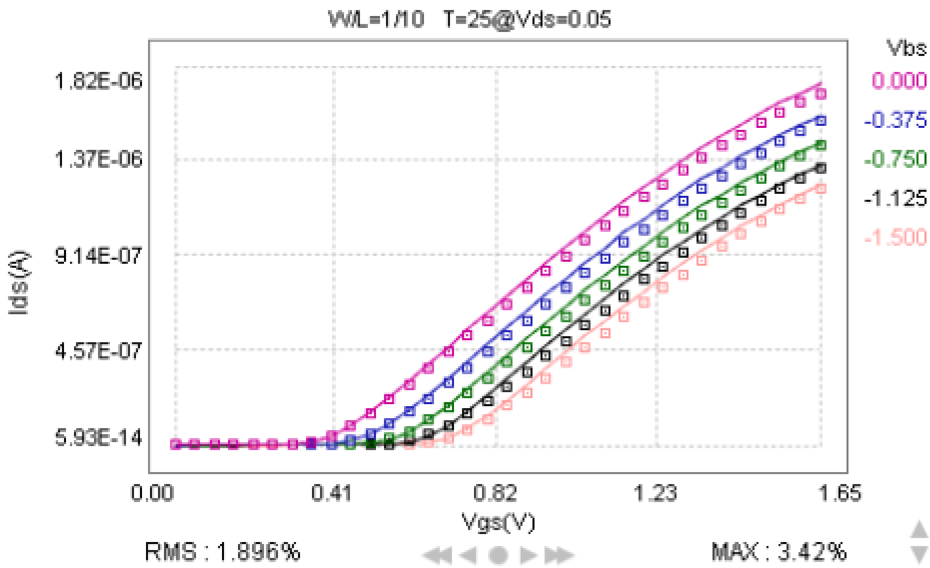
The old (original) model is represented by the solid line and cannot be modified. When one tunes a parameter, the new (current) model, which is represented by the dotted lines, becomes active and changes with the parameter.

On clicking the Confirm icon, the new model is kept. MBP then updates the old model using the parameters of the new model, as shown in Figure: Confirm to use the new model. On clicking the Cancel icon, MBP reverts back to the old model as the current model, as shown in Figure: Cancel to use the old model as the current.

Confirm to use the new model



Cancel to use the old model as the current

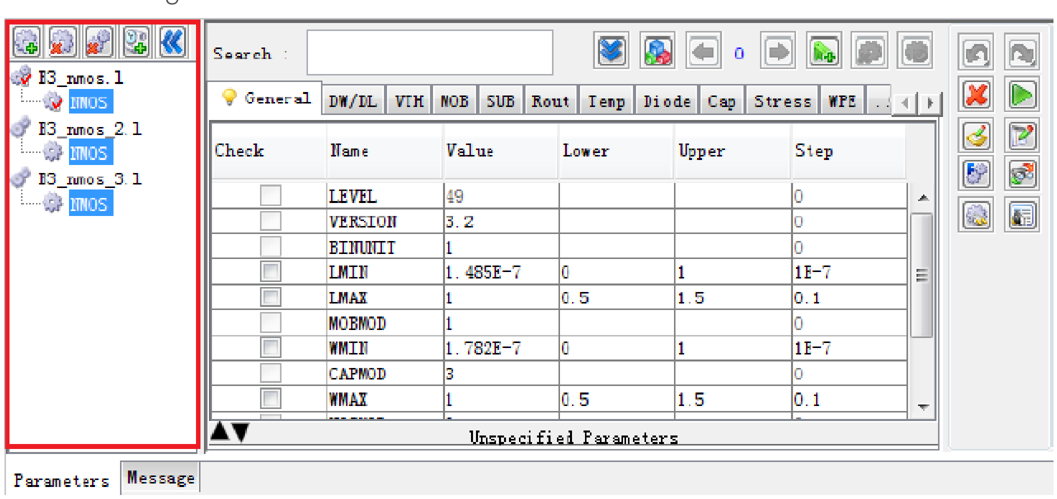


When the double simulation functionality is in use and you want to save the model, only the new model gets saved. In case you quit the double simulation after some adjustment, MBP deletes the old model (solid one) and uses the new one (dotted one) as the current model.

Multiple Models Comparison

In addition to performing double simulation, MBP also allows you to compare multiple models. The model manager interface is located at the bottom-left corner of the main graphical user interface (GUI), as shown in Figure 4. There are five icons at the top of this model manager. From left to right, they are: Add Model, Remove Model, Compare, Save Version, and Hide. A model list is located under these icons. All models added to MBP can be found here. Model cards can be loaded into the model manager. In case you want to load a model library, it can be done by choosing Utilities > Lib Parser from the main menu.

Model manager interface

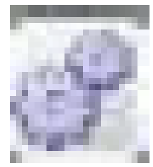


The Hide icon  , as the name indicates, is used to minimize the model manager interface. The Add Model  and Remove Model  icons are for loading /removing models into/from the model manager. The Compare icon  is used to compare models in the model manager.

You can select multiple models (at least two) by clicking the models while pressing the Ctrl button on the keyboard.

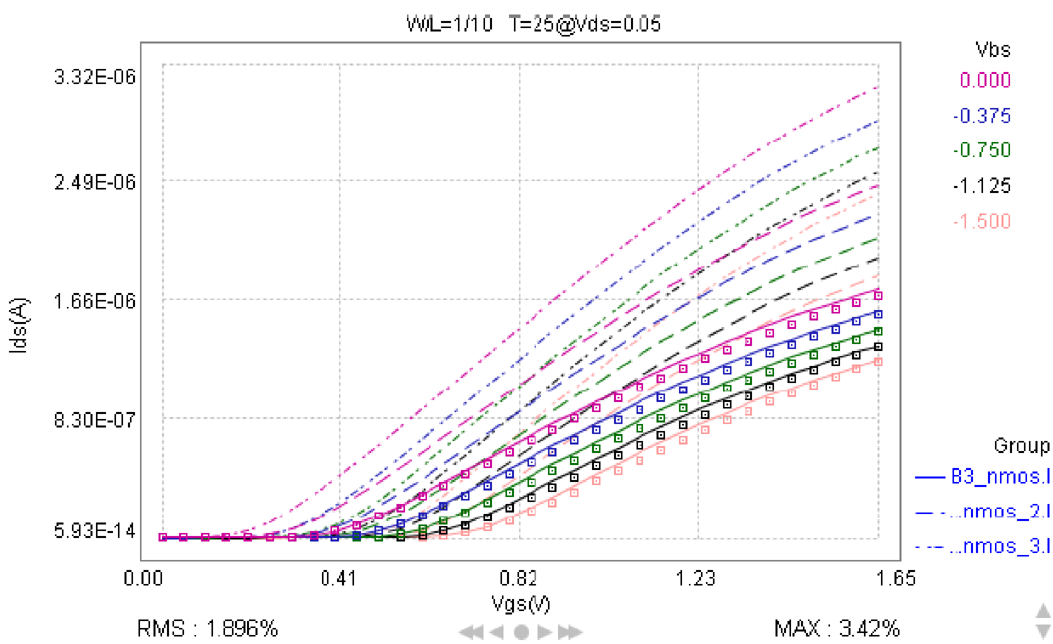
Next, click the Compare icon. The results can be viewed on the plot panel as shown in Figure: Multiple model comparison.

NOTE

During the comparison the **Compare** icon  becomes the **Drop Compare** icon 

.

Multiple models comparison



From above figure, you can then observe the simulation results of three different models. The legend showing line symbols and the corresponding model names can be found at the bottom right corner of the screen.

Statistical Modeling

Overview

Incorporating process variability into models is critical for IC design. Moreover, statistical modeling is today playing an ever important role in ensuring high product yield in the design phase. MBP supports Monte Carlo simulation with statistical modeling. In this document, we first describe the data format supported, plot configuration and Monte Carlo simulation in MBP. Additionally, the built-in flow used to run statistical model extraction with the Backward Propagation of Variance (BPV) method is elaborated. Finally, we use a demo to introduce the steps to configure and plot statistical IMV. For more information go to http://www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at: http://www.keysight.com/find/contactus.

Data, Plot and Simulation

Data Format

MBP supports two kinds of data formats for statistical modeling. The first is based on actual measurement data, while the second allows you to input the mean and sigma value of the target.

Data Format I

Below is an example of the first format (based on measurement data) supported in MBP:

```
staccondition{mode=forward,type=nmos}

Page(name=Ids,target={Ids(vds=0.05,vgs=1.1)},p=vbs){w=0.
18,L=0.15,T=25}

curve{0.0}

* site1           1.863622E-5
* site2           1.859326E-5
* site3           1.826478E-5
* site4           1.79857E-5
* site5           1.857634E-5
* site6           1.89573E-5
* site7           1.831234E-5
* site8           1.895472E-5
* site9           1.901374E-5
* site10         1.882349E-5
.........................
```

The first line of the data file begins with the keyword `staccondition` and contains working mode and device type information. From this example, we know that its a NMOS device in forward working mode.

The second line defines all page related information. The information within the round bracket contains page name, target and P variable. The information within the brace declares the page constants, including geometry and temperature. For any assistance, mail to: mbp_pdl-eesof@keysight.com

The latter part is the data block information. For every curve block, it always begins with the keyword curve and the corresponding P value (vbs, in this example). All data information is listed behind it. There are two values in every line: the site number and the target value (Ids, in this example).

Data Format II

You can also choose the other format for the statistical data, allowing to input the mean and sigma value of the target. An example is as follows:

```
condition{corner = tt,date = oct_20_02} Datatype
{statistical}

Version{1.0} type{nmos} Delimiter{,} Instance\{L, W, T}

Input{vds=0.05, Vgs=1, Vbs=-1,icon=1e-7} Targets{Vth_gm,
Idsat}

Data{L, W, T, vds, vgs,vbs, ids}

0.6, 0.27 , 25 , 5.5, 5.5 , 0, 1.94e-4, 2.44e-6

0.6, 10 , 25 , 5.5, 5.5 , 0, 1.94e-4, 2.44e-6

............................................

Data{L, W, T, vds,vgs,vbs, vth_gm}

0.6, 0.27 , 25 , 5.5, 5.5 , 0, 0.83, 8.95e-3

0.6, 10 , 25 , 5.5, 5.5 , 0, 0.83, 8.95e-3

............................................
```

In this format, the first part of the data file contains the general information such as corner type, time, data type, device type, instance, bias condition, and target. The latter part of the data file is the data block information. Every block begins with the keyword Data following the variables. All data is listed from the second line of each block. Note that the last two values in every line correspond to the mean and sigma of the target. In this example, line 0.6, 0.27,25, 5.5, 5.5, 0, 1.94e-4, 2.44e-6 means L=0.6um, W=0.27um, T=25C, Vds=5.5V, Vgs=5.5V, Vbs=0, Ids(mean)=1.94e-4A, Ids (sigma)=2.44e-6A.

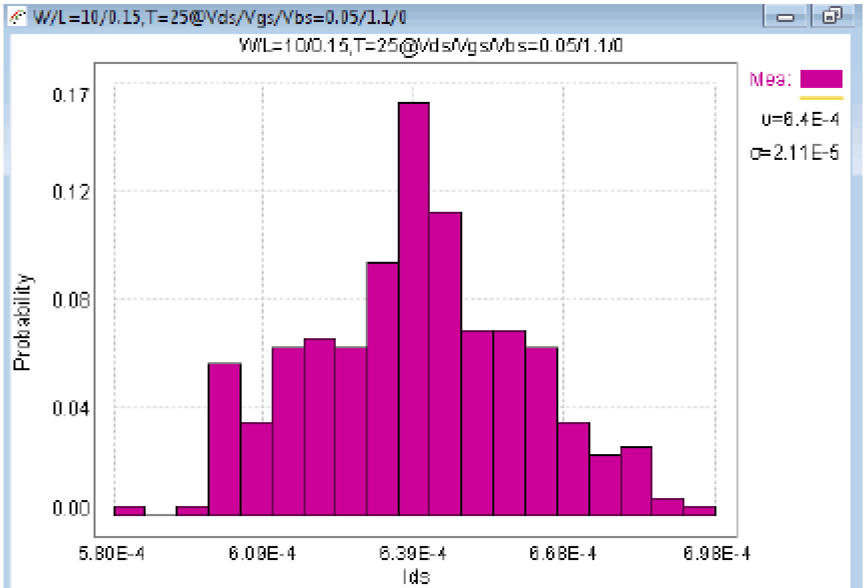
Data Plot

Depending on the number of targets, there are two kinds of plots: histogram and scatter.

Histogram for Single Target

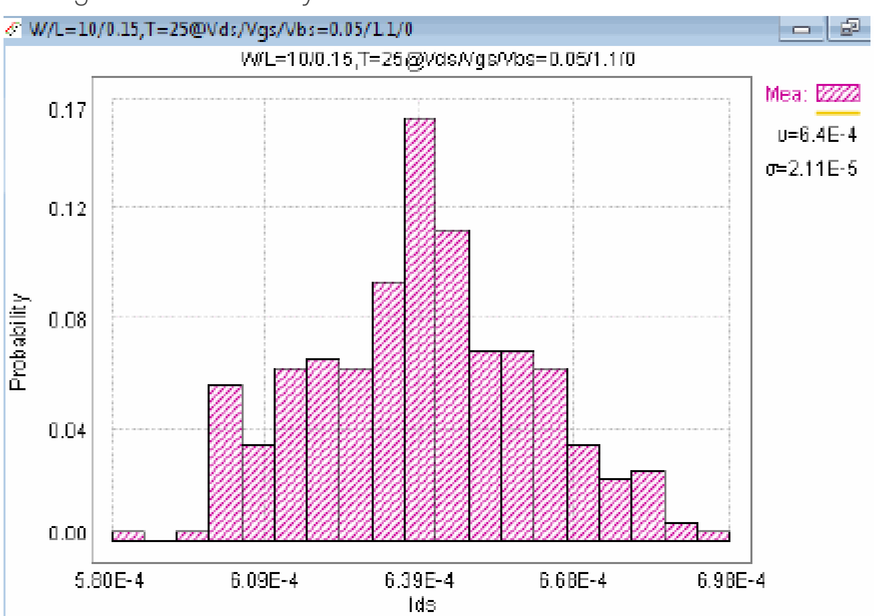
Using the target Ids_ as an example, the data plot(histogram) is shown in following figure.

Histogram with fill style



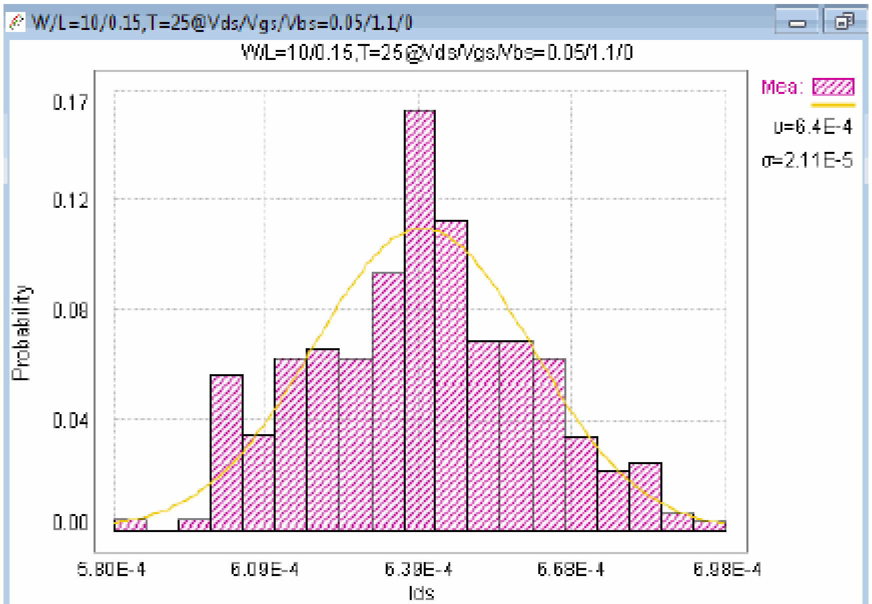
Two styles are available for the histogram: the fill style as shown in Figure 1 and the bias style as shown in following figure. For the Bias style, simply right-click the plot and in the popup menu choose Bar Styles > Bias.

Histogram with Bias style



You can also add a distribution line on the histogram. To do so, right-click on the plot and check the item Draw Distribution(Figure3).

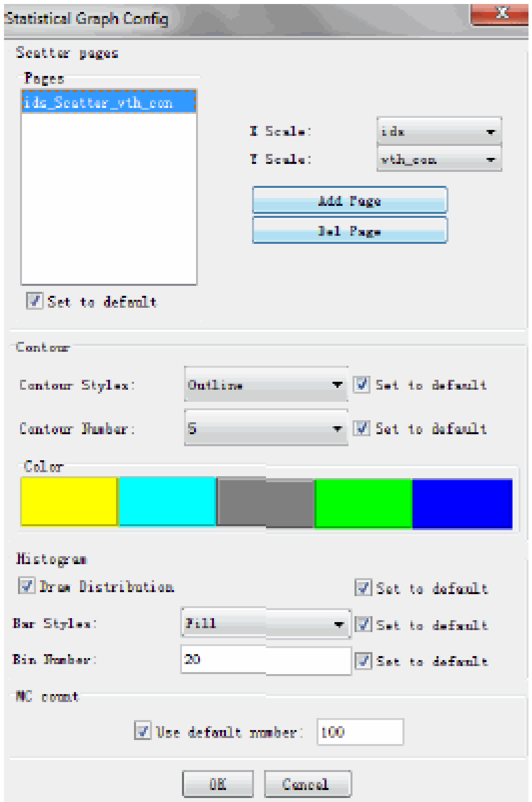
Histogram with distribution line



Scatter Plot for a Pair of Targets

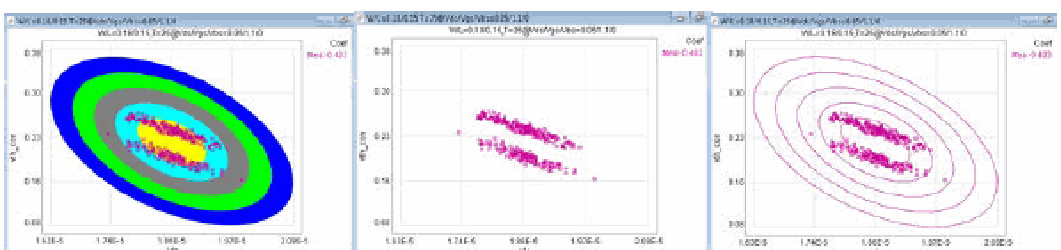
The scatter plot is used for a pair of targets. In this example, the pair of targets are defined as ids and vth_con. A scatter plot can be configured by the user on MBPs graphical user interface (GUI). Simply choose Tool > GUI Options > Statistical Graph Config from the main menu to open the Statistical Graph Configuration window as shown in Statistical graph configuration. MBP automatically parses the targets (e.g., ids, vth_con, vth_gm, Ion, and Ioff, etc.). These targets can be freely chosen as X- or Y-axis values on the scatter pages. In this example, we choose ids as X scale and vth_con as Y scale. Next, click the Add Page button.

Statistical graph configuration



The contour of the scatter plot is also set in this window. Currently, MBP allows you to set the contour number up to 5 (corresponding to the case of five sigma). There are three contour styles available to choose from: fill, null and outline. The corresponding scatter plots are shown in following figure.

Scatter Plots: fill (left-most image), null(center) and outline(right-most image)



MC Simulation

MBPs internal engine is capable of performing Monte Carlo analysis. You may also call an external simulator for verification purposes.

Statistical Model

To enable Monte Carlo analysis in MBP, you need to first build a statistical model. The models key parameters must be set to Gaussian distribution as follows:

```
.param
+p1=AGAUSS(0.37,0.01,1)
.param
+p2=GAUSS(1.21038E-1,0.03,1)
......
+vth0=p1
+u0=p2
......
```

To begin, first load a compact model into the statistical module. Since no Monte Carlo analysis has yet been set, you will get only one point as a simulation output on each plot. MBP supports the ability to load the statistical model with a global or binning core model, or a macro model. It also allows the statistical model to be loaded directly from the model library using Lib Parser.

Set Monte Carlo Count

You can set the Monte Carlo simulation number by right-clicking on the plot and selecting Set Monte Count. You must set an appropriate number here. Note that a large number may lead to a more accurate result, but will also require a longer simulation time.

Fast MC Simulation

MBP provides an option to run fast MC simulation. To achieve this, choose Simulation > Fast-MC from the main menu. This method speeds up the simulation using a unique methodology to deal with the statistical model.

BPV Extraction Flow

A demo case is used here to describe the steps required to run a statistical model extraction through the built-in flow.

Demo Files

The demo folder is `$MBPHOME\demo\Statistical\mosfet`. Here, `$MBPHOME` is the installation path of MBP.
There are a total of three files in this folder:

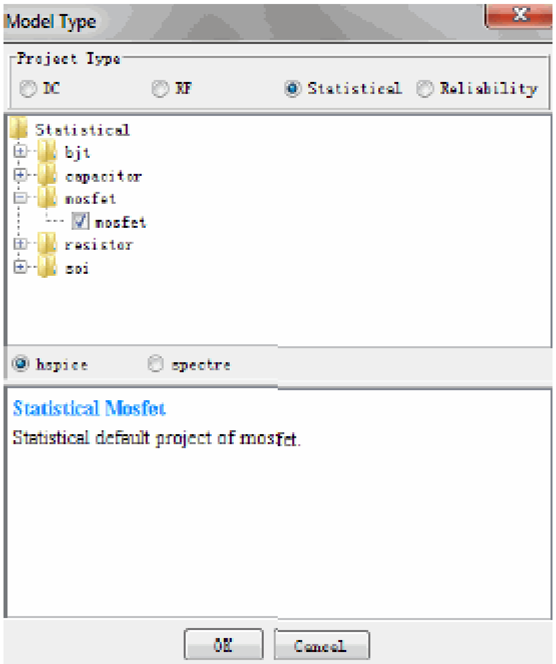
- model_nmos.l, the initial model card;
- param.txt, the parameter list used in the BPV extraction flow; and
- sta_data.mea, the demo data.

You can follow the steps presented below to complete the extraction process.

Set Model Type

First, You must set the mode type. Choose Model > Select Model from the main menu. In the popup Model Type window, choose Statistical as the project type. Then choose mosfet_ as the device type, as shown in following figure.

Model type

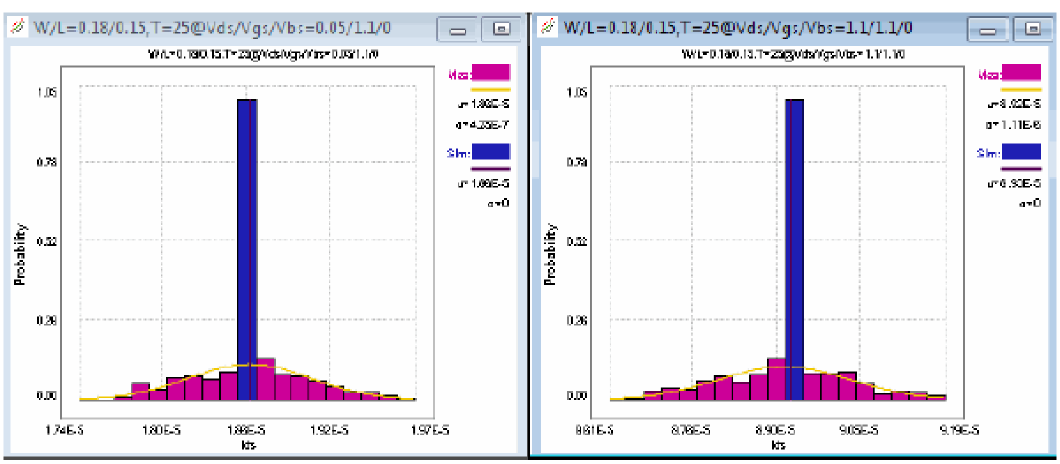


Click the OK button to close the window.

Load Data and Model

To load the data, choose File > Model > Load from the main menu. Load the model file model_nmosl. Since there is no statistical information in the initial model, only one blue column indicating the simulation result is visible (see following figure).

Initial model versus measurement data

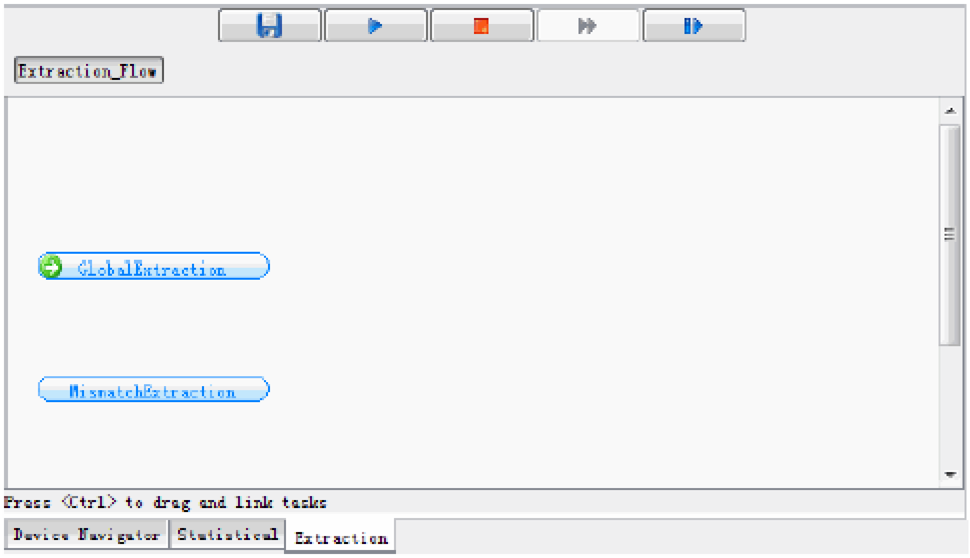


MBP also supports the ability to load a model with existing statistical information. You can use the flow to optimize the previously tuned model.

Run Extraction Flow

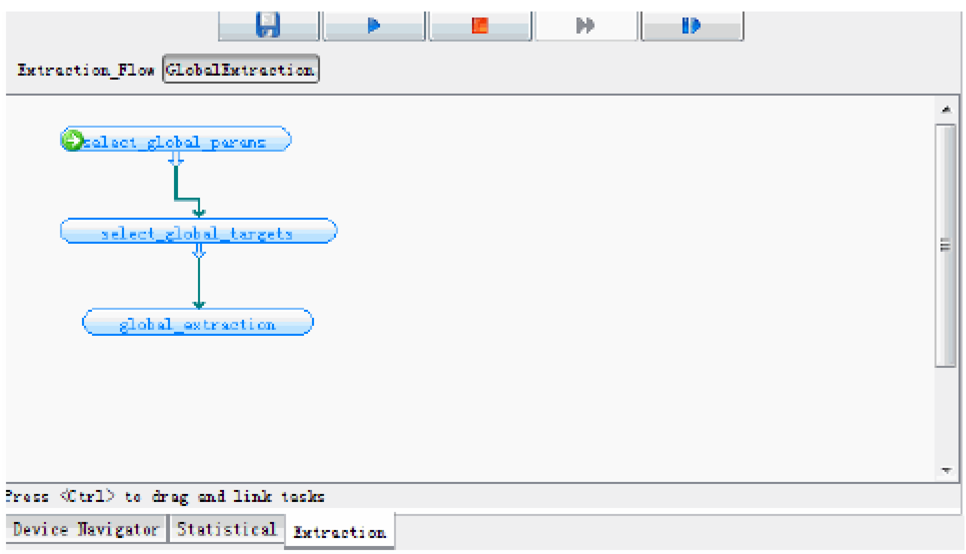
To run the extraction flow choose Extraction > Extraction Flow from the main menu. The extraction panel shown in following figure.

Extraction flow panel



Double click the Global Extraction button to expand the flow as shown in following figure.

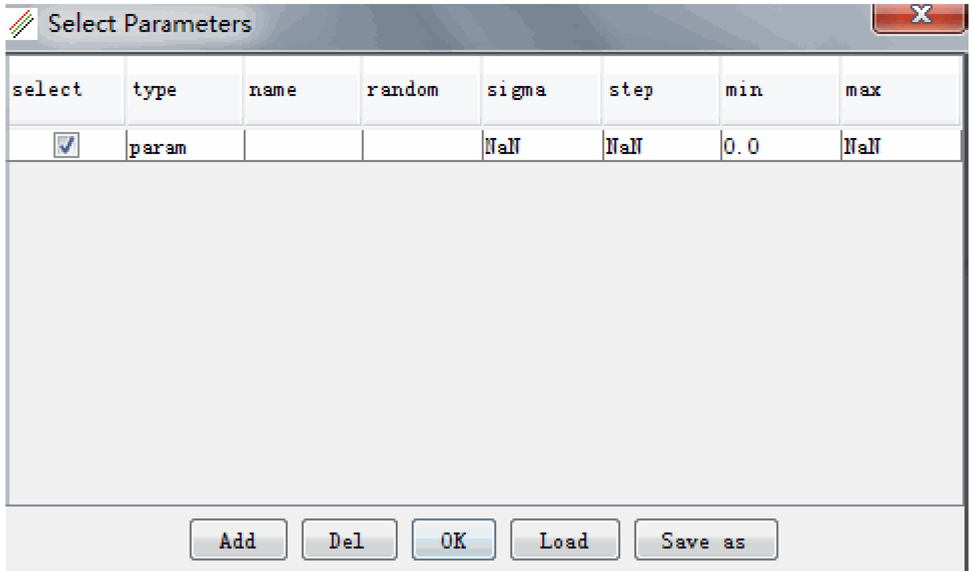
Global extraction flow



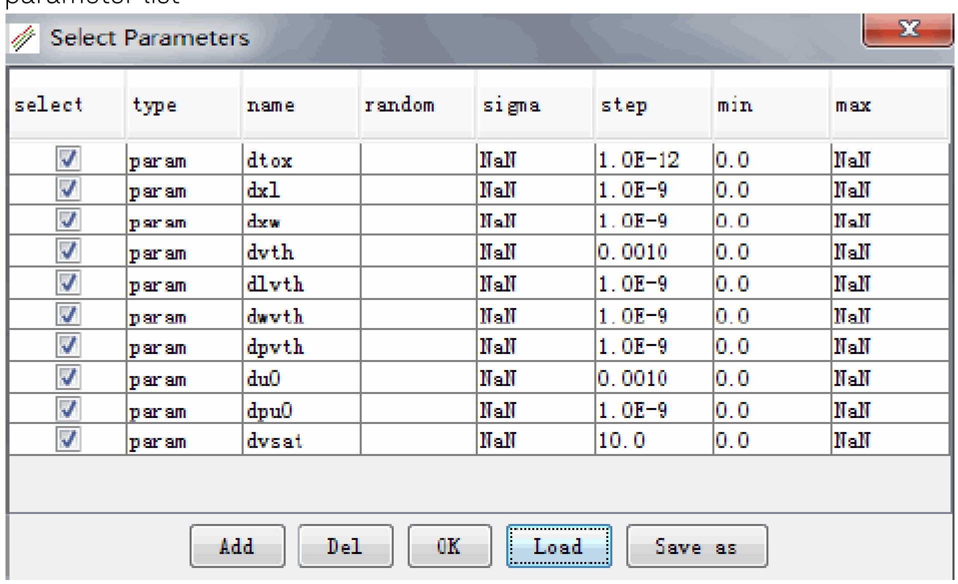
There are three steps in the flow:

- select_global_params,
- select_global_targets and global_extraction.

- Click the run icon to automatically run the global extraction flow. The Select Parameters window will pop up as shown in Figure: Select Parameters window Select Parameters window



Click the Load button to load the parameter list file param.txt. The parameters are arranged as shown in Figure-Load parameter list. Load parameter list



The parameter list window is now visible and can be used to select parameters for statistical calculation.

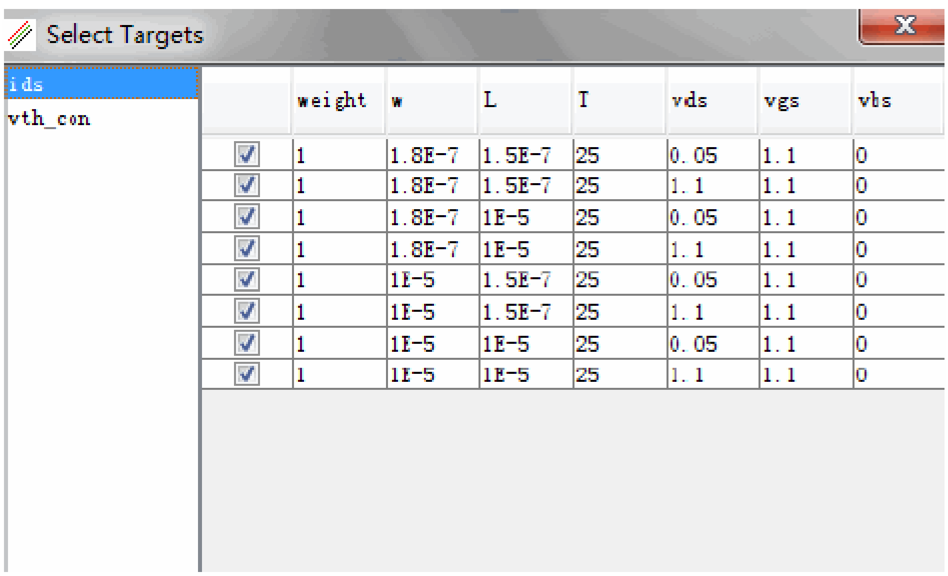
Some comments on the column names are follows:

- Select. When it is checked, the parameter will be re-extracted. If it is unchecked, the parameter will depend on the sigma value. If a sigma value is given, the final value of the parameter will be the sigma value; while if a sigma value is not given, the current value of the parameter will remain as the final one.
- Name: parameter name. When select is unchecked, the parameter name may be blank. At the same time, both random and sigma must have correct values.

- Random: random variable name. If there is a random variable for the parameter in the current model file, you must enter the name of the random variable here. If there is no random variable for the parameter in the current model file, you may either input a new name here
  or keep it blank. In the latter case, a new name will be automatically created. Note that not all names of random variables can be repeated.
- Sigma: sigma of the parameter. An example here is the dtox parameter. If you know the sigma of the TOX (normally the information can be obtained from fabrication), then the value can be input into it directly. The extraction flow then bypasses this parameter and uses the predefined value instead.
- Step: the step for BPV calculation. The step is very important for the extraction flow and is used to calculate the sensitivity of the parameter to the target. You must input a suitable value; one that is not too big or too small. Here, you can carefully monitor the change in IdVd or IdVg plots while adjusting the step number.

Click the OK button to continue. The Select Targets window will pop up as shown in following figure. You must then select the targets for the BPV extraction. Normally, the user will select the targets that are important or you care most about it.

Select Targets



	weight	w	L	T	vds	vgs	vbs
☑	1	1.8E-7	1.5E-7	25	0.05	1.1	0
☑	1	1.8E-7	1.5E-7	25	1.1	1.1	0
☑	1	1.8E-7	1E-5	25	0.05	1.1	0
☑	1	1.8E-7	1E-5	25	1.1	1.1	0
☑	1	1E-5	1.5E-7	25	0.05	1.1	0
☑	1	1E-5	1.5E-7	25	1.1	1.1	0
☑	1	1E-5	1E-5	25	0.05	1.1	0
☑	1	1E-5	1E-5	25	1.1	1.1	0

Select targets Next, you can set different weights for different targets. The default value is 1. Close the window to continue. In the last step, the Save dialog window will pop up. Here, the user inputs a name to save the model file. You will then find that the following statistical parameters have been extracted and added into the generated model file:

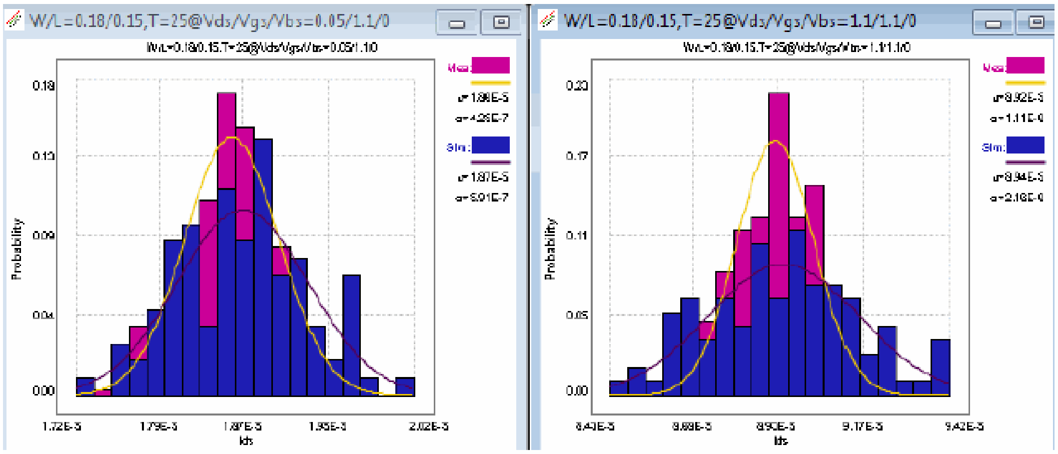
```
.param||

+s_dtox = 4.759276E-14

s_dxl = 2.757199E-13
```

```
s_dxw = 3.874035E-9

+s_dvth = 1.16912E-2

s_dlvth = 8.878411E-7

dwvth = 3.014865E-7

+s_dpvth = 3.740674E-9

s_du0 = 1.47506E-3

s_dpu0 = 2.144652E-8

+s_dvsat = 2.016329


.param

+dtox = 0.0+s_dtox*random1*

dxl = 0.0+s_dxl*random2*

dxw = 0.0+s_dxw*random3*

+dvth = 0.0+s_dvth*random4*

dlvth = 0.0+s_dlvth*random5*

dwvth = 0.0+s_dwvth*random6*

+dpvth = 0.0+s_dpvth*random7* du0 = 0.0+s_du0*random8*
dpu0 = 0.0+s_dpu0*random9*

+dvsat = 0.0+s_dvsat*random10*

.param

+random1 = agauss(0.0,1.0, 1)

+random2 = agauss(0.0,1.0, 1)

+random3 = agauss(0.0,1.0, 1)

+random4 = agauss(0.0,1.0, 1)

+random5 = agauss(0.0,1.0, 1)

+random6 = agauss(0.0,1.0, 1)

+random7 = agauss(0.0,1.0, 1)
```

```
+random8 = agauss(0.0,1.0, 1)

+random9 = agauss(0.0,1.0, 1)

+random10 = agauss(0.0,1.0, 1)
```

The fitting result is shown in following figure.

Fitting result

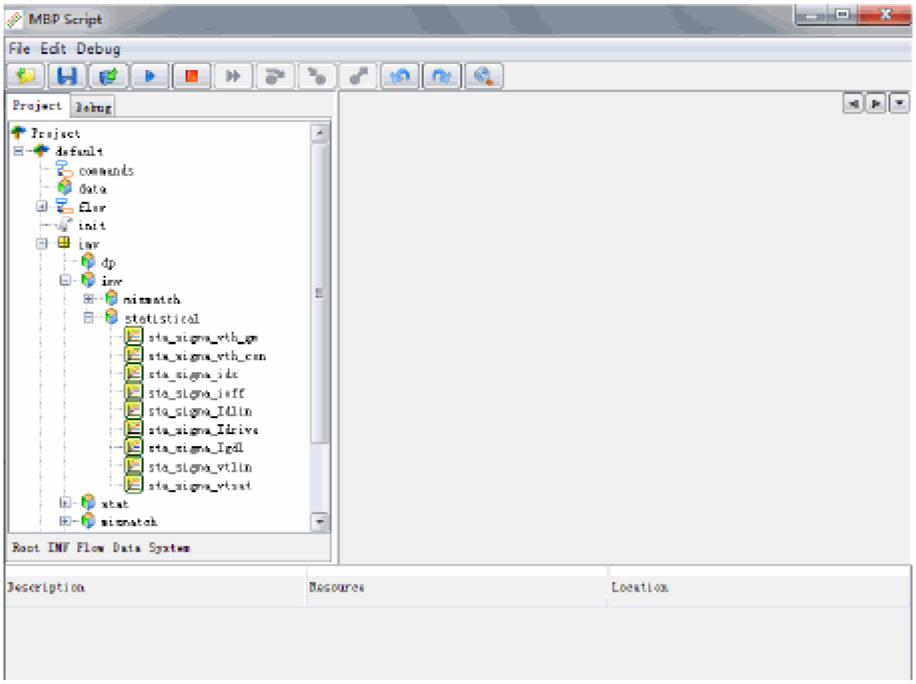


You may continue to fine tune the parameters manually, or simply modify the step number as previously mentioned and rerun the flow until a satisfactory result is obtained.

Statistical IMV

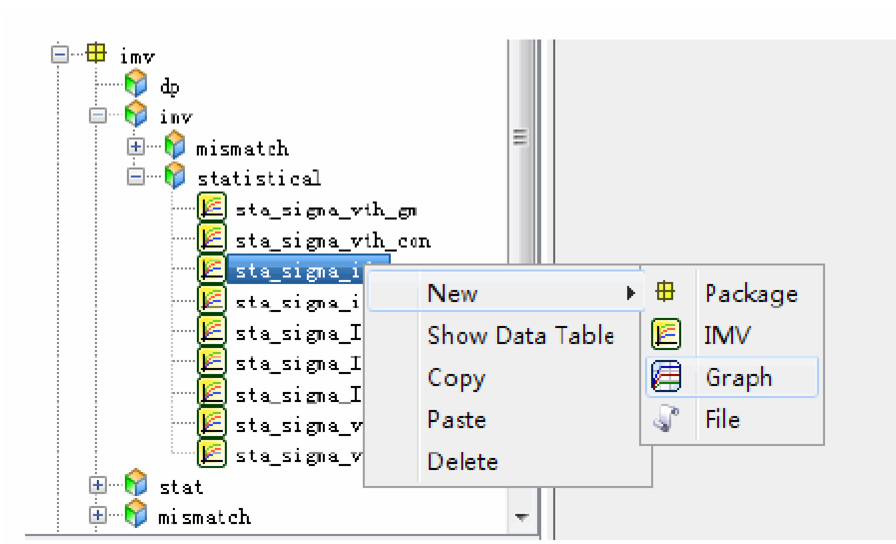
Finally, we use a demo to illuminate how to customize statistical IMV and plot it in MBP. After loading the demo data, choose Script > Script Project from the main menu to pop up the MBP Script interface. In the left Project tab window, click default > imv > imv > statistical to expand the file structure, as shown in Figure MBP Script.

MBP Script



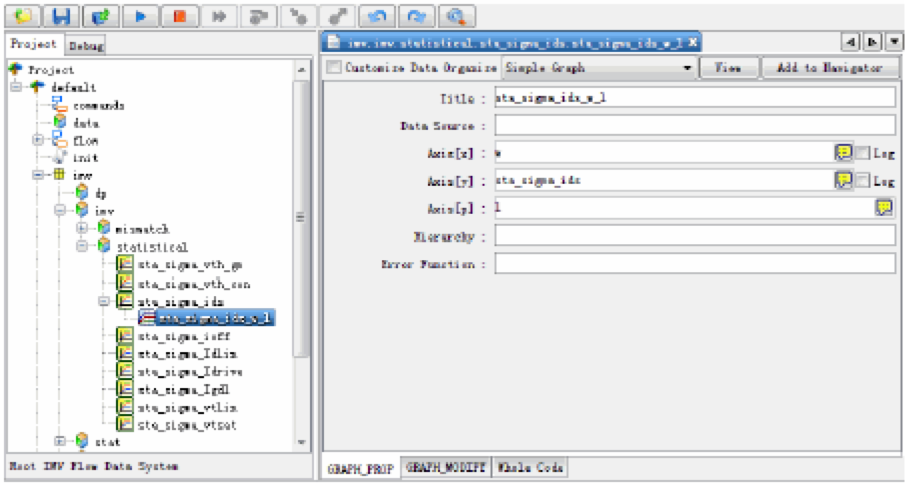
Right click the IMV sta_sigma_ids and choose New > Graph(see following figure).

Create new graph



Input sta_sigma_ids_w_l as the code name. In the GRAPH_PROP tab, choose w as Axis[x], sta_sigma_ids as Axis[x] and l as Axis[x],as shown in see following figure.

Configure graph

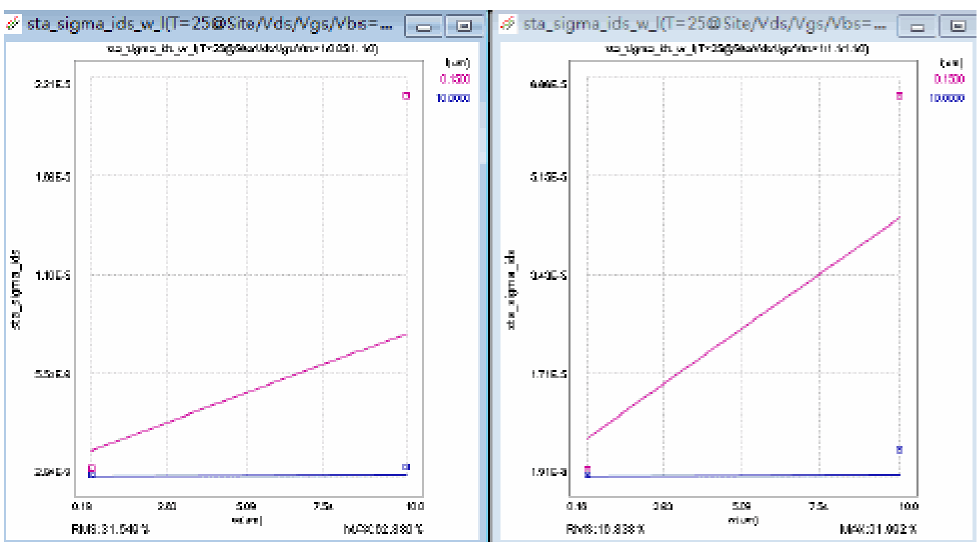


Click the icon  to save the current code. In the main menu of MBP, choose **Extraction** > **IMV** > **IMV Pages** to open the IMV page.

Click the icon  to refresh.

Then, you can view the customized IMV page (sta_sigma_ids_w_l) as shown in following figure.

IMV pages



Tmi Aging Model Application

TMI Aging Model Application

This application note describes how to implement a TSMC Modeling Interface (TMI) aging model in Model Builder Program (MBP).

NOTE This document was originally released for MBP V2011.1.0 in August 2011.

Overview

TMI or TSMC Modeling Interface is a C-based modeling application programming interface (API) developed to support extensions of standard compact models. TMI is an add-on to standard models. Therefore, the compatibility and robustness of standard models dictated by the Compact Model Council (CMC) is retained when applied to different simulators and platforms.

MBP V2011.1.0 offers an environment for TMI reliability aging model simulation. With it, you can measure the device performance degradation over time and evaluate the effects of stress.

This document provides information on how to run a TMI aging model simulation and optimization in MBP. For more information go to http://www.keysight.com/find/eesof or contact your local Keysight office. The complete list is available at http://www.keysight.com/find/contactus.

Preparation

In preparation for using the TMI aging model application, all configuration files regarding TMI models must be placed under `$MBP_HOME\etc\hspice\tmi` where `$MBP_HOME` is the MBP installation path. You need to specify one directory as the TMI library path (`TMIPath`) on the computer. Then, a subfolder under `TMIPath` must be created to store the compiled, shared library file (`libTMImodel.dll_`) according to different operating system (OS) platforms. For example, if the OS is a 32-bit version of Microsoft ® Windows ® , the library file should be placed under `TMIPath/WIN/.`

See following figure for more information on the directory structure for shared libraries of different platforms.

Directory structure for shared libraries of different platforms

```
TMIPath
 — RH/ libTMImodel.so (RedHat LINUX 32bit)
 — RH_64/ libTMImodel.so (RedHat LINUX 64bit)
 — SUN/ libTMImodel.so(SUNOS 32bit on SPARC)
 — SUN64/ libTMImodel.so (SUNOS 64bit on SPARC)
 — SUSE/ libTMImodel.so (SUSE 32bit)
 — SUSE_64/ libTMImodel.so (SUSE 64bit)
 — X86SUN/ libTMImodel.so (SUNOS 32bit on X86)
 — X86SUN_64/ libTMImodel.so (SUNOS 64bit on X86)
 — WIN/ libTMImodel.dll (MS Windows 32bit)
 — WIN_64/ libTMImodel.dll (MS Windows 64bit)
```

In MBP, the data format for TMI aging analysis is the same as the MOS Reliability Analysis (MOSRA) data format. For more details, refer to the *MBP user guide* or *The MOSRA Model Support in MBP Application Note*.

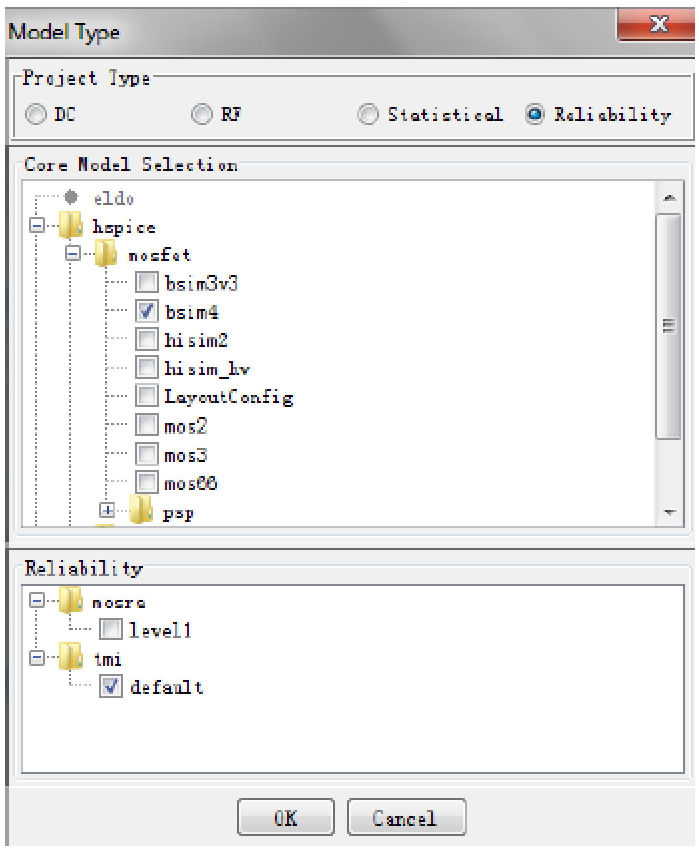
Simulation and Parameter Extraction

MBP invokes the external simulator (Synopsys HSPICE) for TMI aging model simulation. You must ensure HSPICE has been installed properly prior to the simulation. Then, the following steps must be followed.

Choose Model Type

Choose Model > Select Model from the main menu and select Reliability in the Project Type. Then, select one core model (bsim4 in this example) in the upper Core Model Selection section and tmi-default in the lower Reliability section, as shown in following figure.

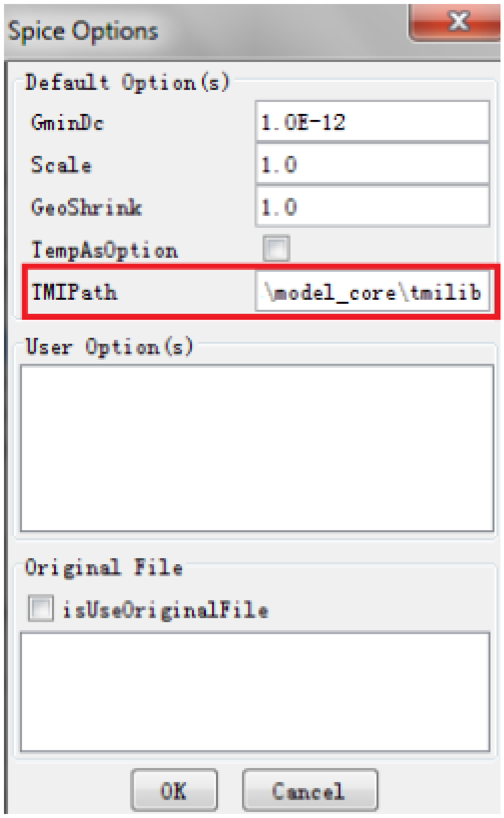
Choose model type



Set TMI library path

Choose Extraction > Options > SPICE Options from the main menu to set the TMI library path (&`TMIPath`) in the Spice Options window, as shown in following figure.

TMI path setting

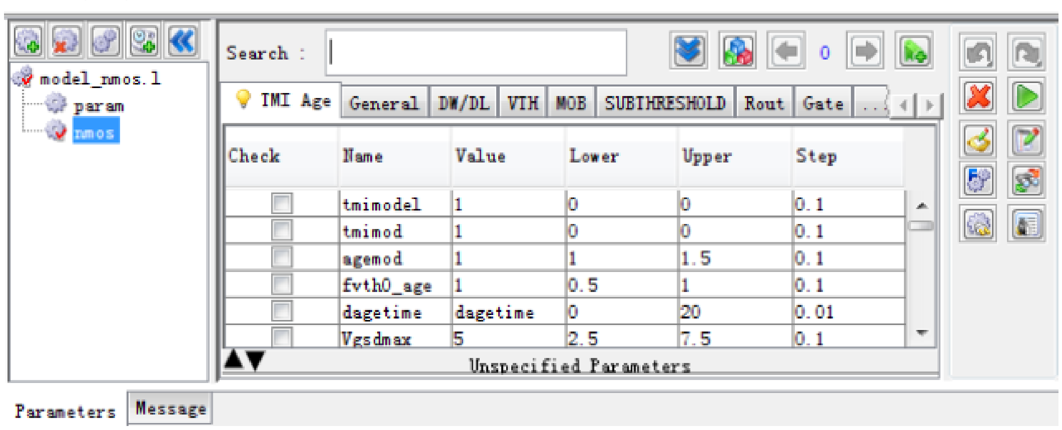


In this example, we set `~\etc\hspice\tmi\bsim4\default\model_core\tmilib` as the `TMIPath`_ Finally, click OK.

Model Parameters Panel

As shown in following figure, the TMI aging parameters are sorted out in the TMI Age tab of the model card.

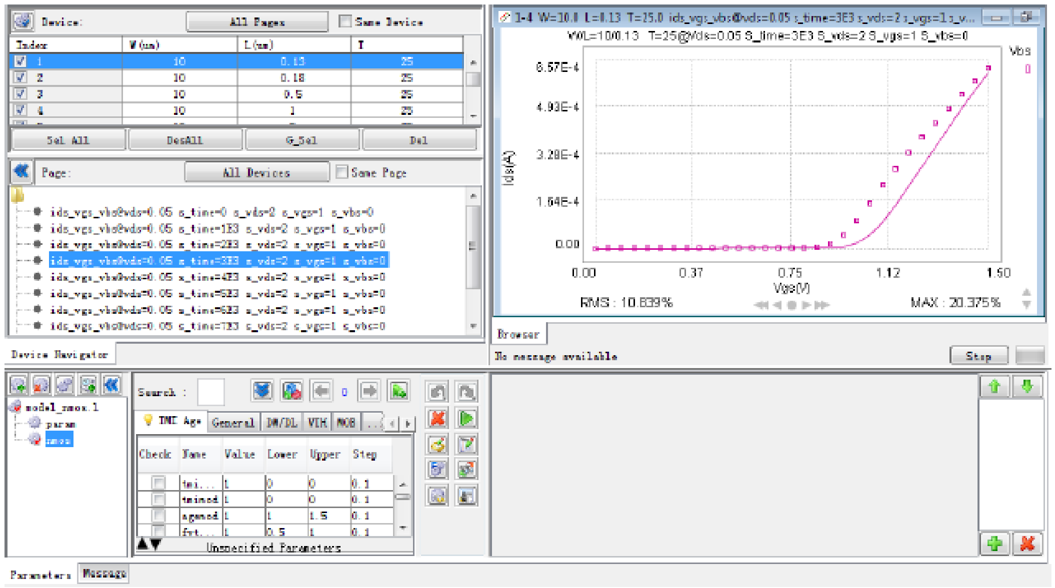
TMI aging parameters



Load Model and Data

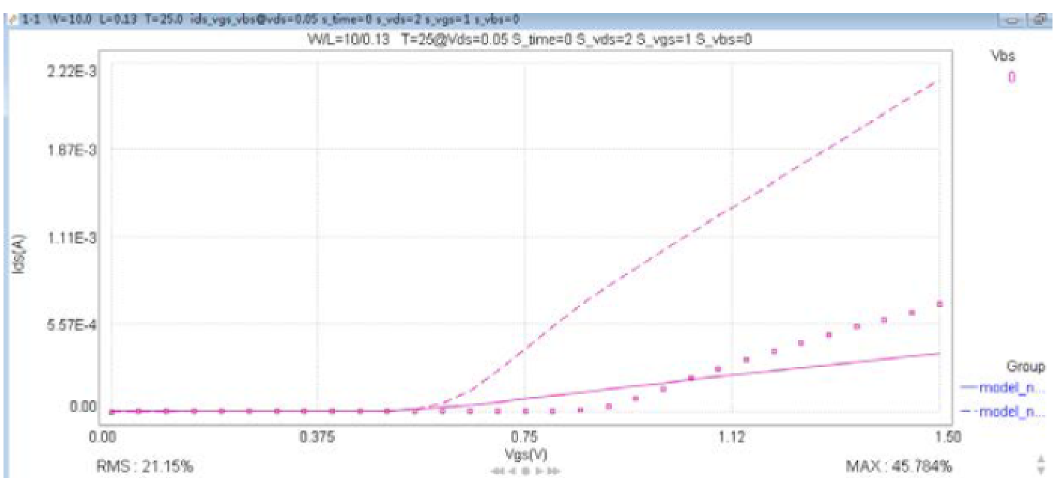
In the main menu, choose File > Model> Load to load the model file. Next, choose File > Data > Load from the main menu to load the data file. The window with the TMI aging model and data is shown in following figure.

Load model and data



You can now select model parameters and adjust them to fit the measurement data. MBP also allows to compare two different models. To do so, simply click the Add icon  to append another model for comparison. Select the two models by pressing the Ctrl button and clicking Compare icon  to compare them. The resulting window is shown in following figure.

Compare two models



Microsoft is a U.S. registered trademark of Microsoft Corporation. Windows and MS Windows are U.S. registered trademarks of Microsoft Corporation.

Transient Simulation

Transient Simulation

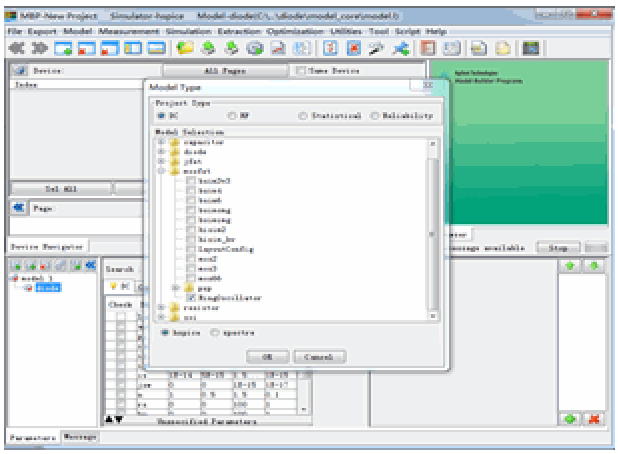
Overview

MBP Supports transient simulation for both internal engine and external simulator (hspice , spectre). And build the demo case for this application.

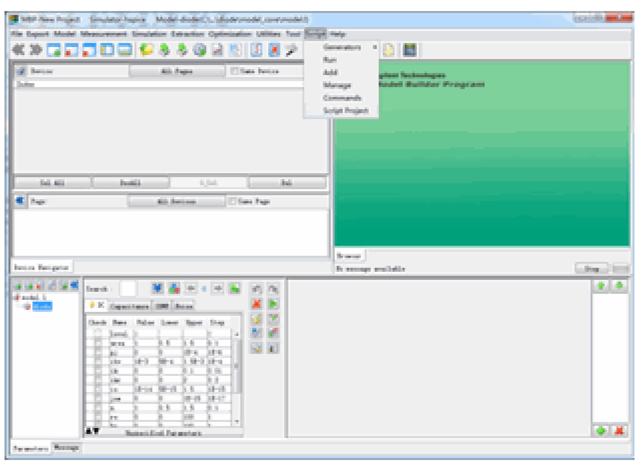
Ring oscillator demo case

MBP takes Ring oscillator as an example, which is a widely used test circuit.

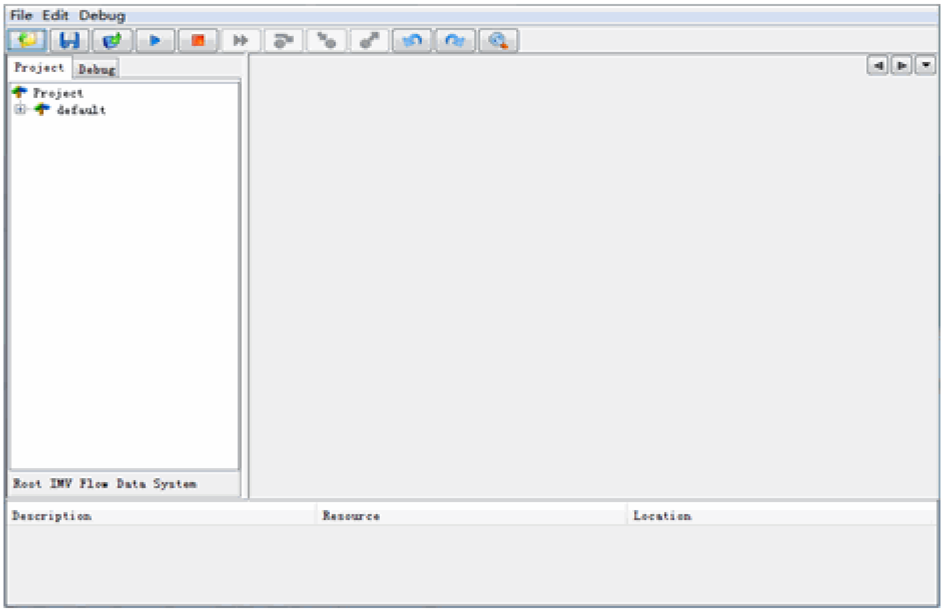
1. Load the model for demo\DC\mosfet\RO



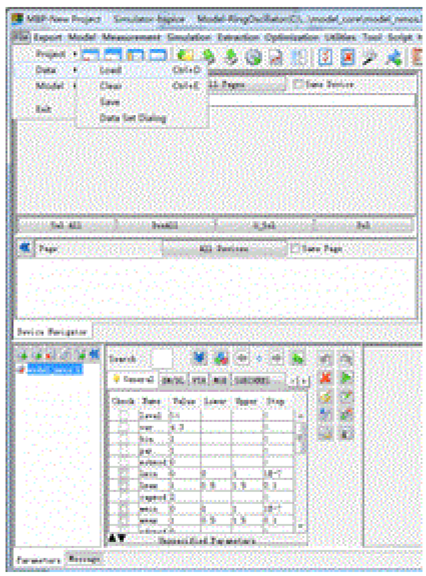
2. Load the script project.
   Select from script > script > project



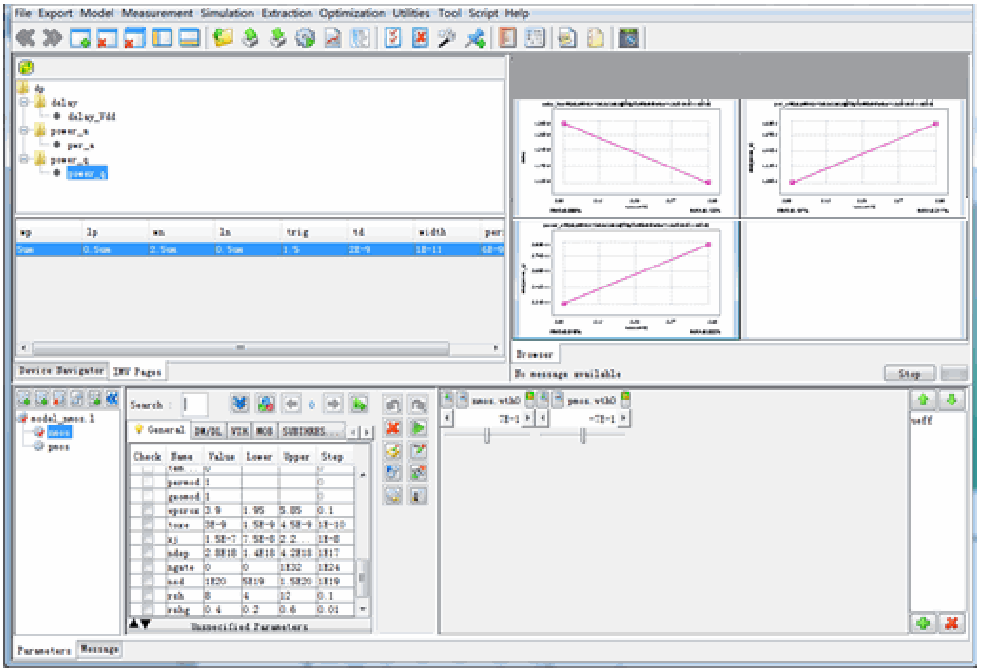
   Click the Open button, and load script project from demo\DC\mosfet\RO



3. Load measurement data from demo\DC\mosfet\RO



4. Check the interesting targets and tweak the parameter



PCA Model Extraction

PCA Model Extraction

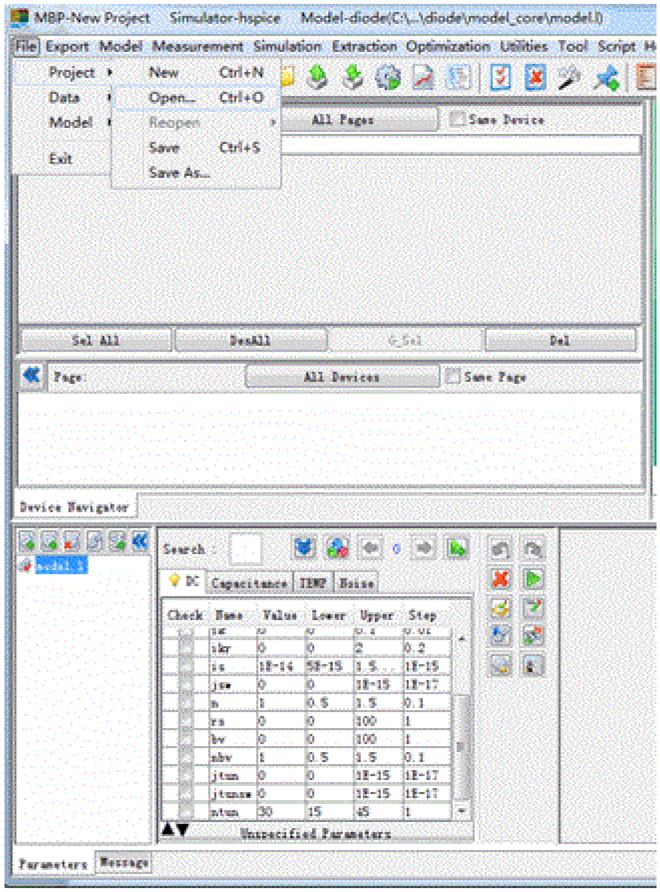
Overview

PCA (Principal Component Analysis), is a mathematical procedure that uses an orthogonal transformation to convert a set of observations of possibly correlated variables into a set of values of linearly uncorrelated variables called principal components. It uses major, independent components to describe the majority of variations of a larger number of characteristics and their correlation.
In MBP, the PCA flow has two steps:

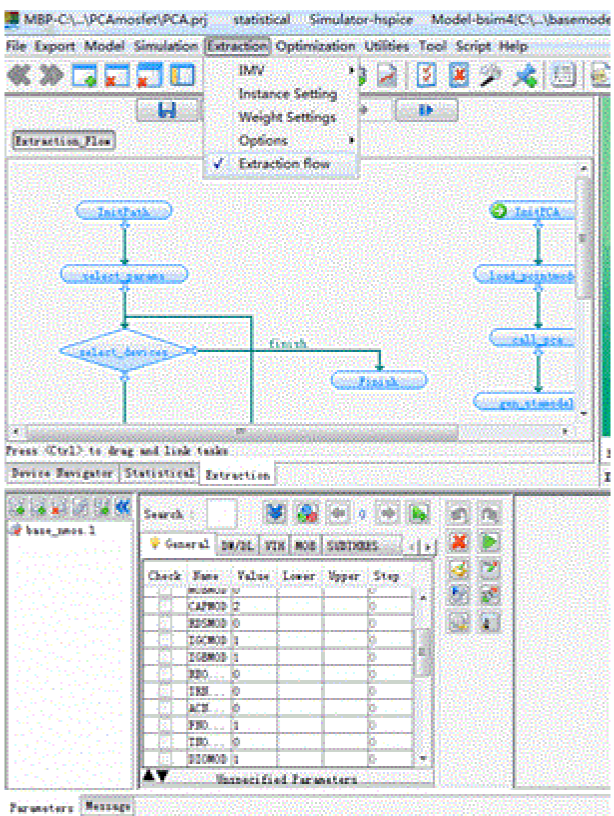
1. The first one extracts the point models from measurement data.
2. The second one generates the statistical model from the generated point models at last step.

Democase

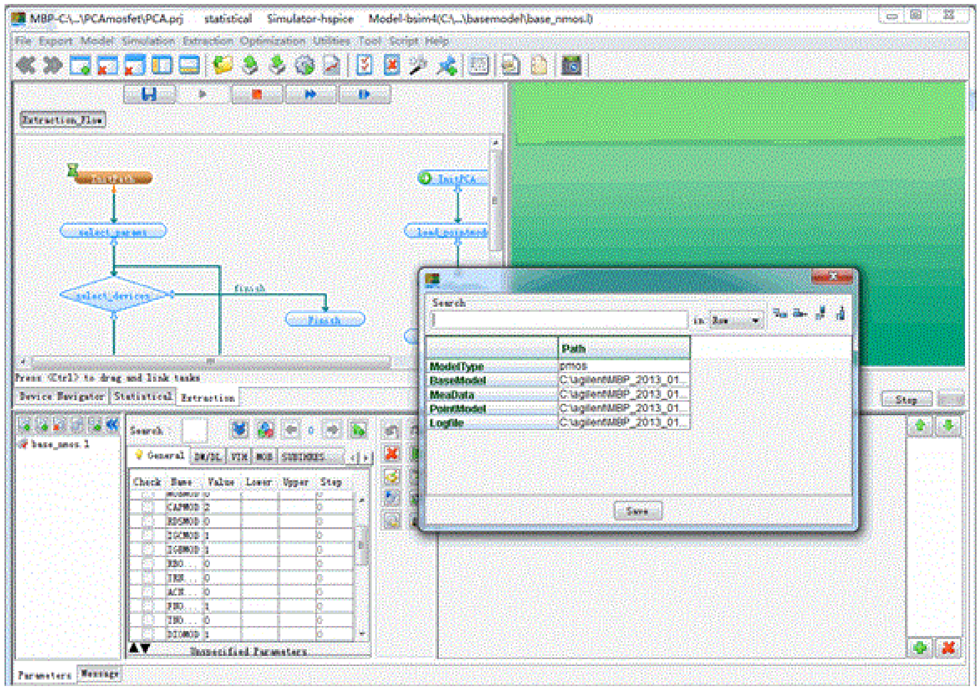
1. Open the project from \demo\Statistical\PCAmosfet



2. Load the task tree

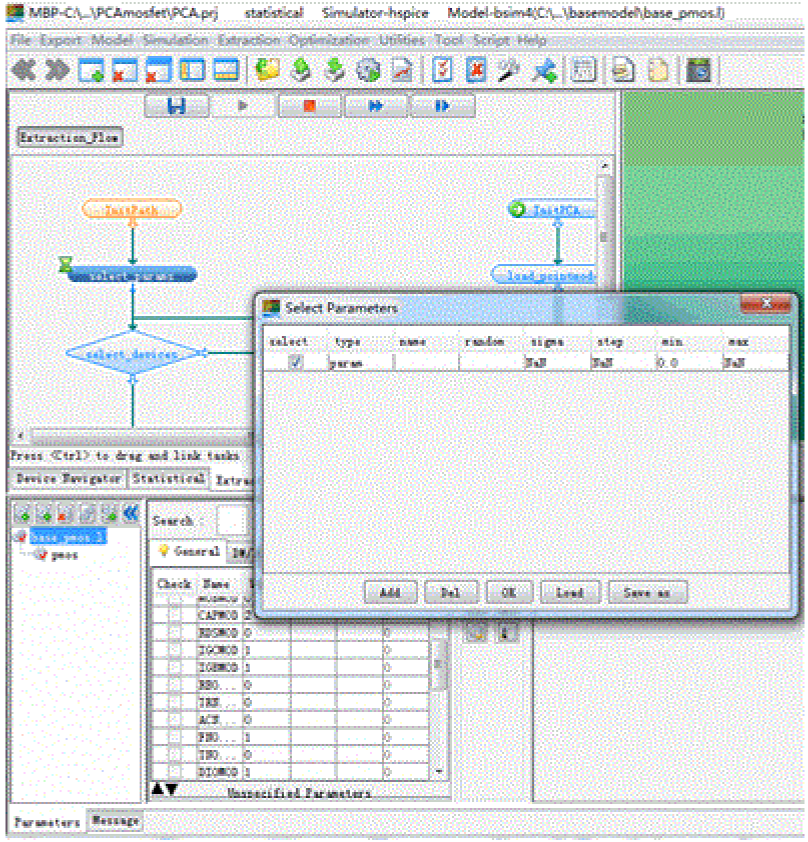


There are two task trees, the left one is the first step of PCA flow. It will extract the point models. And the right one is the second step of PCA flow. It will generate the statistical model from the point models.

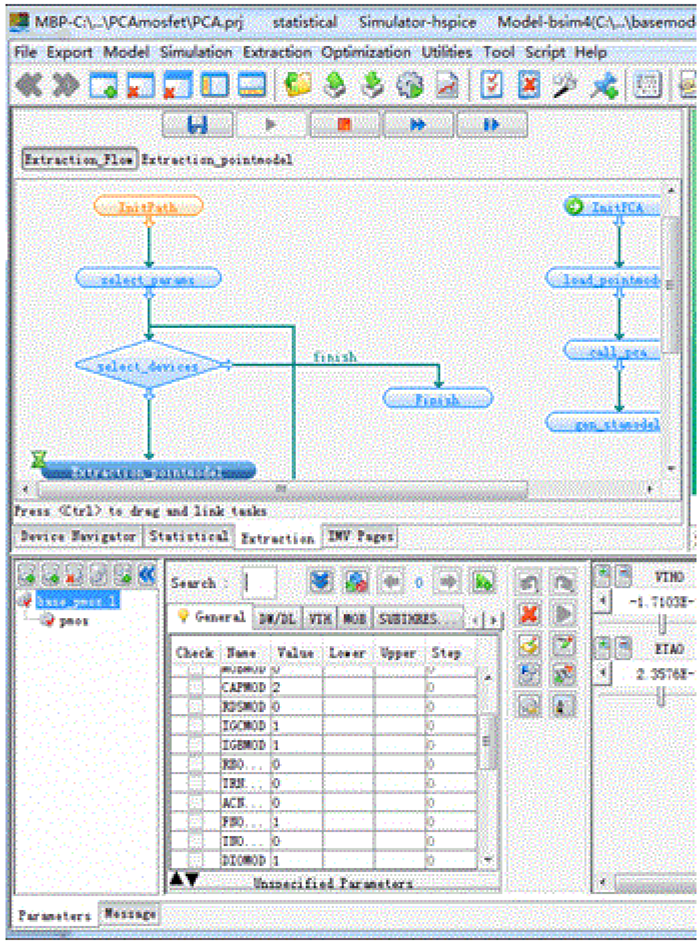


3. Initial Setting
   Select the InitPath , and click run button. An input table window will be popped.

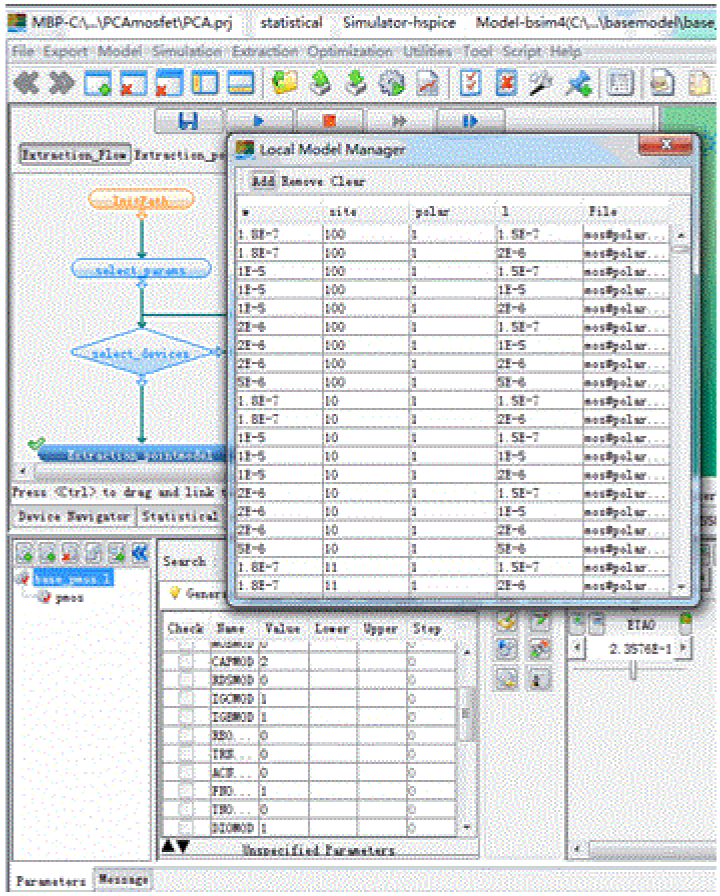
- Modeltype: pmos or nmos.
- BaseModel: The path of base model, the base model is the initial model for the point models extraction.
- MeaData: The path for measurement data.
- Pointmodel: The path for the point model saving.
- Logfile : The path for the log file saving.
  Close this window and the tree will run to next step(Parameter select).



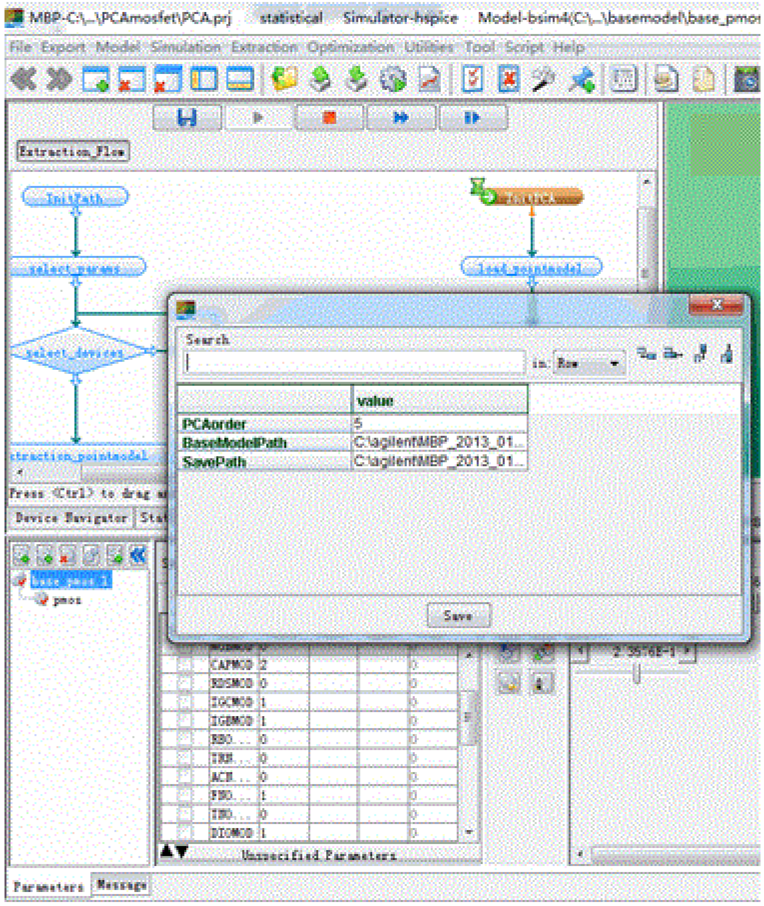
Click the Load button, and select the demo\Statistical\PCAmosfet\ opt_nparam.txt for nmos and demo\Statistical\PCAmosfet\ opt_pparam.txt for pmos (
point models extraction). The point models will be automatically generated, and all the point models will be saved to the specified fold. We can repeat this flow for nmos if necessary.



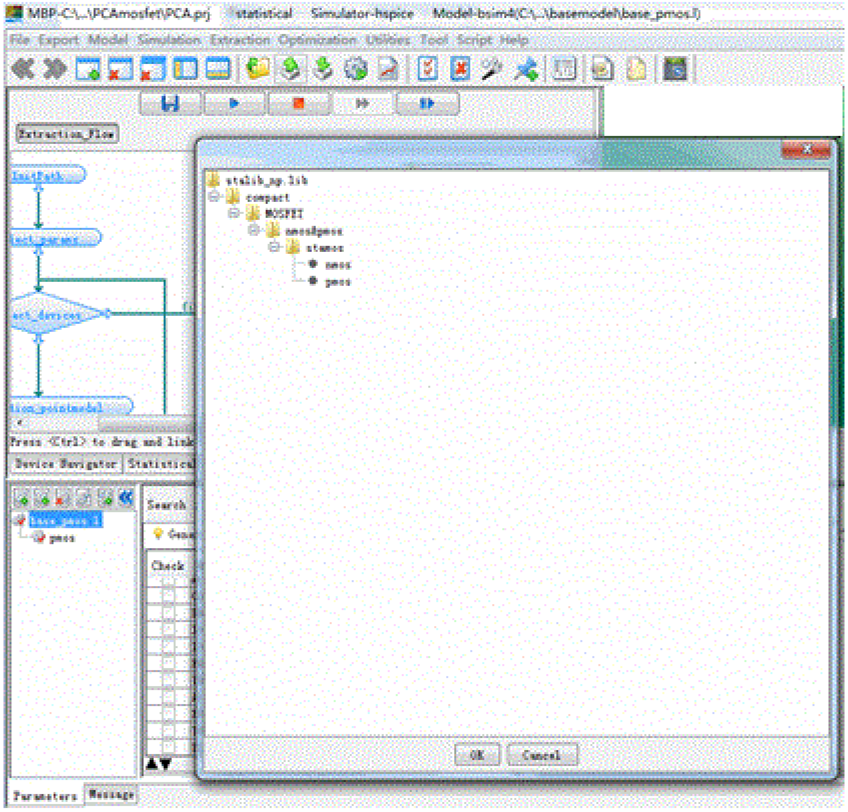
1. Load Point Model
   Select from model > Local model manager, and add all the points models to the table.



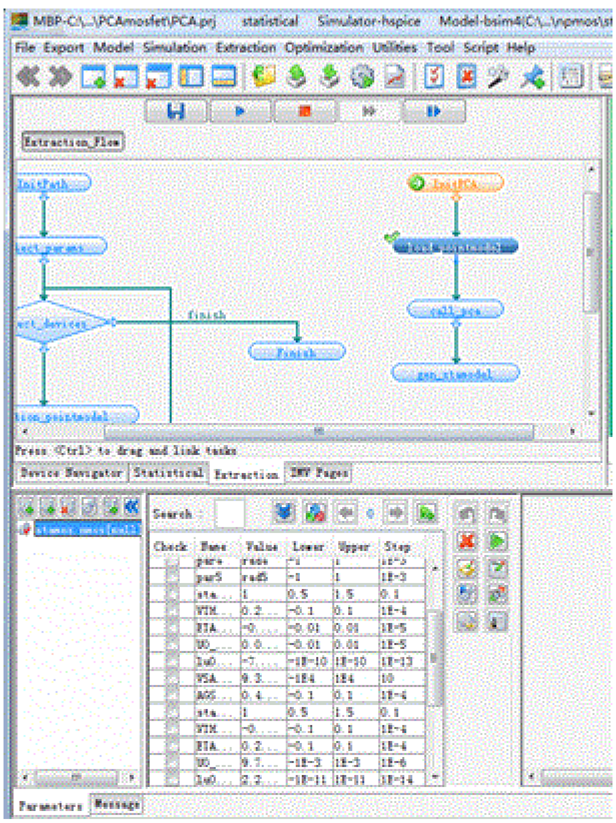
2. Initial setting for statistical model setting
   Select InitPCA and click run button, a input table window will be poped.
   PCAorder: The order-number for PCA algorithm.
   BaseModelPath: The path of base model, the base model is the initial model for the point models extraction.
   Savepath: The path for generated statistical model saving.



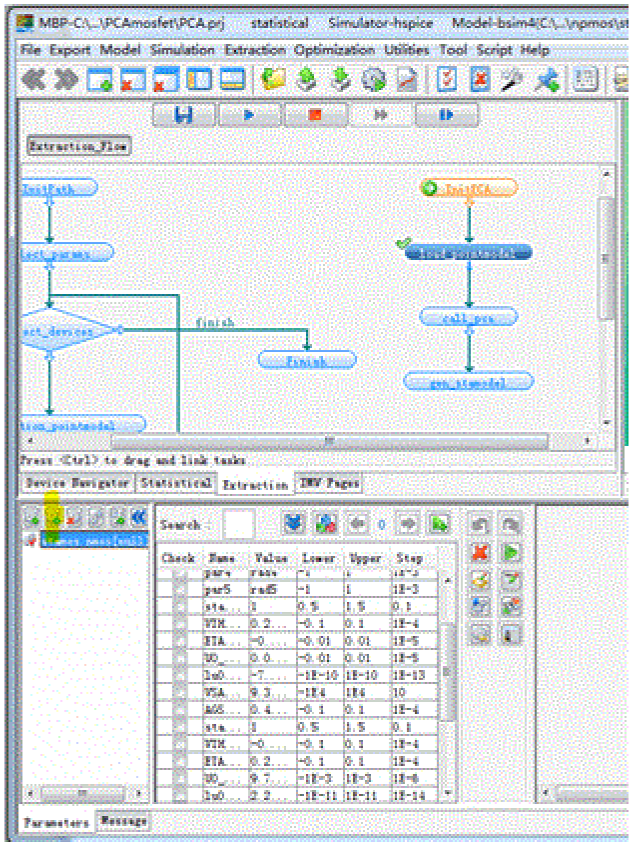
3. Load the generated statistical model
   A select window will be popped



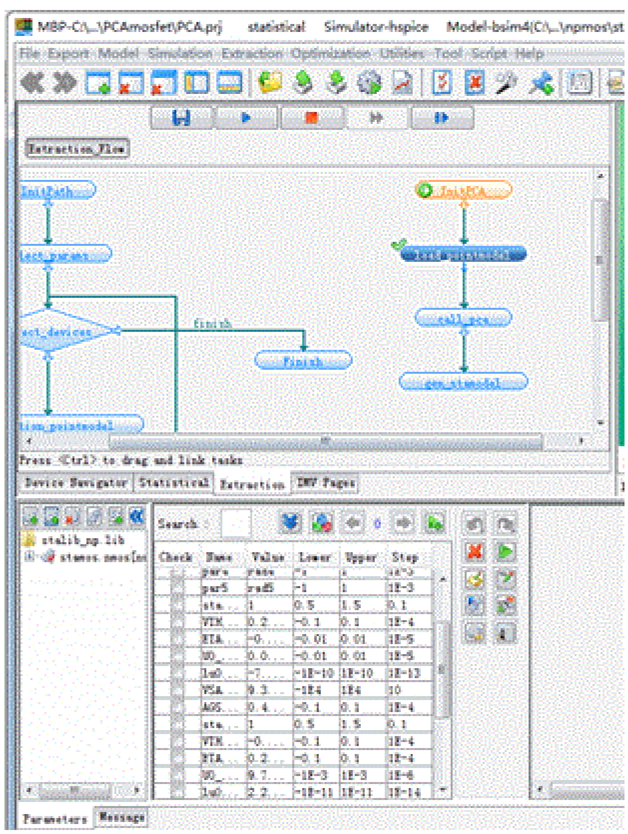
Select the model



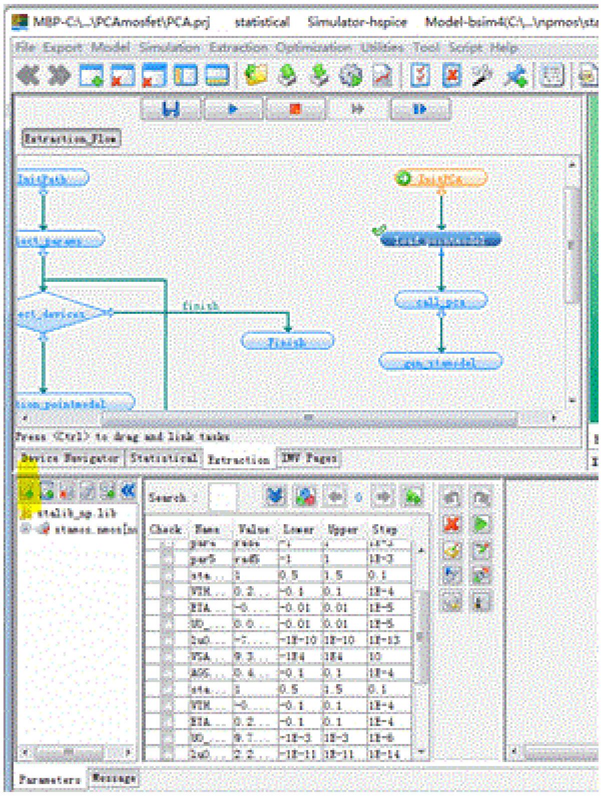
Click the add button

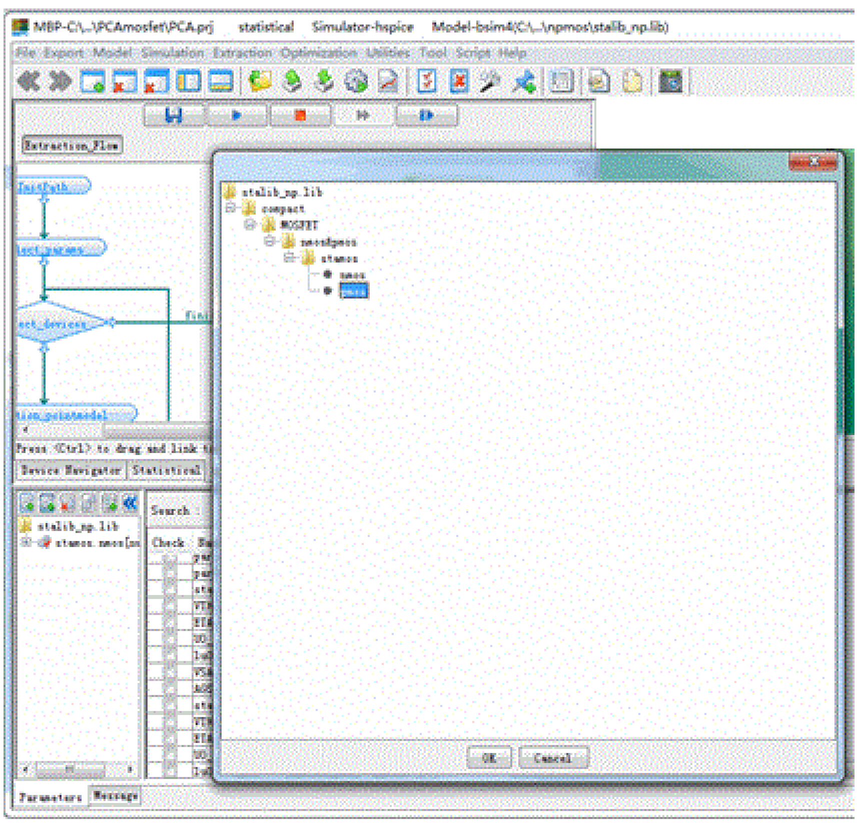


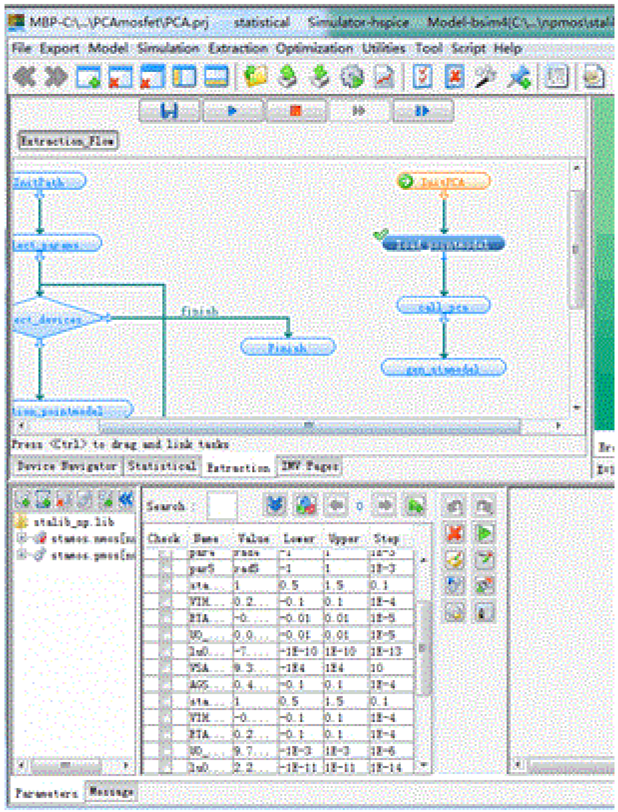
Add a group



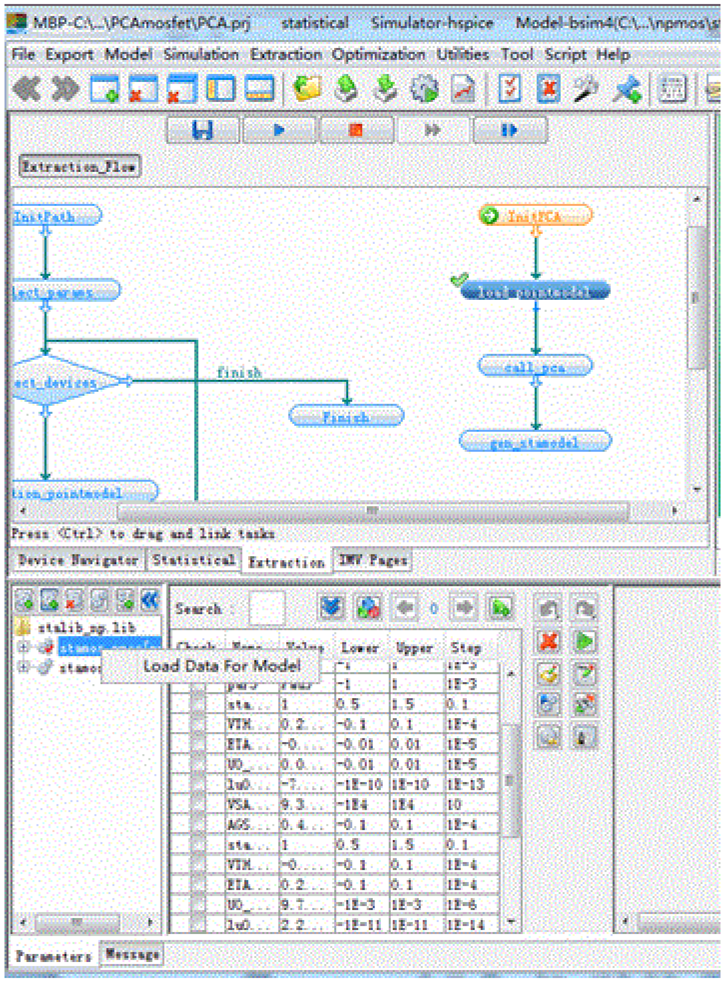
Add more model to the group



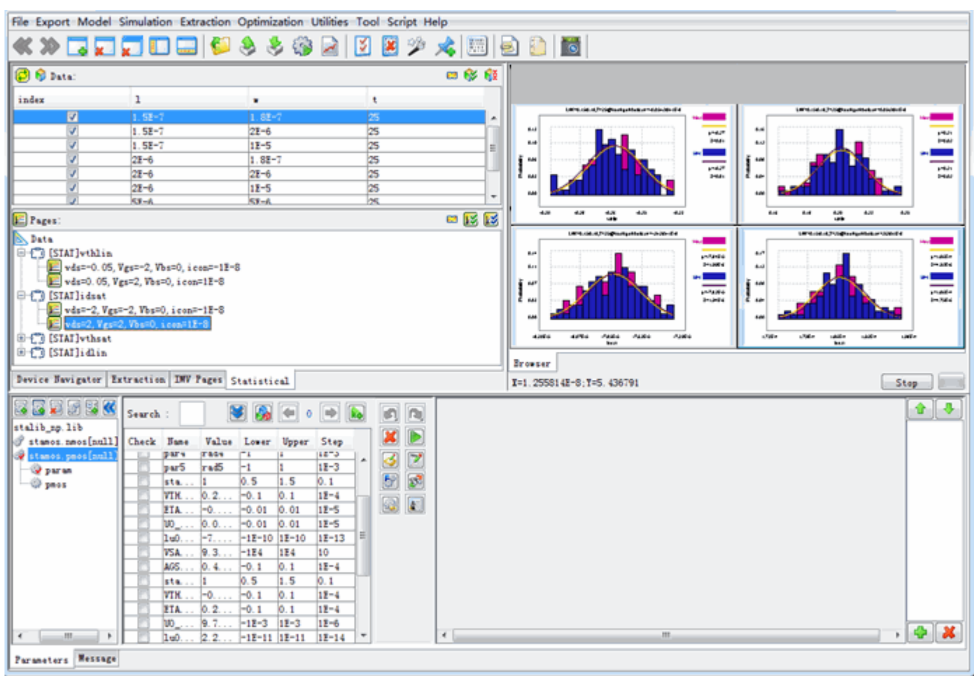




Select model, right click and load the measurement data for every model respectively.

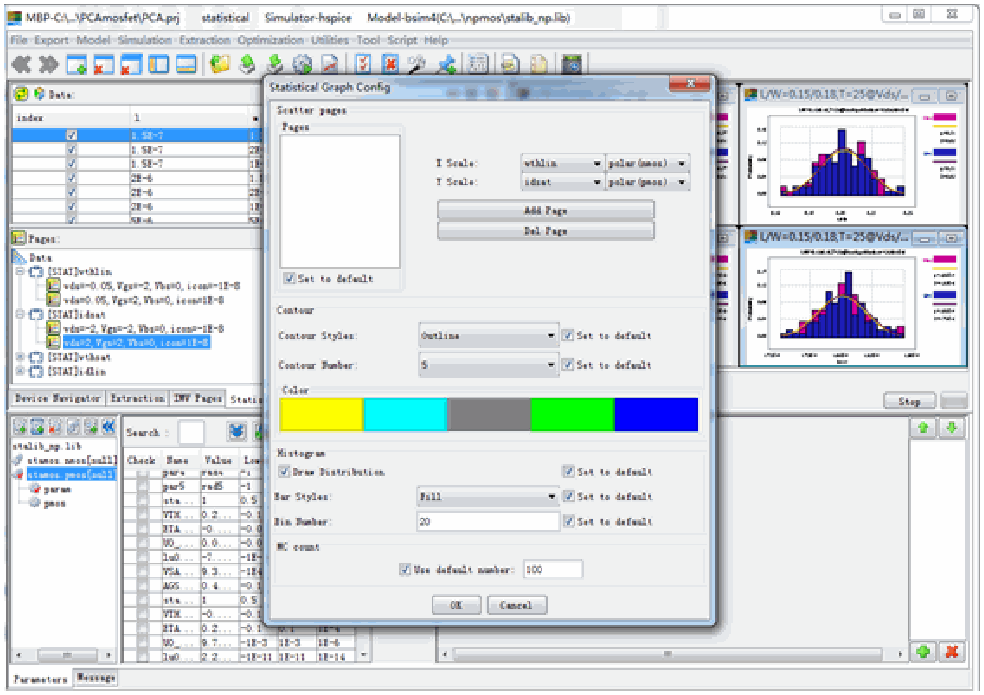


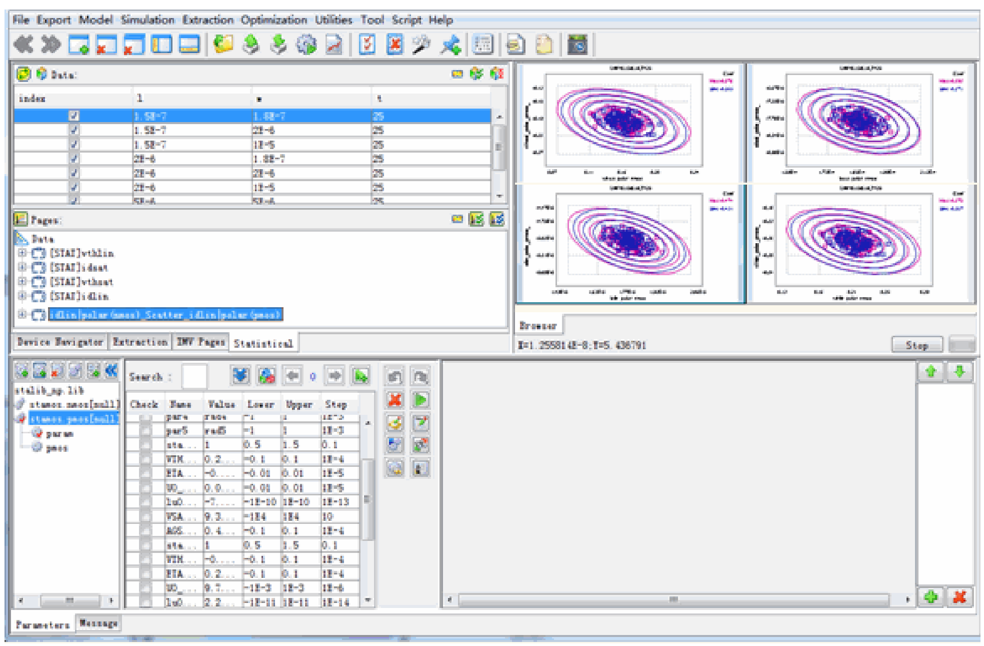
MBP-C:\...\PCAmosfet\PCA.prj statistical Simulator-hspice Model-bsim4(C:\...\npmos\s
File Export Model Simulation Extraction Optimization Utilities Tool Script Help
Extraction_Flow
InitPath
InitPCA
load_pointmodel
finish
Finish
call_pca
gen_stamodel
Press <Ctrl> to drag and link tasks
Device Navigator Statistical Extraction IMV Pages
Search
stalib_sp.lib
Load Data For Model
Check Name Value Lower Upper Step
Parameters Message



File Export Model Simulation Extraction Optimization Utilities Tool Script Help
Data:
Pages:
[STAT]vthlin
[STAT]idsat
[STAT]vthsat
[STAT]idlin
Device Navigator Extraction IMV Pages Statistical
Browser
Stop
Search
stalib_sp.lib
Check Name Value Lower Upper Step
Parameters Message

To see N,P correlation plot, go to Tool > GUI Options > Statistical Graph Config, user is able to choose targets for NMOS and PMOS, and then add the page.





Statistical Model Extraction Module

Introduction

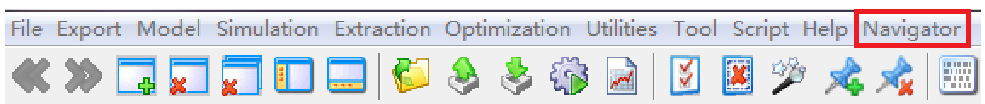
MBP's statistical model extraction module is fully developed for generating SPICE model with Monte Carlo simulation capability, a capability of both local mismatch and global statistical.

The module is designed to satisfy any input model format: both a single model card and a model library can be the start point of a statistical model extraction process, and the whole extraction progress is fully automatic, customizable.
It contains a comprehensive data dispose procedure which can separate the global variation from the total variation, and the re-centering feature allow user tweaking either the model or the data before go further into the statistical model extraction progress.
Final fitting result is very good, take not only the variation itself but also the correlation between NMOS and PMOS into account.

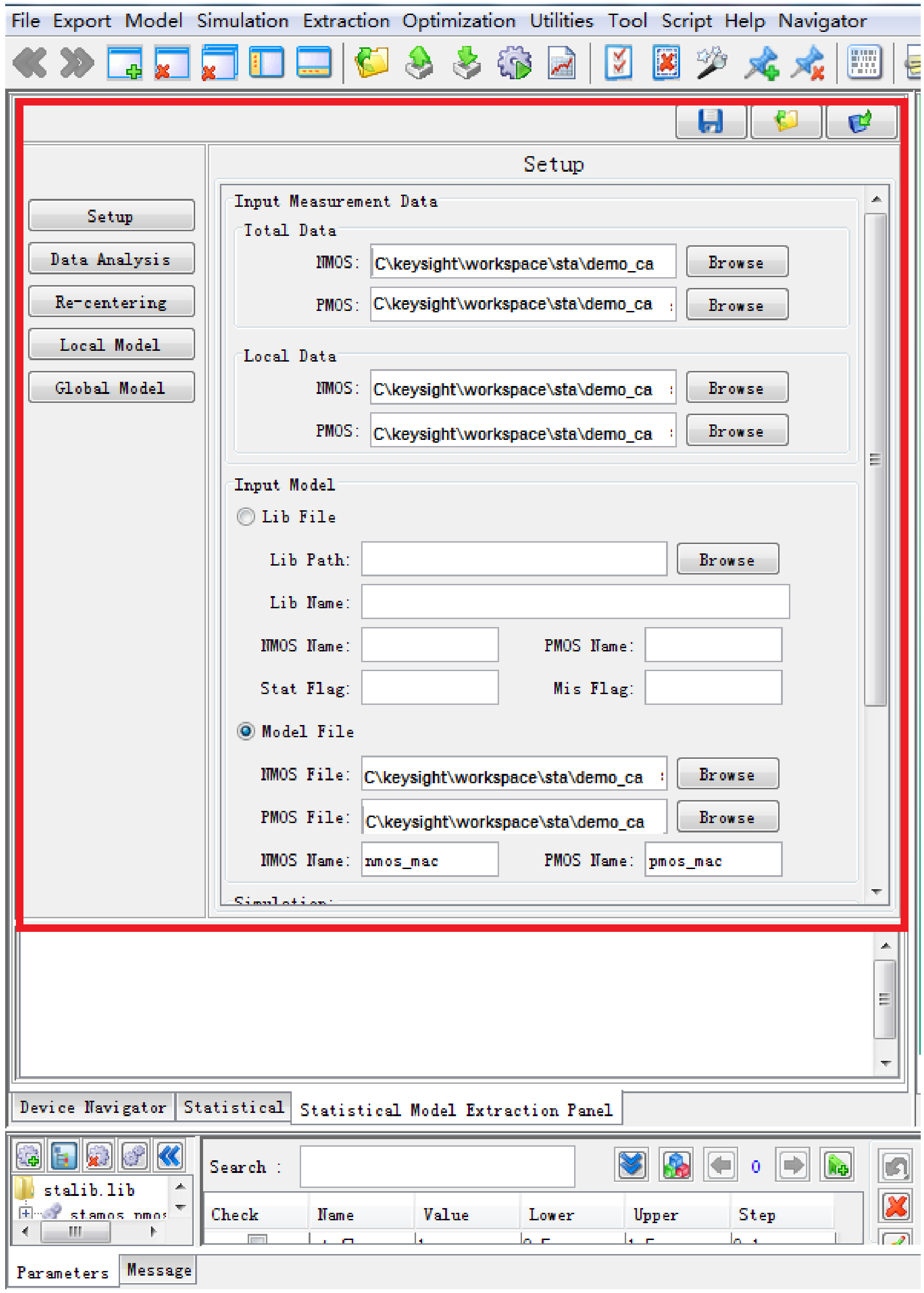
Start the statistical model extraction module

Follow below four steps:

1. Switch to MOSFET statistical project model Select Model Statistical mosfet.
2. Load the statistical modeling package under: MBP_HOME\demo\Statistical\GUIBasedSolution\mosfet\demo_case_baseban
3. A drop-down menu named "*Navigator*" appears in the menu bar then



4. Click the menu "Statistical Model Extraction Panel" in the drop-down menu to show/hide the Statistical Model Extraction Panel.



Setup the statistical model extraction progress

There are five buttons in the panel: *Setup*, *Data Analysis*, *Re-targeting*, *Local model* , *Global model*, Stand for the five main steps to complete a Statistical Model Extraction Progress.

Setup

User need setup the original data and model in follow four steps.

- Set up Raw Data Path
- Input Model Information

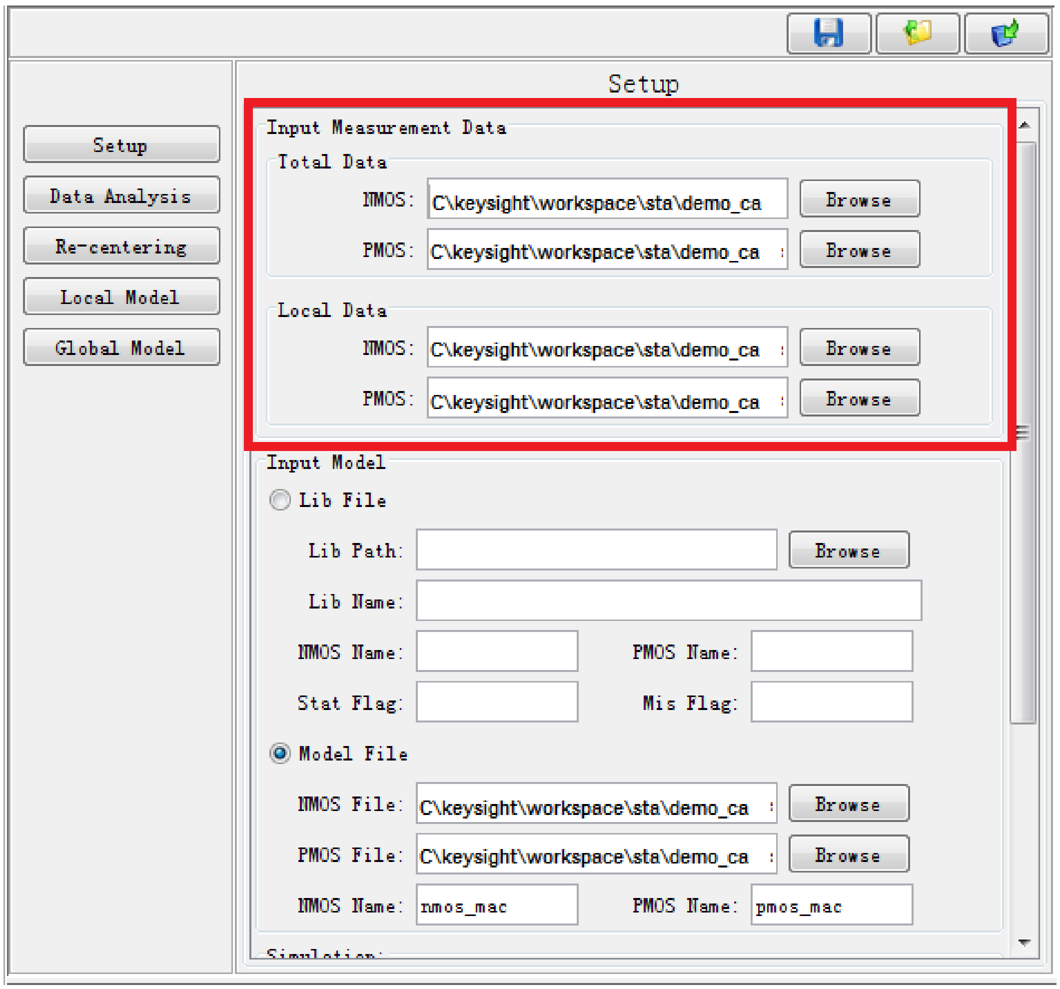
- Select Simulation Method
- Add Comments

Set up Raw Data Path

Two parts of this step:

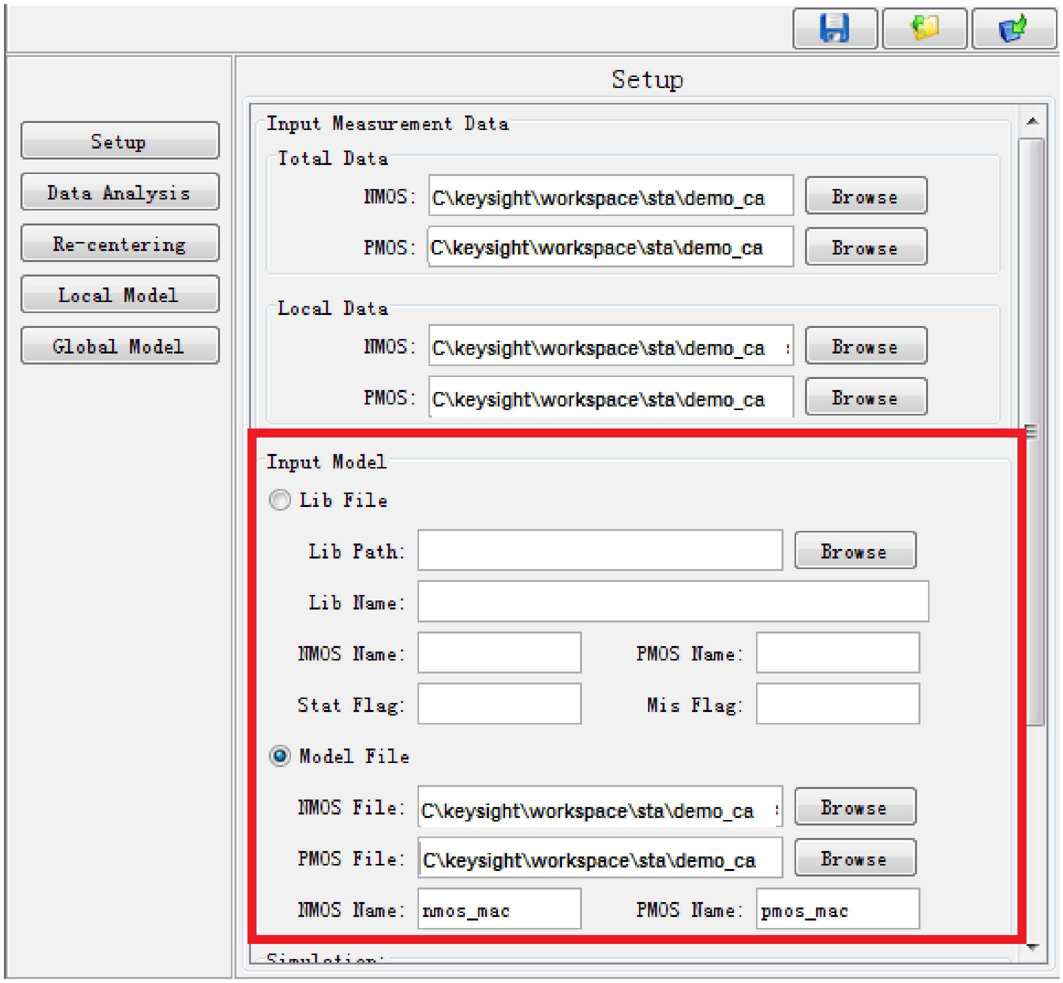
- Total Data (refer to statistical data)
- Local Data (refer to mismatch data)

User may only fill in one of them, if you want to extract either global model or local model.
User could also fill in NMOS or PMOS only.



Input Model Information

MBP statistical module accept either model library or single model card.



- By using model library, please input library path, library name, NMOS model name, PMOS model name, *stat flag*, *Mis flag*.
  - *Stat flag*: the switch parameter in the model card, to switch *on/off* the statistical function of the model.
    For example, we set *Stat flag* as "staflag", if staflag=1, all statistical parameters works; if staflag=0, all statistical parameters disabled.
  - *Mis flag*: the switch parameter in the model card, to switch *on/off* the mismatch capability of the model.
    For example we set *Mis flag* as "misflag", if misflag=1, all mismatch parameters works, if misflag=0, all mismatch parameters disabled.
- By using baseband model, you need to fill in nmos model path or PMOS model path, or both of them.
  - The model must be a sub-ckt model and here must input the sub-ckt model name.

Select Simulation Method

MBP provides three simulation methods in its internal engine: *Normal-MC simulation* method, *RSM simulation* method and *FMC simulation* method.

1. *Normal-MC*: traditional Monte Carlo simulation method
2. *RSM simulation*: speed-up the mc-simulation with Response-Surface Method
3. *FMC simulation*: speed-up the mc-simulation with linear approximation

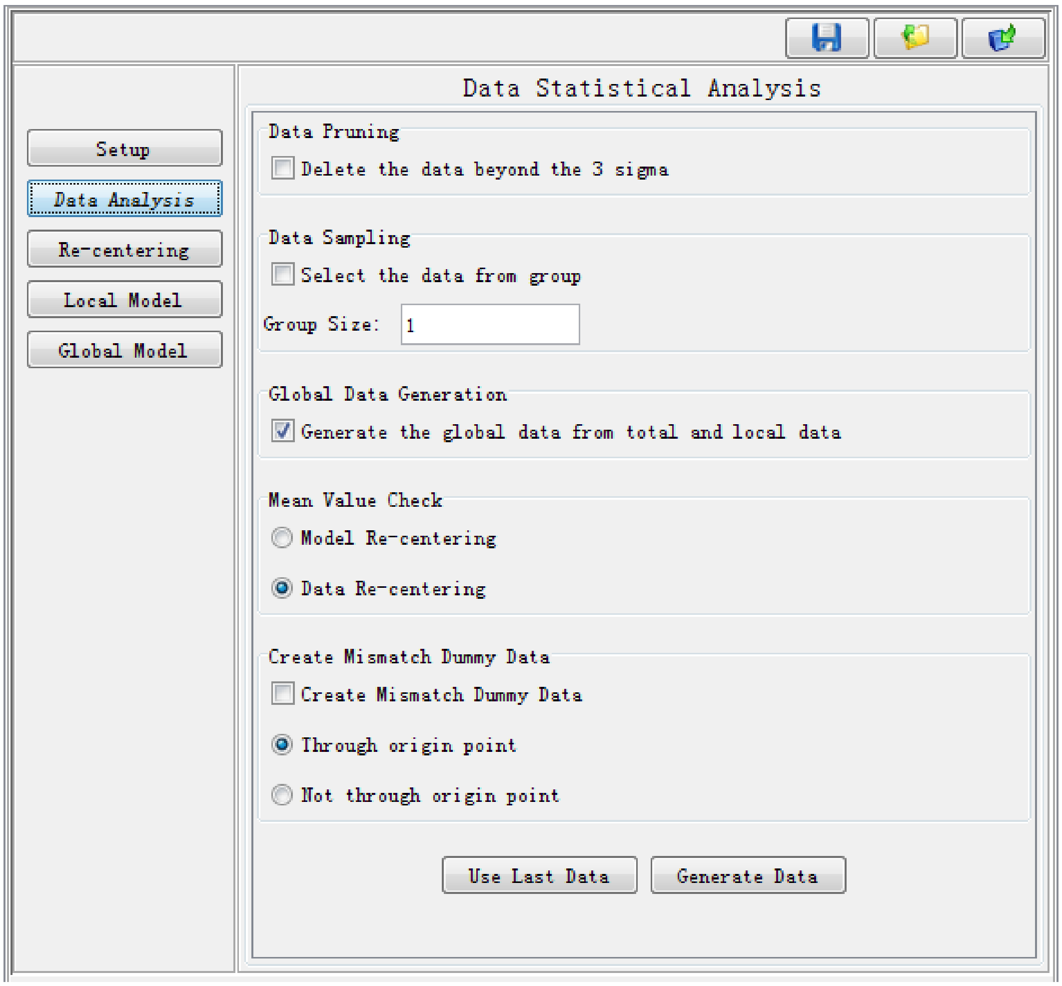
Add Comments

Any comments can be filled into the text box.
Click "Update" after all the steps accomplished,.

CAUTION Without completing "***Setup***", user cannot go to "***Data Analysis***".

Data Analysis

There are five steps in this section, four of them are optional.



Data Pruning

This option is to eliminate the data points beyond 3-sigma.

Data Sampling

This option is to minimize the data with "Group Size". For example, if "Group Size" is set to 10, MBP will select data every ten sites. If the entire data amount is too huge, user can sped up the progress by using this option.

Global Data Generation

This option is to generate global data from total and local data.

Mean Value Check

In case measurement total data's mean value is not equal to simulation total data's mean value, user could use this option to do data tweaking.
Two methods for the tweaking:

1. *Tweak model*: Tweak the baseband model to fit the measurement, user need go to next step "Baseband Model" to select parameters for tweaking.
2. *Tweak measurement total data*: MBP will adjust measurement data to fit model simulation. The tweaked data will be used in following steps.

Create Mismatch Dummy Data

This step is to create mismatch dummy data:

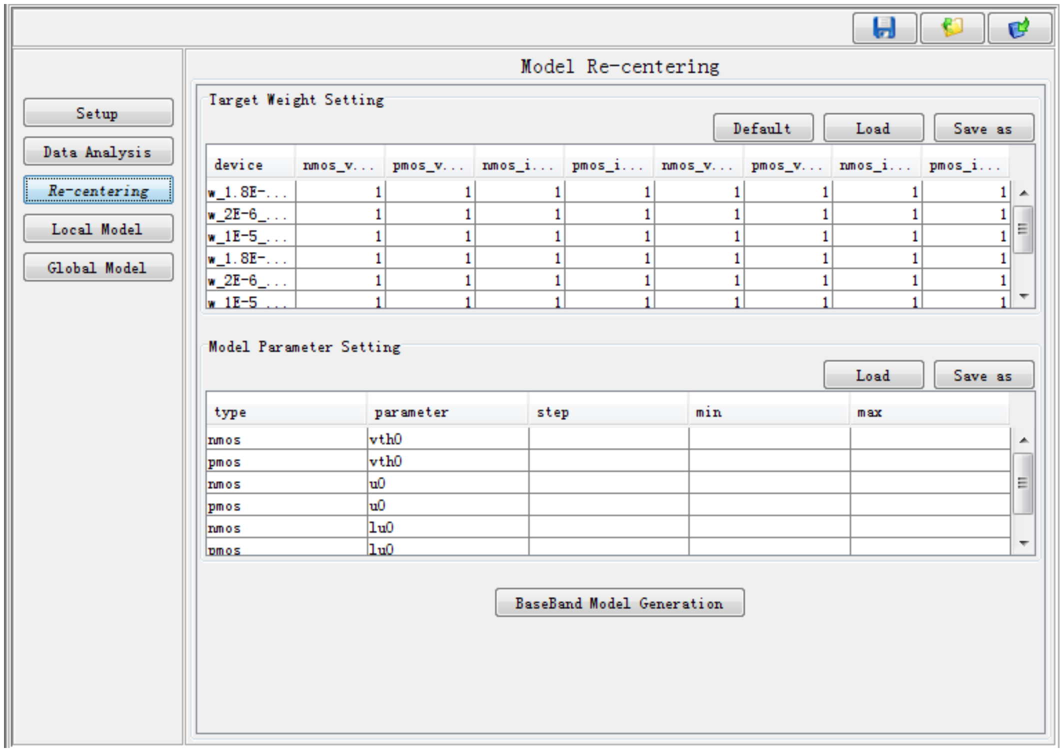
- The dummy data is a straight line, replace the original point data.
- User can choose the straight line through origin point or not before the dummy data generation.
- The dummy data line will be the optimization target during following steps; it can give better trend fitting result than original data points.

Click "Generate Data" button after all the steps to execute data dispose.
If user are working on an existing project and want to reuse the data generated in the last data dispose operation, user can click the button "Use Last Data".

Re-centering

CAUTION Must finished "Data Analysis" the can go to this step.
Re-centering is used when "***Tweak model***" is selected at "***Mean Value Check"*** steps, if user select "***Tweak data to fit model***", user could skip this step.

The step is used to select parameters for model tweaking, as below.



Target Weight Setting

Click "Default", a weight table will appear. The left side of the table is device column, the upside of the table is target row, and all the default weights are set to 1.0.

User can set one value to the entire column or the entire row. By set the first column(or the first row)'s weight and then right-click on this column(or row) select "Set value to the whole column" or "Set value to the whole row".

User can load pre-defined configuration file by "*Load*", also can save the configuration file by "*Save as*".

Model Parameter Setting

Fill in the parameters for model tweaking in this table, together with parameter's type ("nmos" or "pmos"), step, min/max value.
Pre-defined configuration file can be loaded by "*Load*". A new configuration file can be saved by "*Save as*".
Click "*Baseband Model Generation*", then the model will be tweaked automatically to fit measurement data

Local Model

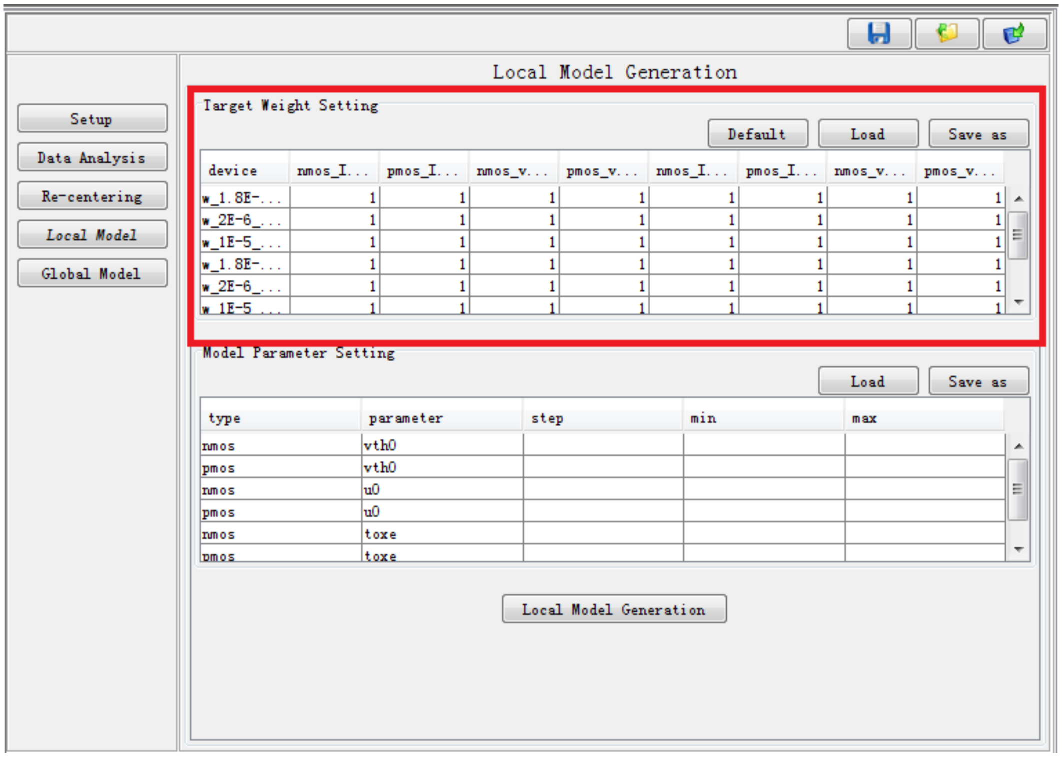
NOTE Note: Make sure you've finished "Data Dispose" before go to this step.

Local Model contains two parts:

- Target Weight Setting

- Model Parameter Setting

Target Weight Setting

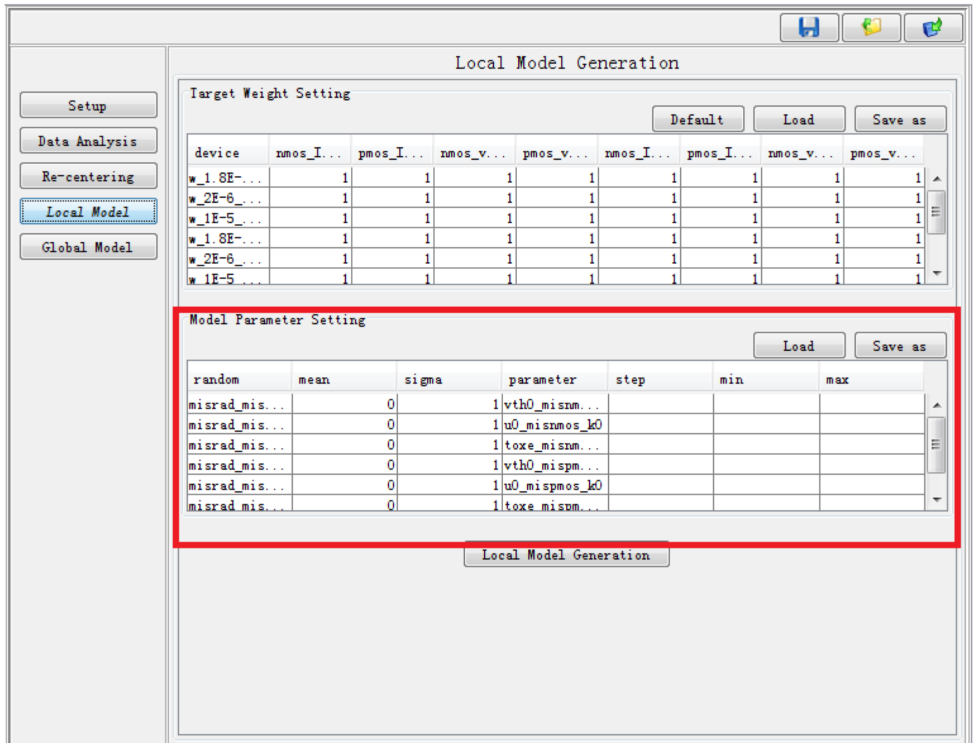


You can load your pre-defined configuration file by clicking "Load". And you can save the configuration file by clicking "Save as".

Model Parameter Setting

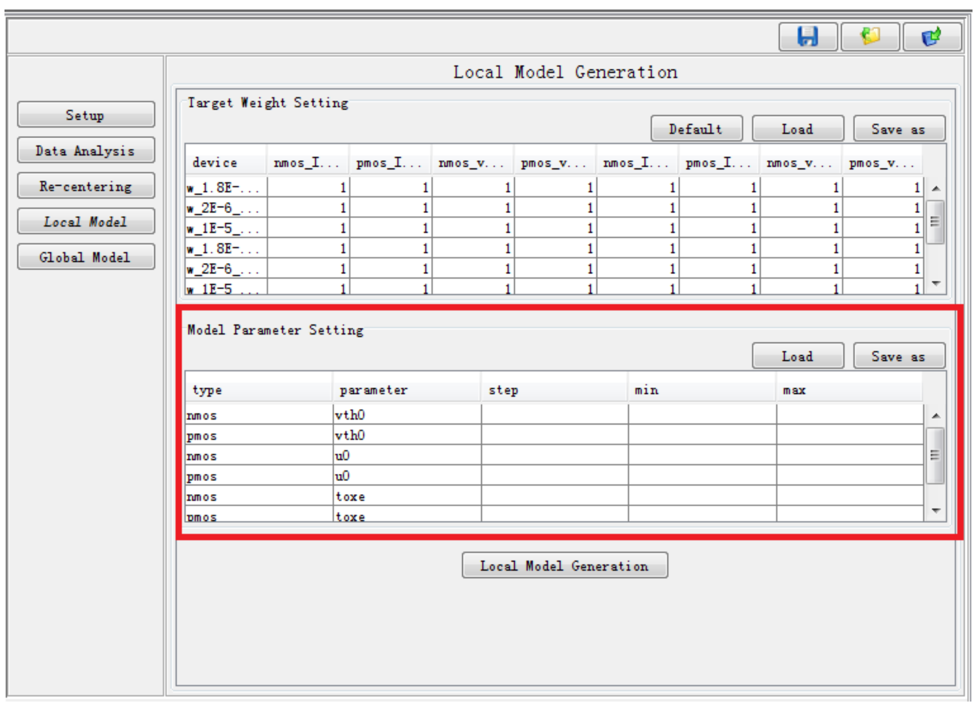
Two different table formats will appear depend on what format of model we start with: model library or single model card.

- For model library, the table shows as below:



Fill in this table with user-defined mismatch model parameters and corresponding random parameters, the mean and sigma value of the parameter are "1" and "0" by default respectively.

- If single model card is used, the table is as below:

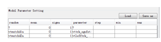


The table is very similar to "Baseband Model".
User need fill this table with parameters for tweaking, and specify parameters' polar type, polar type can be "NMOS" or "PMOS".

User can also assign parameter's step, min value and max value.
After setting up all the target weights and model parameters, click "*Local Model Generation*" button, then the mismatch model will be generated automatically.

NOTE Please setup your model library as the MBP's demo example. Below are some useful trick in the model parameter setting:

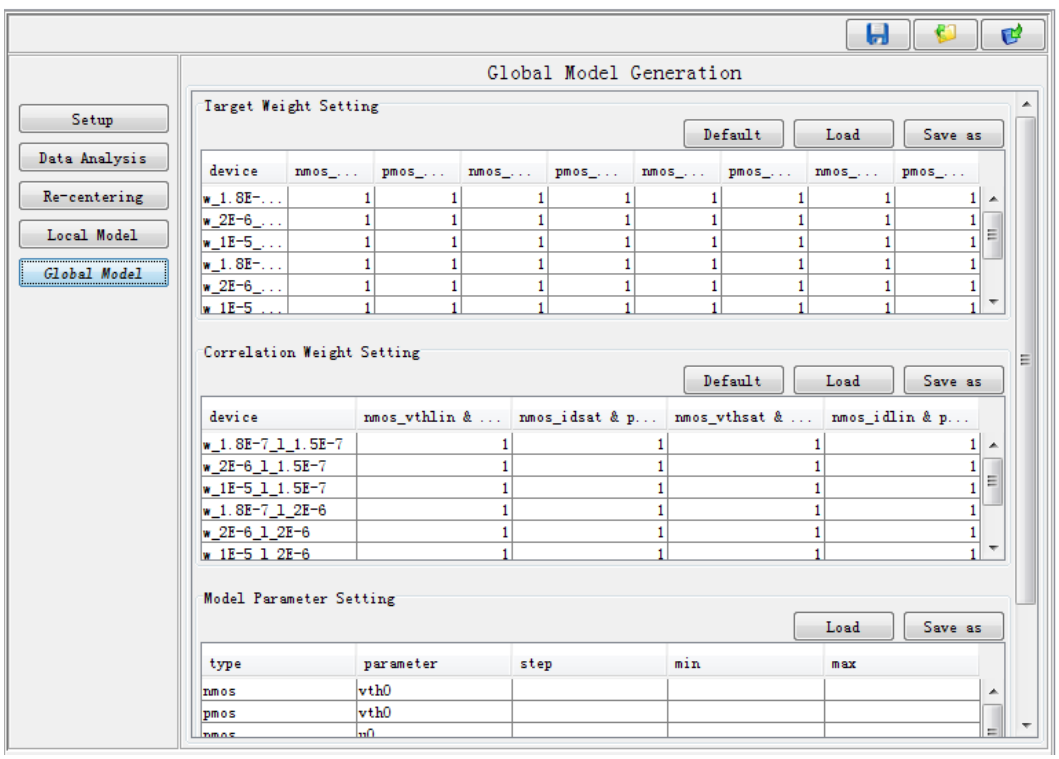
1. Add ":" after parameter name, it means this parameter's input value will be the initial value for next step
2. Add "=" after parameter name, it means this parameter's value will always keep constant
3. If the input lib is complex, and the relation between the random and parameter is not very clear, please add "?" at first line and add the tuning parameters in the following lines.



Global Model

NOTE Before go to this step, make sure "Data Dispose" is done.

This section is used to generate global statistical model automatically.

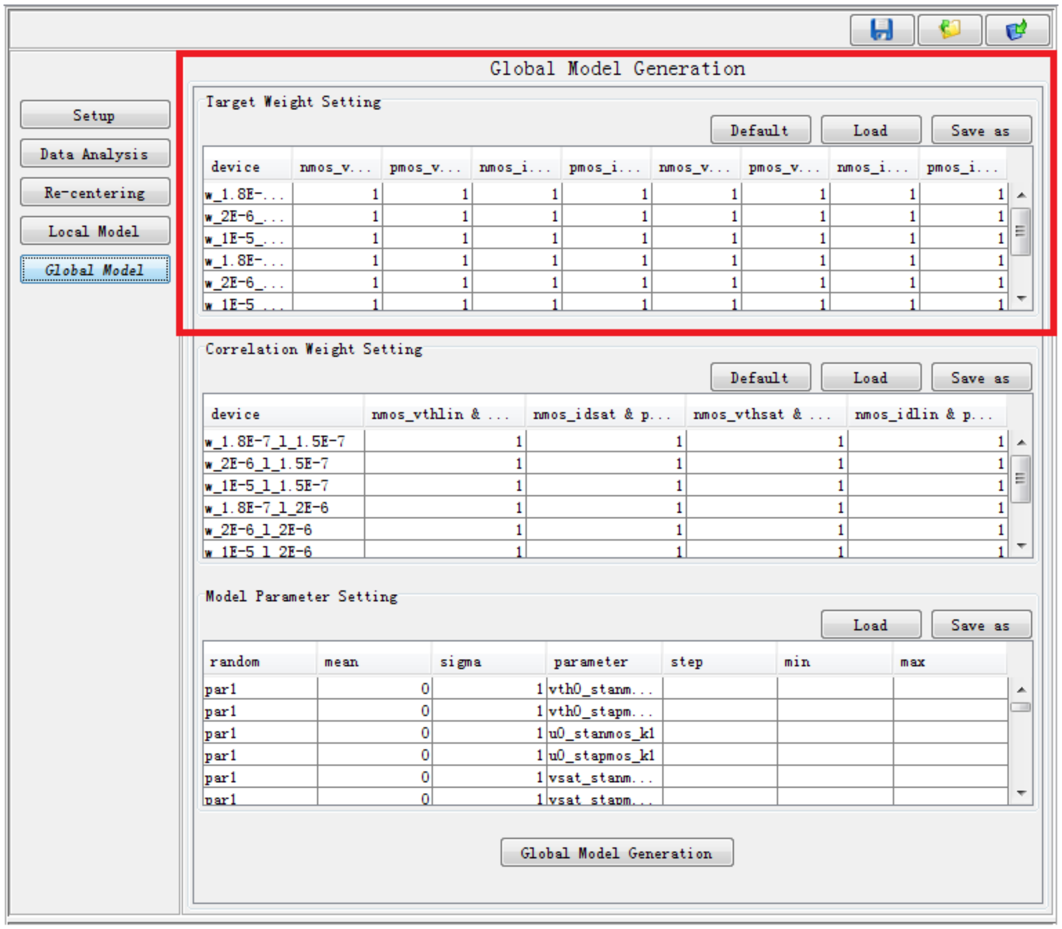


Three steps to complete a "Global Model" table:

- Target Weight Setting
- Correlation Weight Setting
- Model Parameter Setting

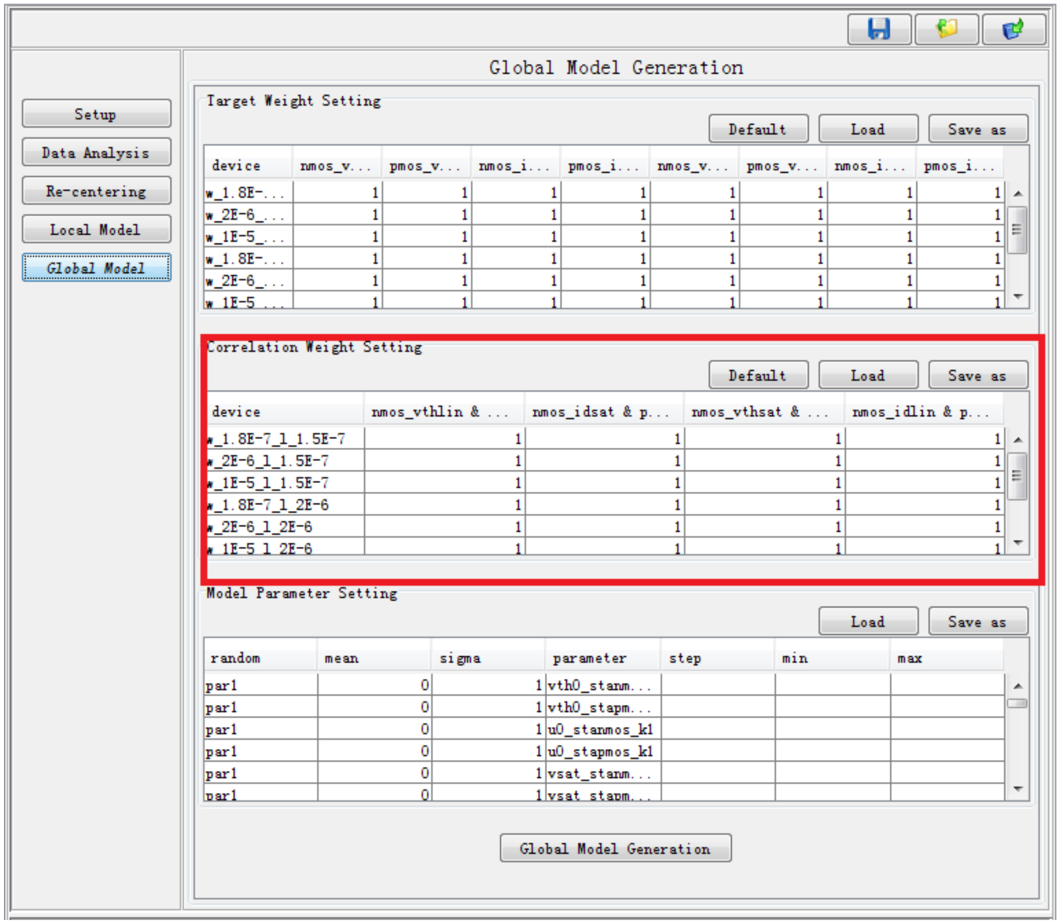
Target Weight Setting

User is suggested to use the default setting by click "*Default*" button.
Different weight can be set to different target, MBP will put more effort on the target with high weight
Configuration file can be save and load by "*Save as*" and "*Load*" button.



Correlation Weight Setting

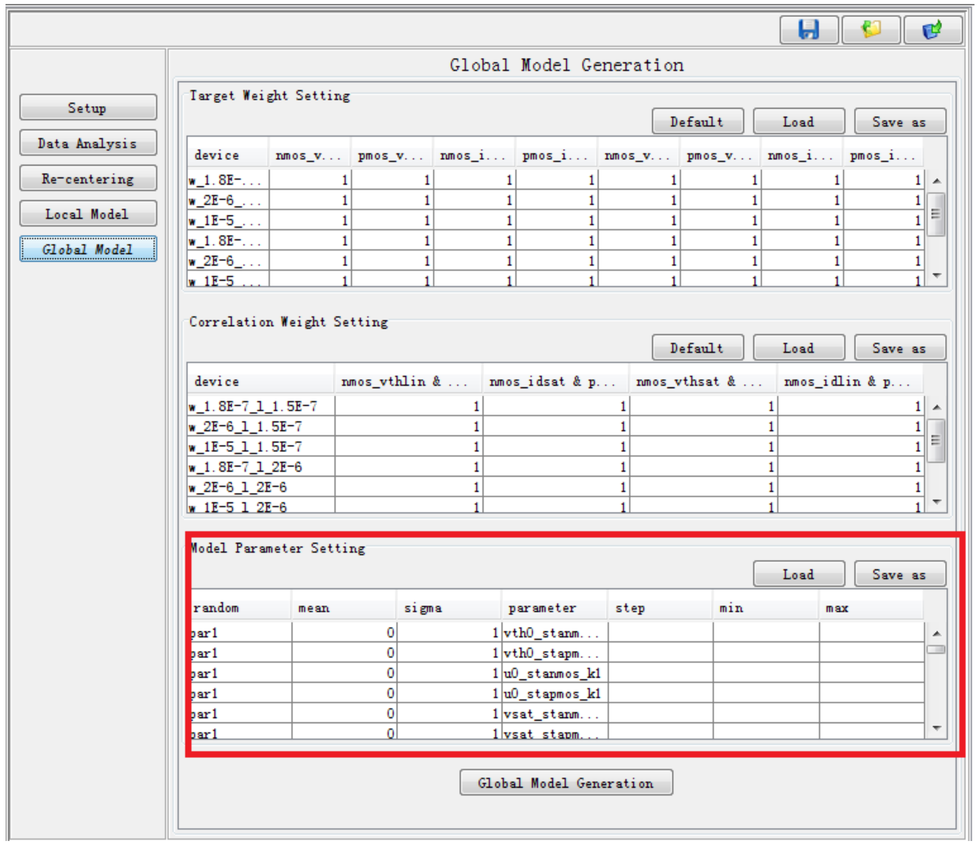
If both NMOS and PMOS data is loaded within one extraction progress, user are enabled to set weight for NMOS and PMOS correlation.
MBP put more effort to the correlation with higher weight, may have better fitting accuracy normally.



Model Parameter Setting

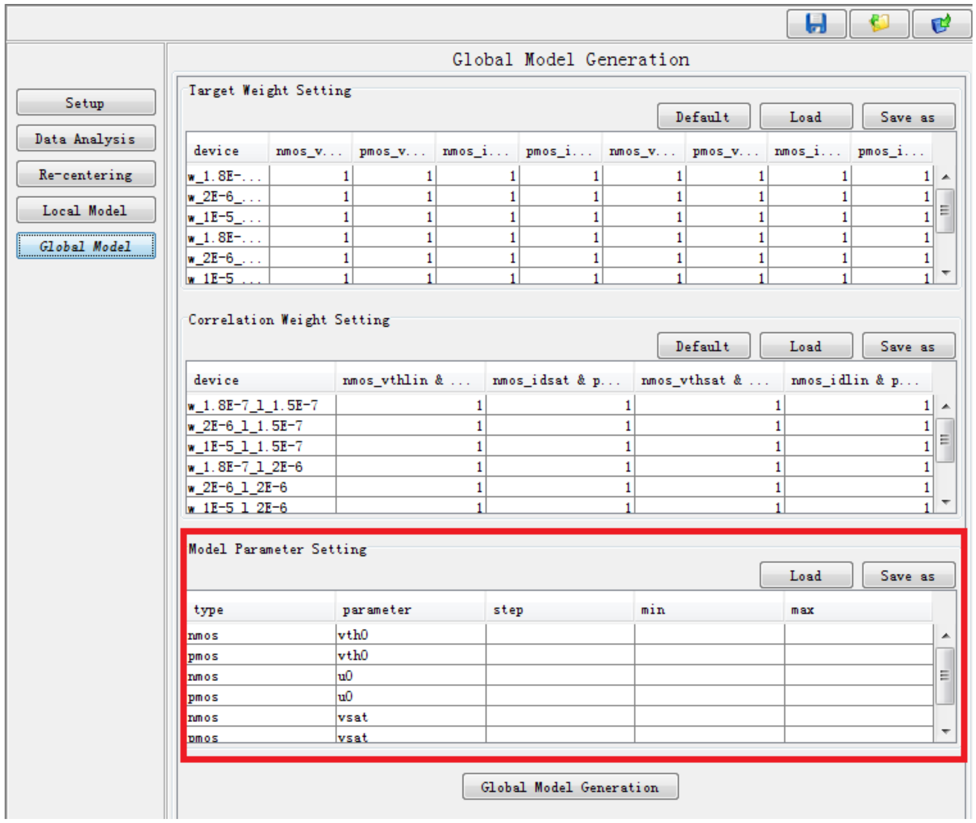
The table has two forms depending on the input model format.

- Library model, the table is as below:



User need fill in this table with user-defined statistical model parameters and corresponding random parameters.
Parameters' mean and sigma value are "1" and "0" by default respectively.
User can also assign parameter's step, min/ max value.
The configuration file can be save and load by "*Save as*" and "*Load*" button.

- If single model card is used, the table is as below:

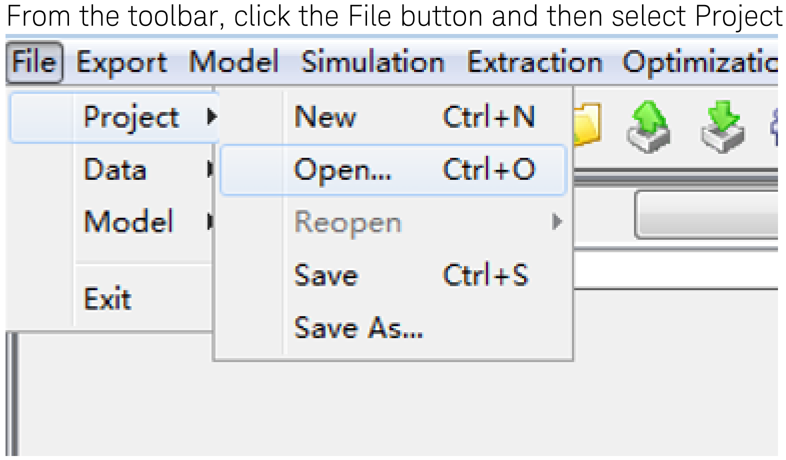


User need fill in this table with parameters for tweaking, and specify parameters' corresponding polar type.
User can also assign parameter's step, min/max value.
The configuration file can be save and load by *"Save as*" and "*Load*" button.
Click "*Global Model Generation*" button after completed all setups, then the global statistical model will be generated automatically.

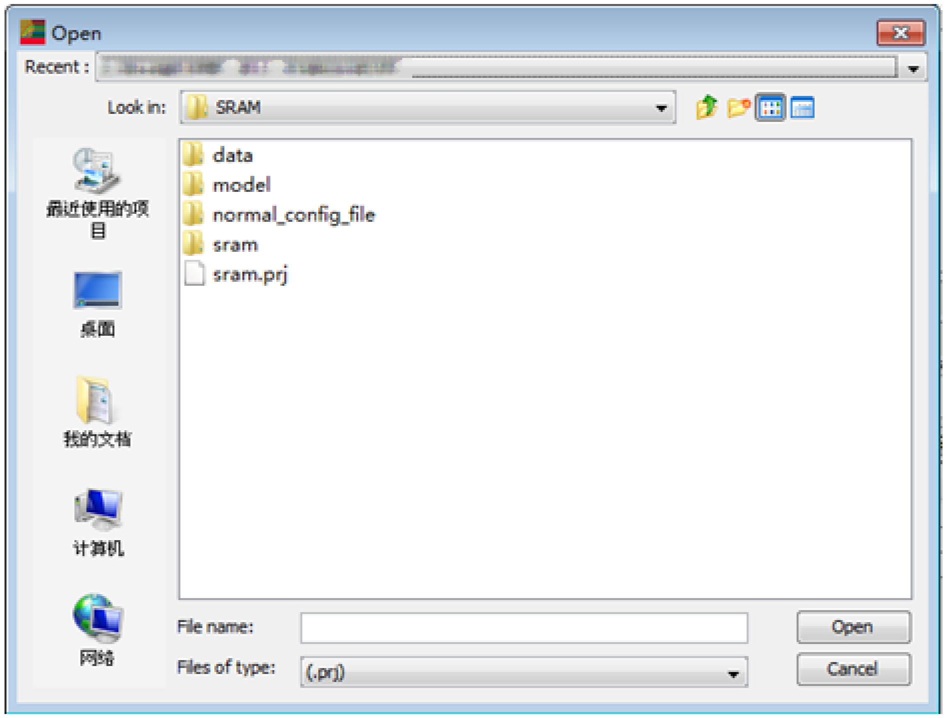
SRAM Solution Operation Note

SRAM solution is an integrated solution in MBP for SRAM application. With this solution, the customer can check both SRAM and MOSFET device at the same time. All the details will be introduced in the following sections.

1. Load project
   From the toolbar, click the File button and then select Project > Open...

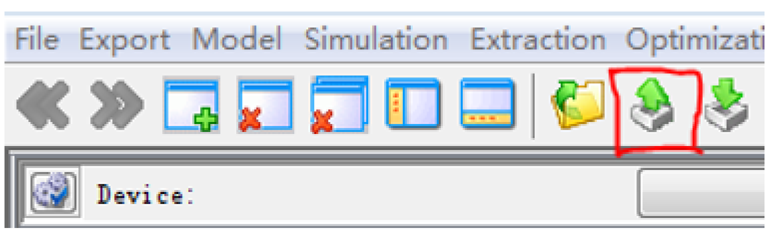


Select a project from the pop-up window. A demo example has been included at :$MBP_home\demo\SRAM\sram.prj.

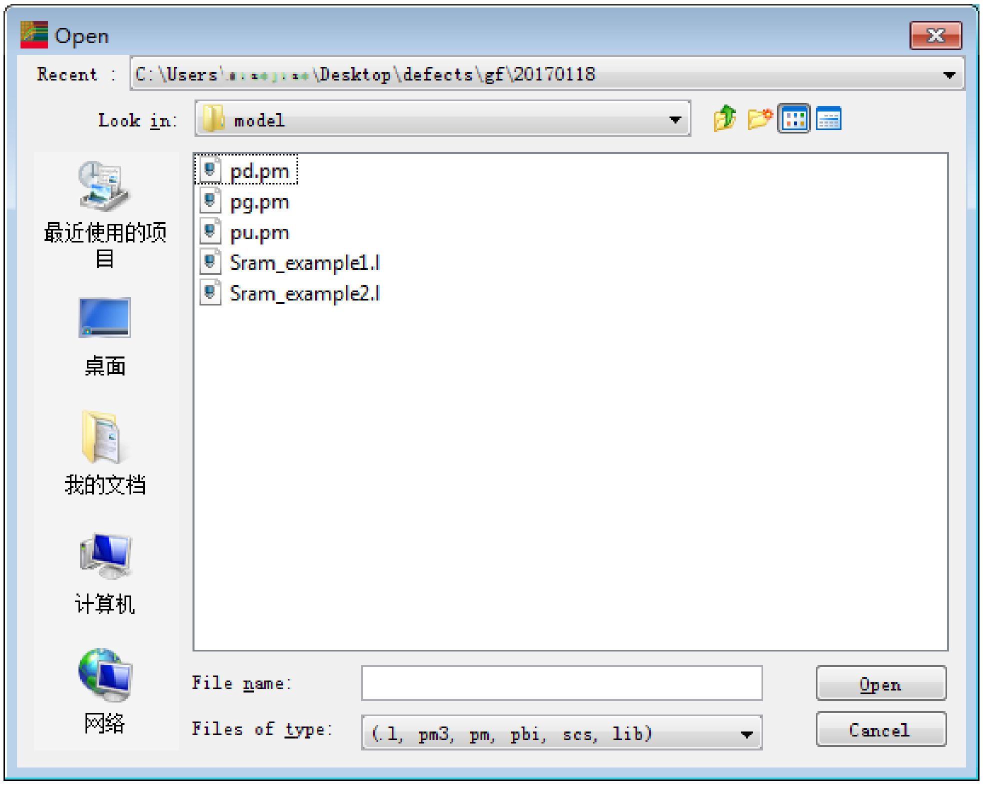


2. Load model

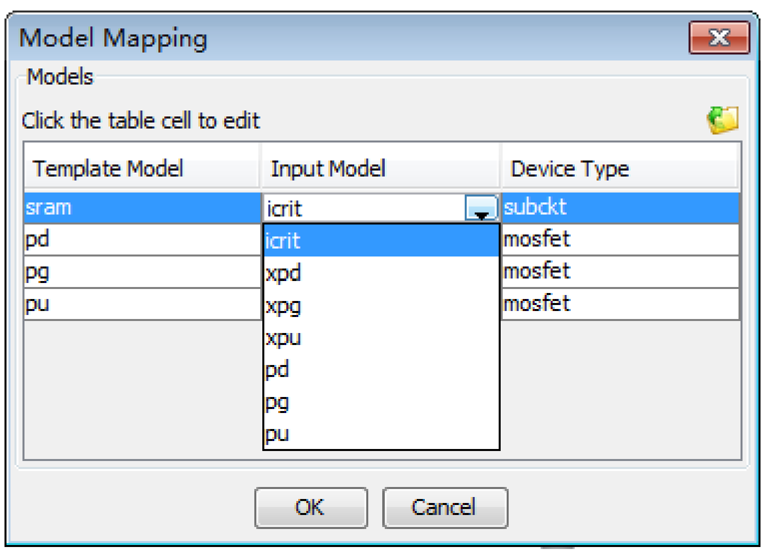
Click Load Model button.



Then load the model from the browser window:



After model selection, a model mapping window will pop-up. Map the input model with the Template Model categorization correctly.

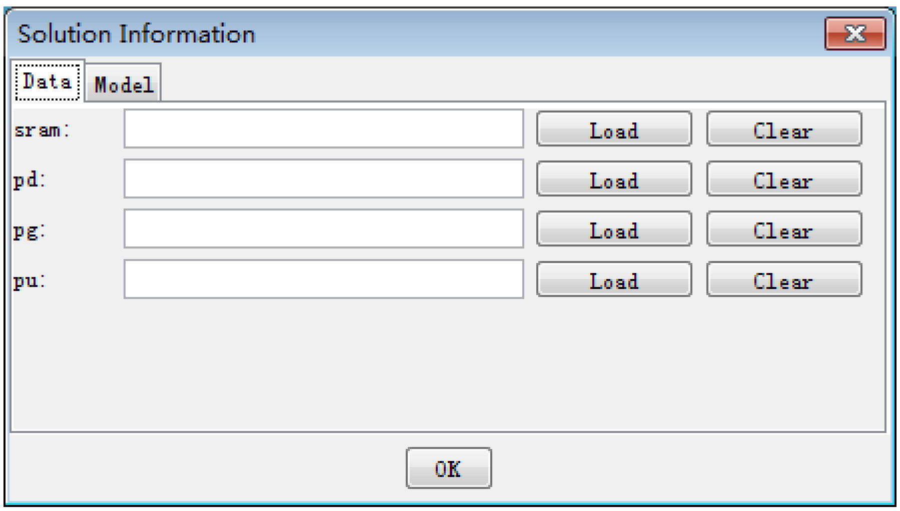


Actually, in SRAM application, any customer input model can be categorized as SRAM,pd, pg,pu (pull-down, pass-gate, and pull-up MOSFET), just like the "Template Model" column list.

In the example, “icrit” in the input model is the SRAM -kind model. With these mapping, it will ensure that MBP script draws and display the correct target plot.

3. Load data
   Click the load data button.

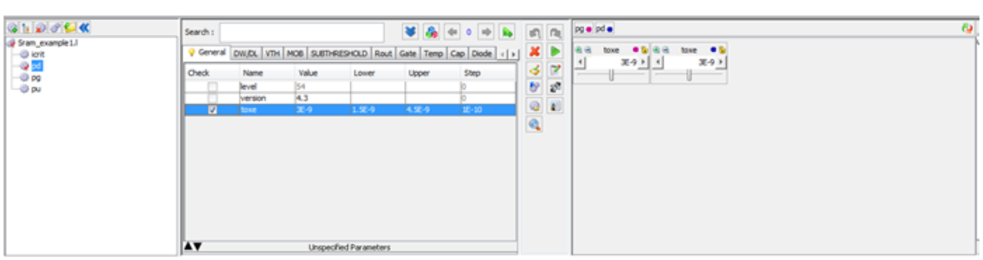
Then the loading data window will pop-up.



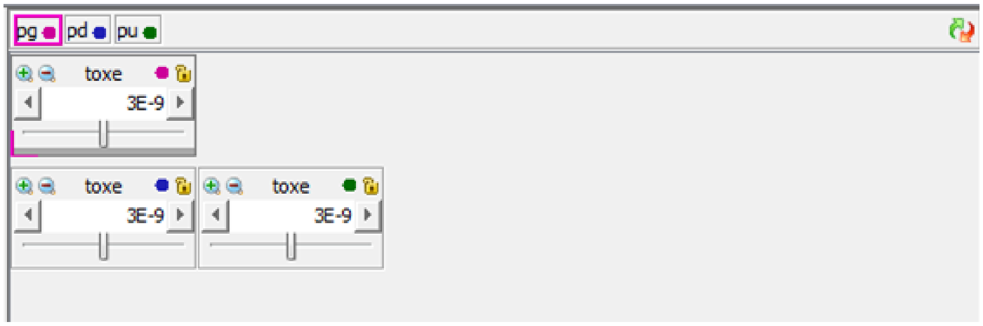
Click the **Load** button to load data for every kind of model respectively, and it is ok to select more than one data file at the one time. Click the **Clear** button to empty the data you have selected. After these setting up, click the **OK** button to finish loading data.

4. Parameter tuning
   Click model in the model navigator and select model parameters in the parameter navigator to tune like the following picture.

   The selected model parameters will be displayed in the parameter panel, and different model’s parameter will be set with different colors.

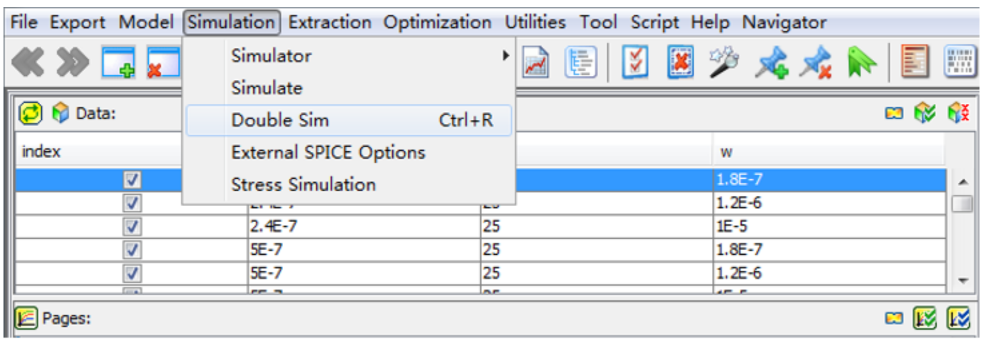


Click the button on the top of parameter panel (pd, pg, pu in the following picture). The selected model's parameters will put into the front of the panel.

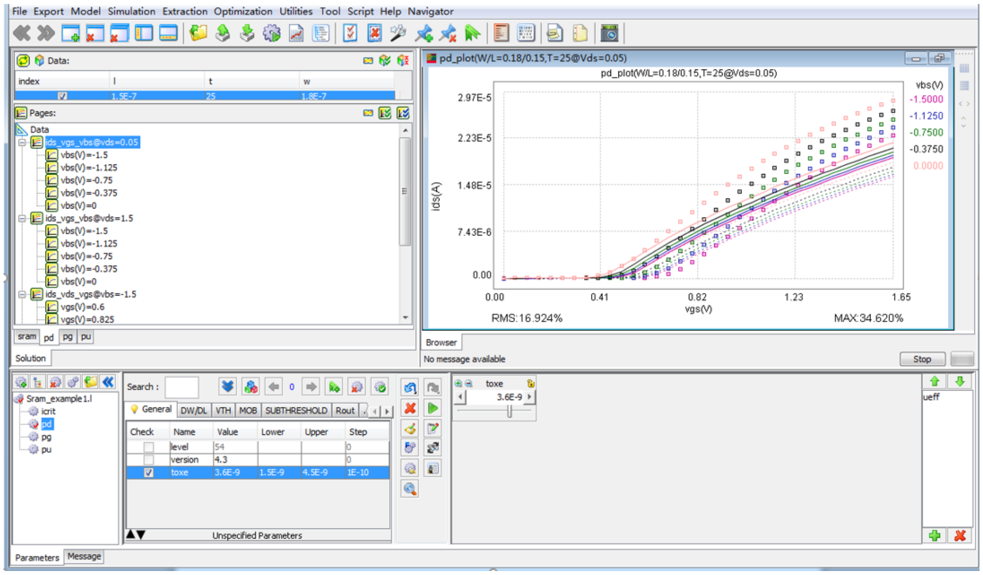


5. Double simulation
   Double simulation is a power function to check parameter's effect on the circuit or device performance.

   Click Double Simulation from the menu.

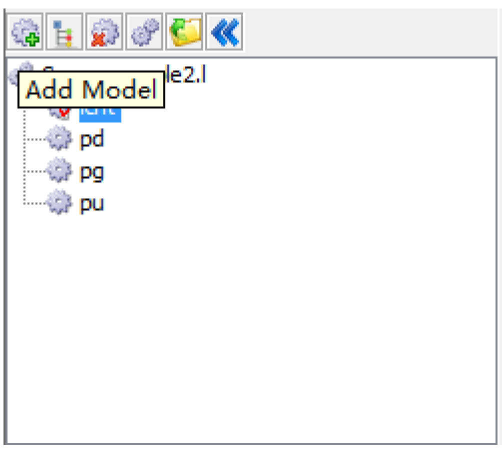


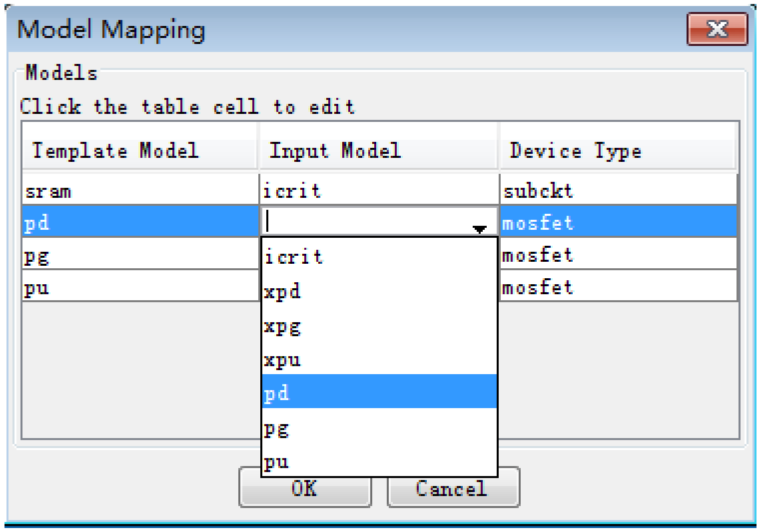
   Select some parameters from the parameter navigator and tweak them. And in the plot, square point line is the measurement data, other two curves are simulation result. The solid line represents the simulation result from original parameter value, and dot line is for updated parameters' simulation result.



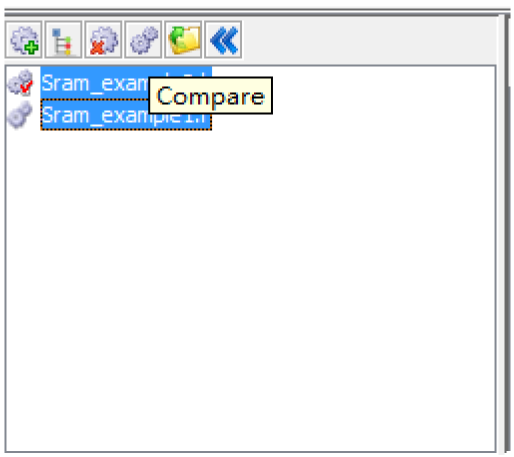
6. Model Compare

   Model Compare can be used to compare two models which have same Electrical Target and different model parameter.
   Firstly, click the Add Model button to add a new model into MBP.

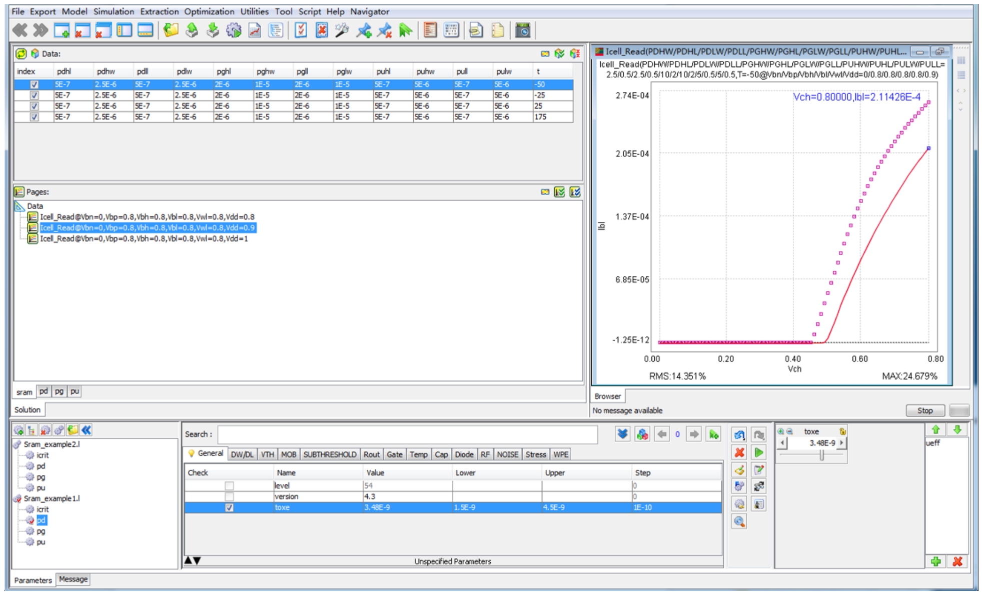




Select both models and click **compare** from the button.

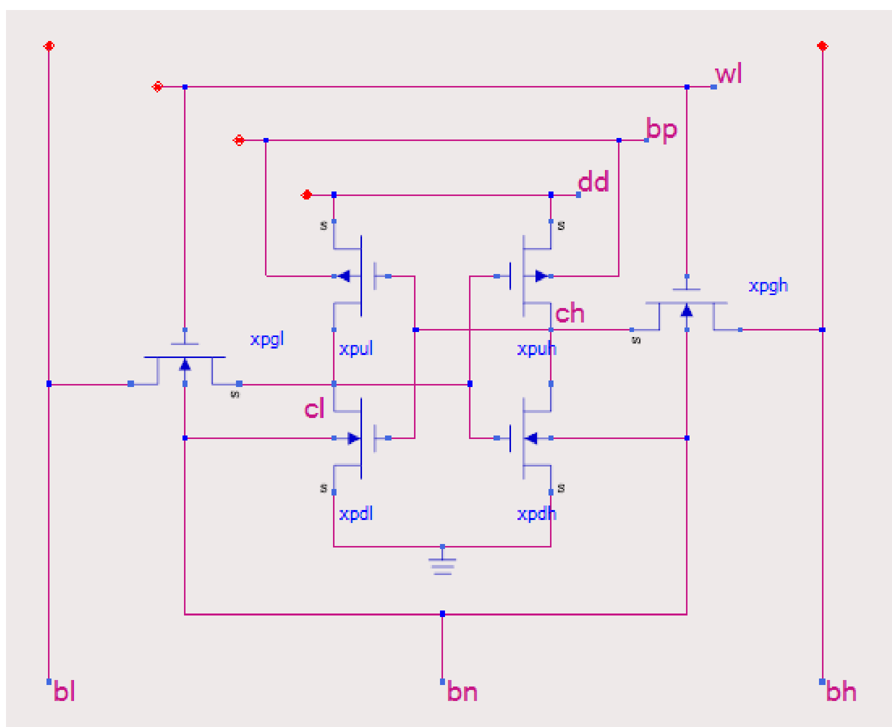


Tweak some parameter to check plots.



7. Appendix

   a. 6T SRAM circuit



   b. Mosfet name

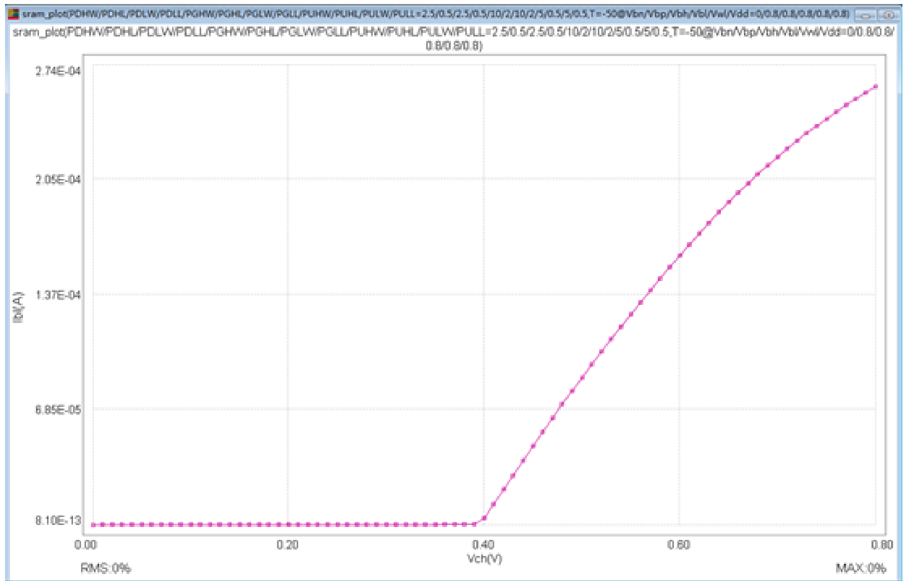
   pu: means "pull up"mosfet

   pd: means "pull down"mosfet

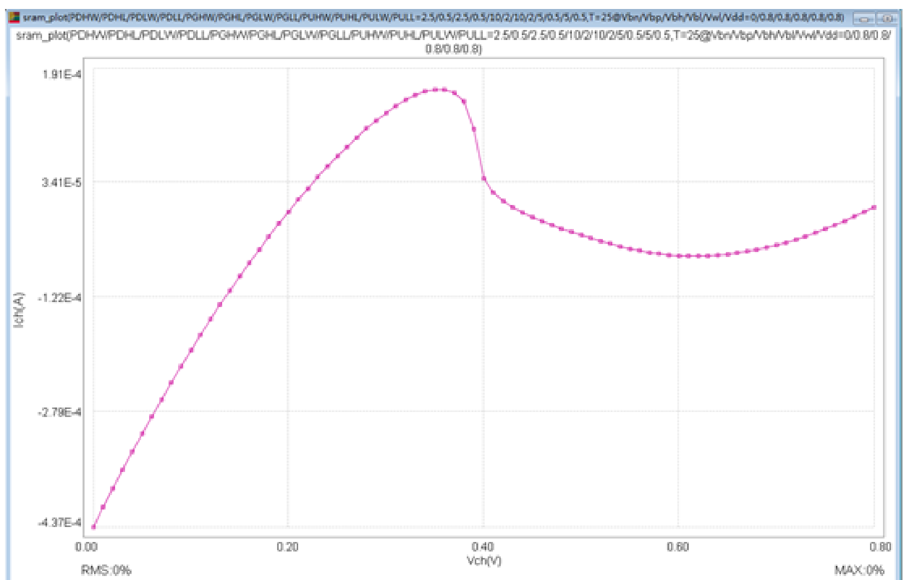
   pg: means "pass gate"mosfet

c. Plot definition

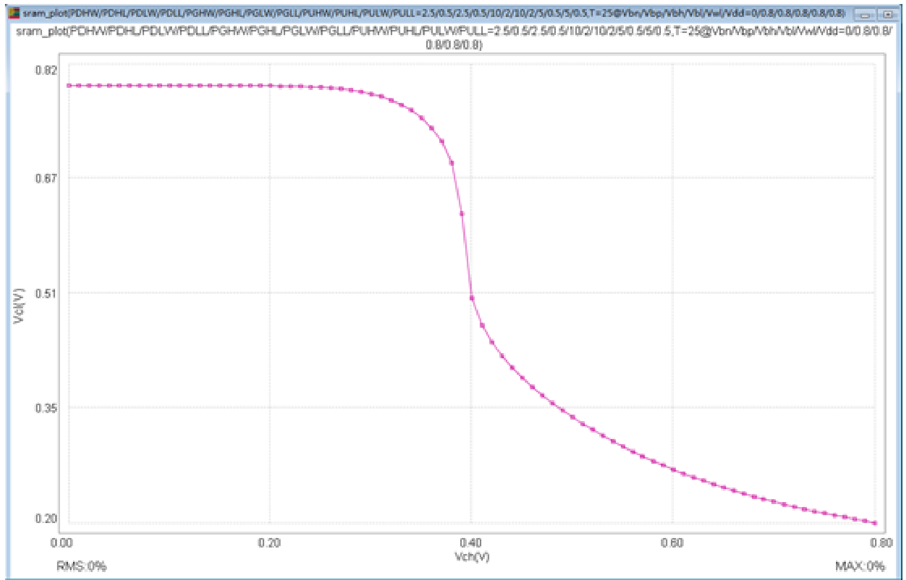
Vch_Ibl: sweep voltage at node “ch”, and get the current value to node “bl”.



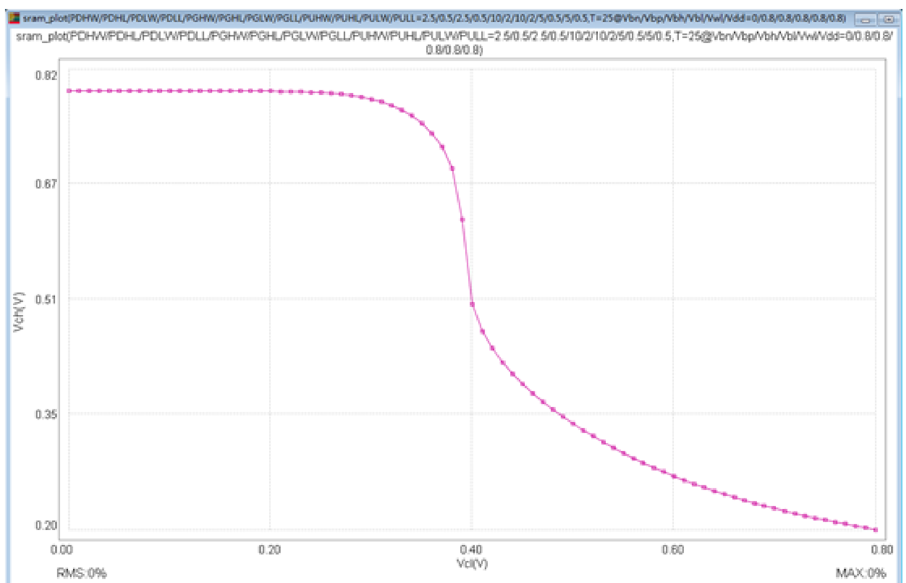
Vch_Ich: sweep voltage at node ch, and get the current value to node “ch”.



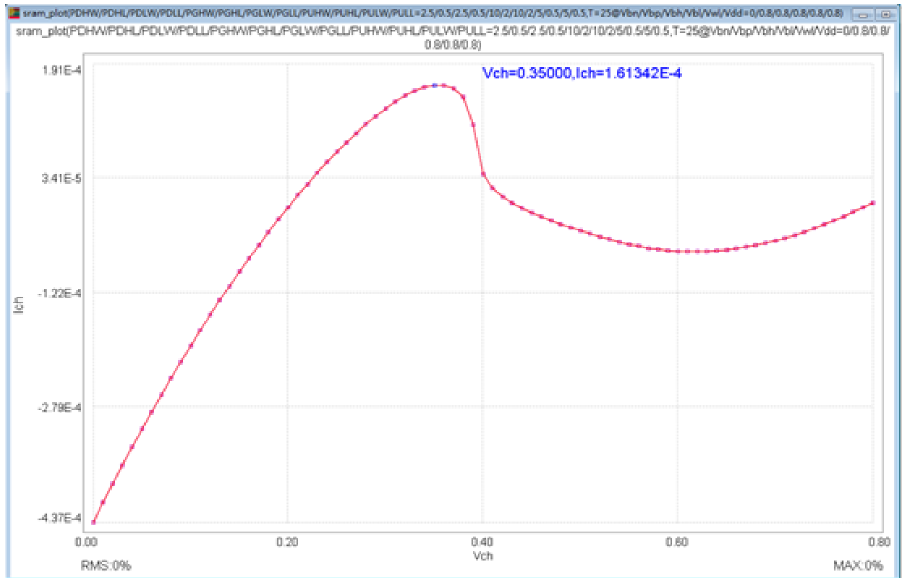
Vch_Vcl: sweep voltage at node ch, and get the voltage value at node “cl”.



Vcl_Vch: sweep voltage at node cl, and get the voltage value at node “ch”.

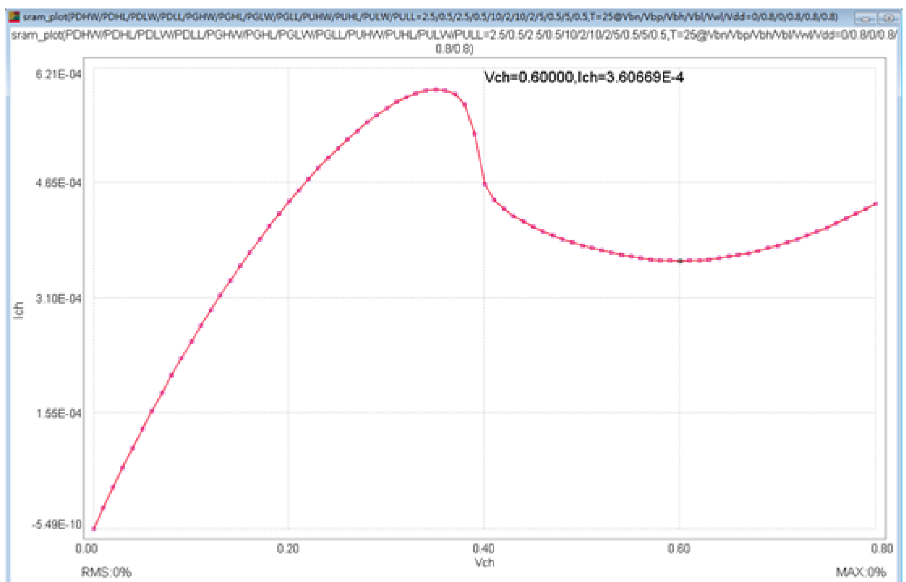


d. Target introduction
   Icrit_RD[1]:
   SRAM work at read mode (Vbh=vdd, Vbl=vdd), get the maximum current value at Vch_Ich plot, the graph displays this point’s voltage and current value on Vch_Ich plot.



Icrit_WR[1]:

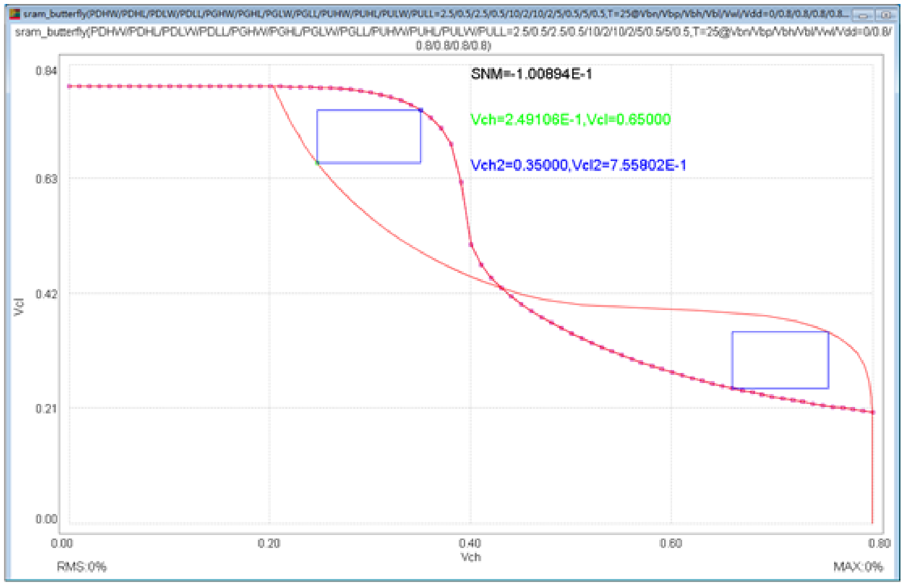
SRAM work at write mode (Vbh=0, Vbl=vdd), get the minimum current value which Vch'svalue is between 0.3*vdd and vddatVch_Ich plot, the graph displays this point's voltage and current value on Vch_Ich plot.



Butterfly plot and SNM target:

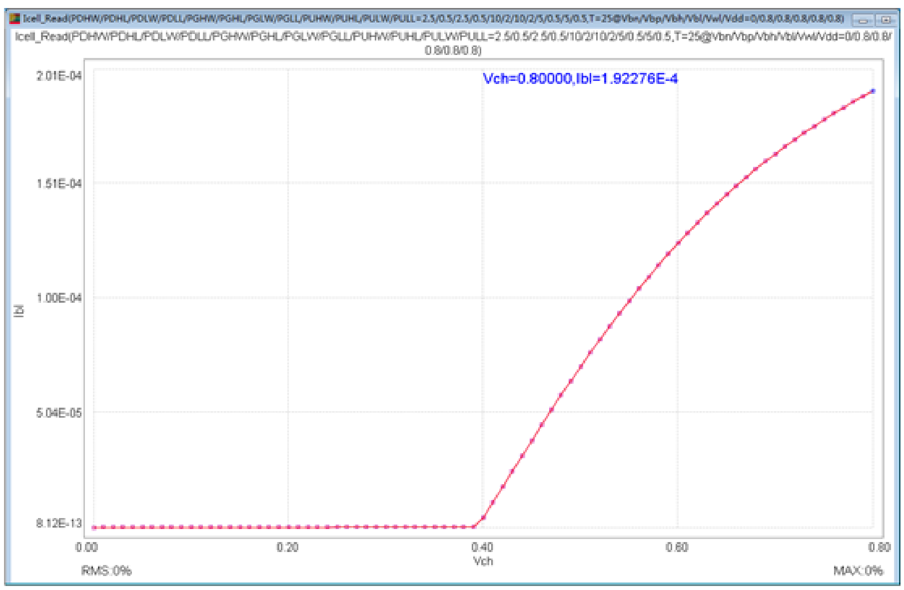
Butterfly plot: The Graph overlays the two plots Vch_Vcl and Vcl_Vch to one plot to get butterfly plot.

SNM target: static noise margin, the graph displays SNM's value on the butterfly plot.



Icell_Read[2]:

SRAM work at the read mode, get the maximum current value on Vch_Ibl plot, the graph displays this point's voltage and current value on Vch_Ibl plot.



Reference:

1. Critical Current (ICR1T) Based SPICE Model Extraction for SRAM Cell
2. CMOS SRAM Circuit Design and Parametric Test in Nano-Scaled Technologies

Statistical Model Extraction Module

General Introduction

MBP's statistical model extraction module is fully developed for generating SPICE model with Monte Carlo simulation capability, a capability of both local mismatch and global statistical. The module provides GUI-Based Turn-Key solution and its work-flow chart is like following:3

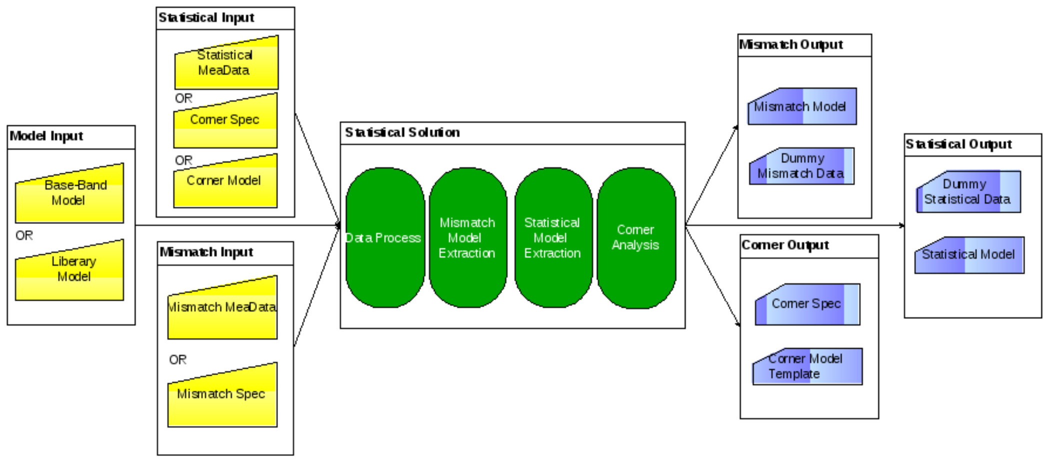


Figure 1: General work-flow chart

In the Module, it supports the following input:

Inputs for this solution

Mismatch Input:

For mismatch model extrication, select one of these two kinds input.

1. Mismatch Measurement Data.
2. Mismatch Spec, dummy mismatch data will be generated from these specifications and replace the measurement data for mismatch model extraction.

Statistical Input:

For statistical model extrication, select one of these three kinds input.

1. Total Measurement Data;
2. Corner Spec, dummy statistical data will be generated from these corner spec and replace the measurement data for the statistical model extraction;
3. Corner Model, the statistical model could be generated directly from this corner model;

Model Input:

1. Baseband Model, no statistical-equation has been integrated into model card and our solution will generate equations automatically;
2. Library Model, it has pre-defined statistical-equation in model card and our solution will tweak the equation's coefficients to match targets;

Output from this solution

With our solution, it can get all kinds of output in the mismatch/statistical model extraction.

Mismatch Output:

1. Mismatch Model;
2. Generated dummy mismatch Data;
3. Mismatch Plot

Statistical Output:

1. Statistical Model;
2. Generated dummy statistical Data;
3. Histogram/Distribution Plot
4. Scatter/Contour Plot
5. Scatter/Corner Plot

Corner output:

1. Corner Spec;
2. Corner Model Template;

Typical Application Examples

With proper setup, this solution could be applied widely for various cases. Following table lists some typical application examples. These examples could be good references for quick start. And all the technical details will be presented in following section.

Tab 1. Typical Application Examples

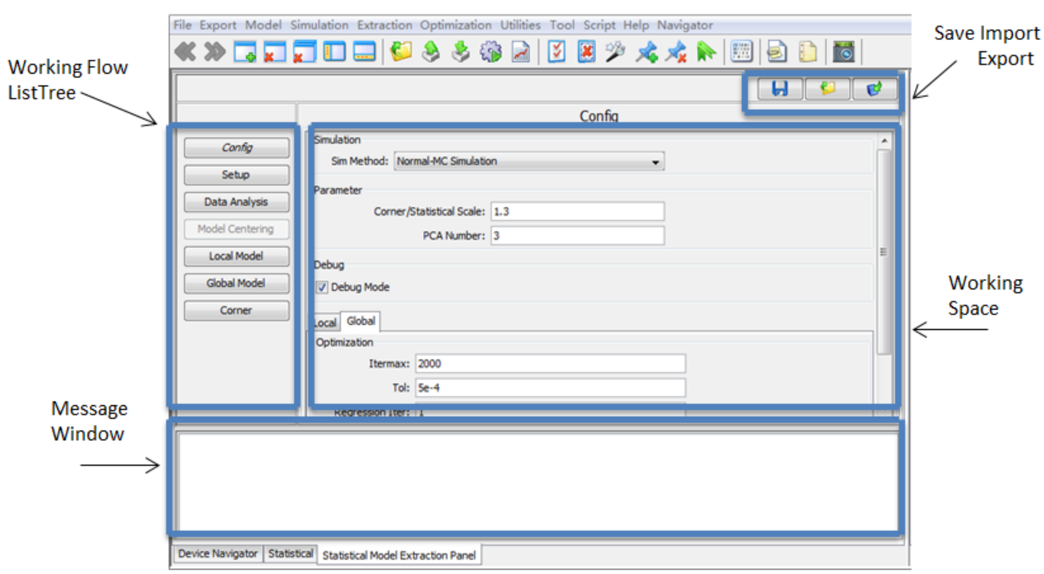
	Input	Solution	Output	Comments
1.	1.Mismatch Measurement data; 2.Statistical Measurement data; 3.Baseband Model;	1.Data Analysis; 2.Mismatch Model Extraction; 3.Statistical Model Extraction	1.Mismatch Model; 2.Statistical Model;	Don't need expertise on the statistical model extraction

	Input	Solution	Output	Comments
2.	1.Mismatch Measurement data; 2.Statistical Measurement data; 3.Liberary Model;	1.Data Analysis; 2.Mismatch Model Extraction; 3.Statistical Model Extraction	1.Mismatch Model; 2.Statistical Model;	Flexible , and designed for specific applications
3.	1.Mismatch Spec; 2.Baseband Model;	1.Dummy Data generation; 2.Mismatch Model Extraction;	1.Dummy Mismatch Data 2.Mismatch Model;	Mismatch model extraction without real measurement data
4.	1.Corner Spec; 2.Baseband Model;	1.Dummy Data generation; 2.Statistical Model Extraction;	1.Dummy Statistical Data 2. Statistical Model;	Statistical model extraction without real measurement data
5.	1.Corner Model; 2.Baseband Model;	1.Dummy Data generation; 2.Statistical Model generation directly;	1.Dummy Statistical Data 2. Statistical Model;	Generate statistical model from corner model directly

Work Flow

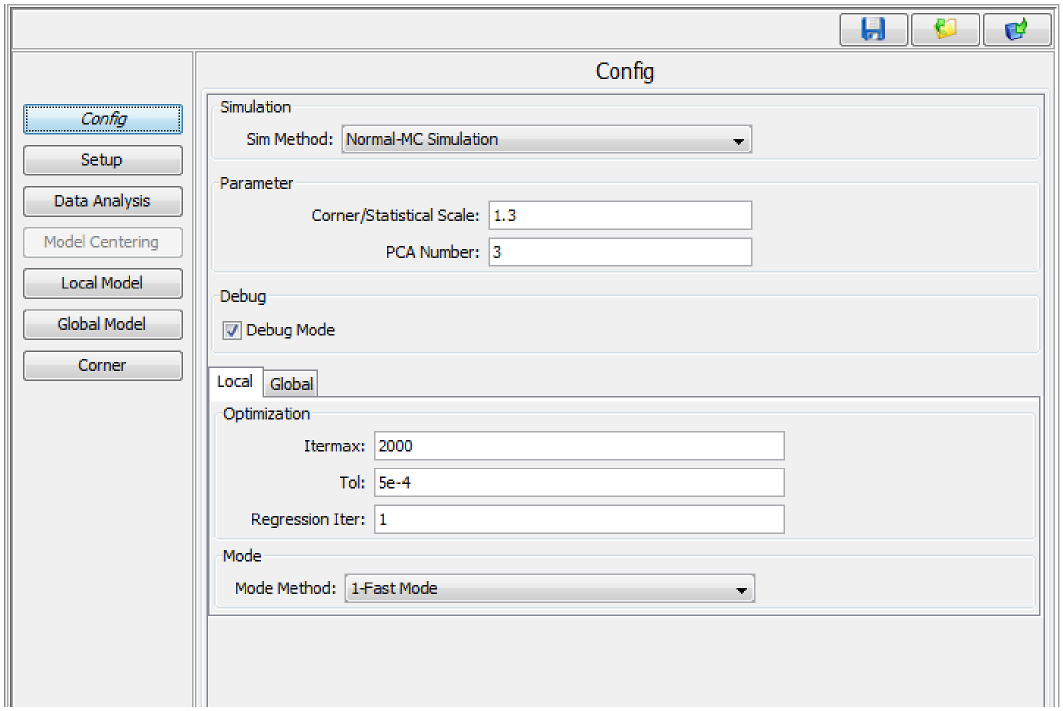
Load Project

Load Statistical Project from MBP's demo folder, Select Navigator > Statistical Model Extraction Panel.



Config

In this window, user can configure the system parameters.



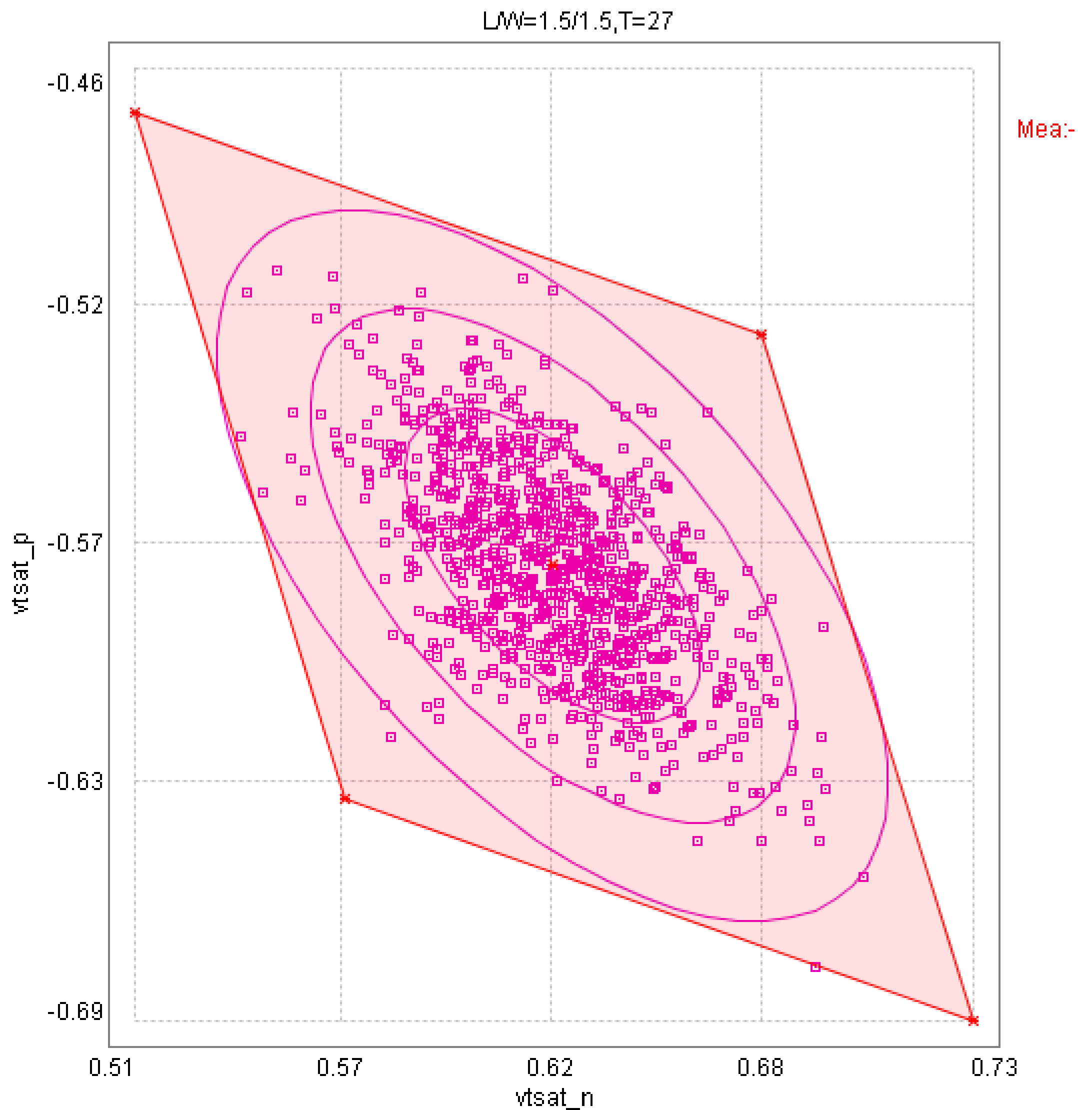
Simulation: Switch between 3 simulation methods

Normal-MC:	General Monte Carlo simulation, high accurate;
RSM Simulation:	Response surface method simulation, fast speed;
FMC Simulation:	Monte Carlo simulation with linear approximation, fast speed;

Parameter:

Corner/Statistical Scale:	It is a parameter to define ration scale between corner and statistical; please take table2. 1 as reference;
PCA Number:	It is order number in Principal Component Analysis; Just like following example.

Tab 2. corner/statistical scale



Corner/Statistical Scale=1

Debug Mode:

Debug Mode:	If Debug Mode is open, the extraction detail information will be printed to the log file for the trouble-shooting. Usually, the log file is saved to the project_path\ statistical\output\log\statdebug.txt

Local Optimization Parameters: optimization parameters' setting for local model extraction

Itermax:	Max iteration times in optimization ;
Tol:	Target tolerance for optimization convergence criterion;
Regress Iter:	Iteration times for regression calculation;

Local Extraction Method:

Fast Mode:	Use linear algorithm for mismatch model extraction, fast speed;
Normal Mode:	Use general optimization for mismatch model extraction;

Global Optimization Parameters: optimization parameters' setting for global model extraction

Itermax:	Max iteration times in optimization ;
Tol:	Target tolerance for optimization convergence criterion;
Regress Iter:	Iteration times for regression calculation;

Global Extraction Method:

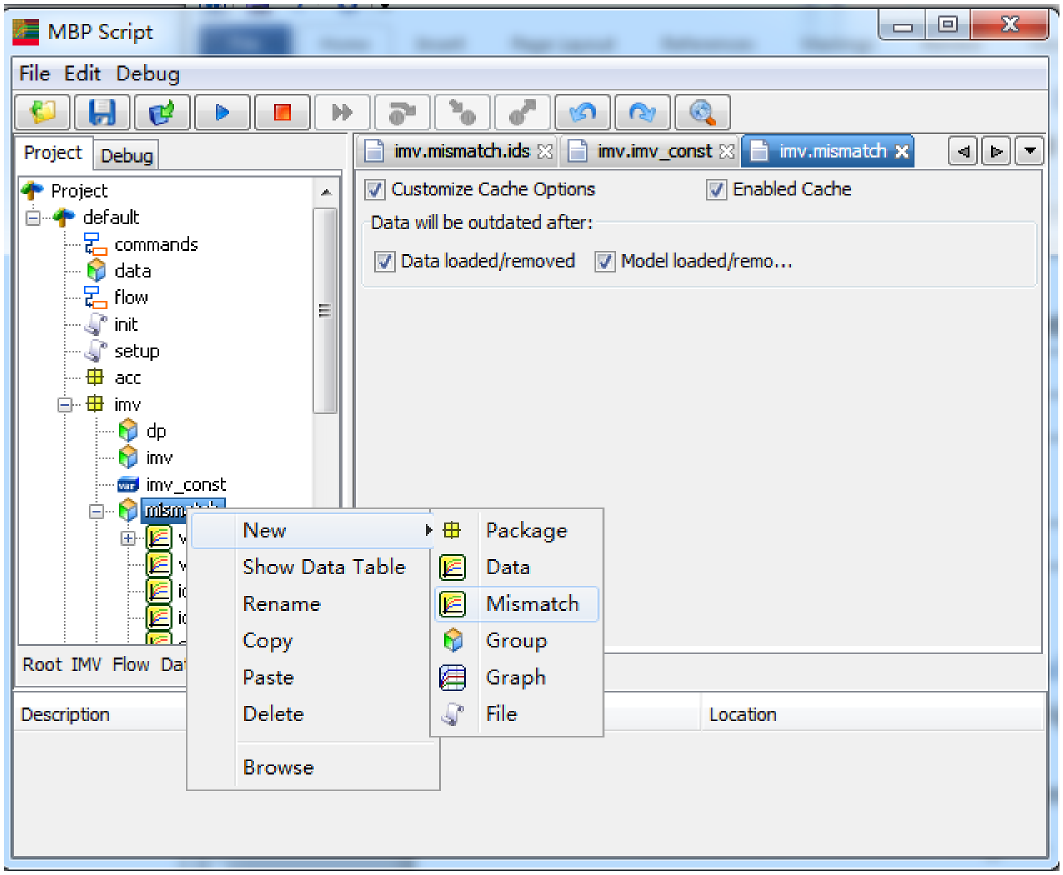
Fast Mode:	Use linear algorithm for statistical model extraction, fast speed;
Accurate Mode:	Use advanced optimization for statistical model extraction;
Advanced Mode:	Use advanced optimization for statistical model extraction, and user can set optimize-weight for nonlinear characters;
Corner Input Mode:	Input corner model and generate statistical model directly;

Mismatch/Statistical Target Define

Before model extraction, it should be make sure all mismatch and statistical targets have been defined in MBP correctly. MBP has defined some default targets and user also can define customized mismatch or statistical target with script.

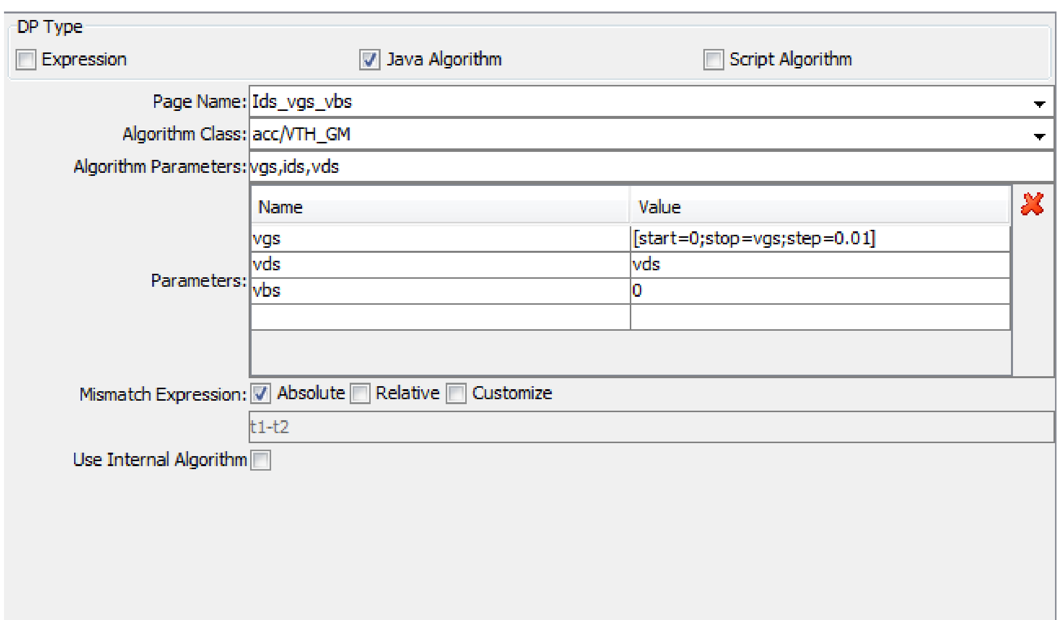
Mismatch Target

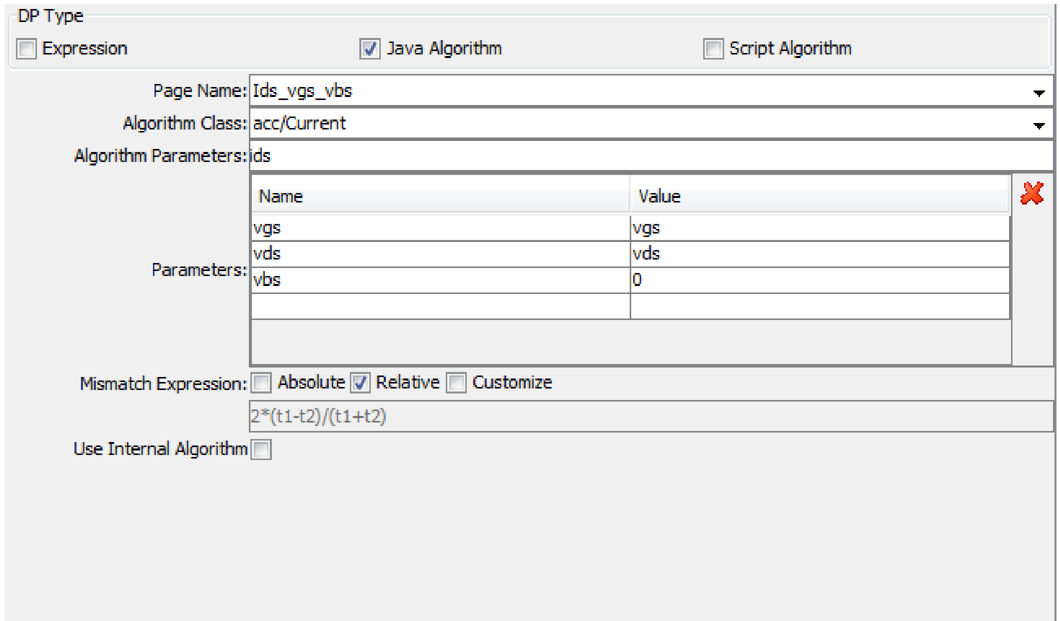
Select from menu Script > Script Project, MBP Script Windows will pop. Select Default > imv > mismatch in the left panel, right click and select new > mismatch to define new mismatch targets.



Set the new target's name in the popped window, and it can be edit in the right panel. How to edit target in MBP Script can reference MBP's help on IMV setting.

MBP supports both absolute and relative difference for mismatch targets.

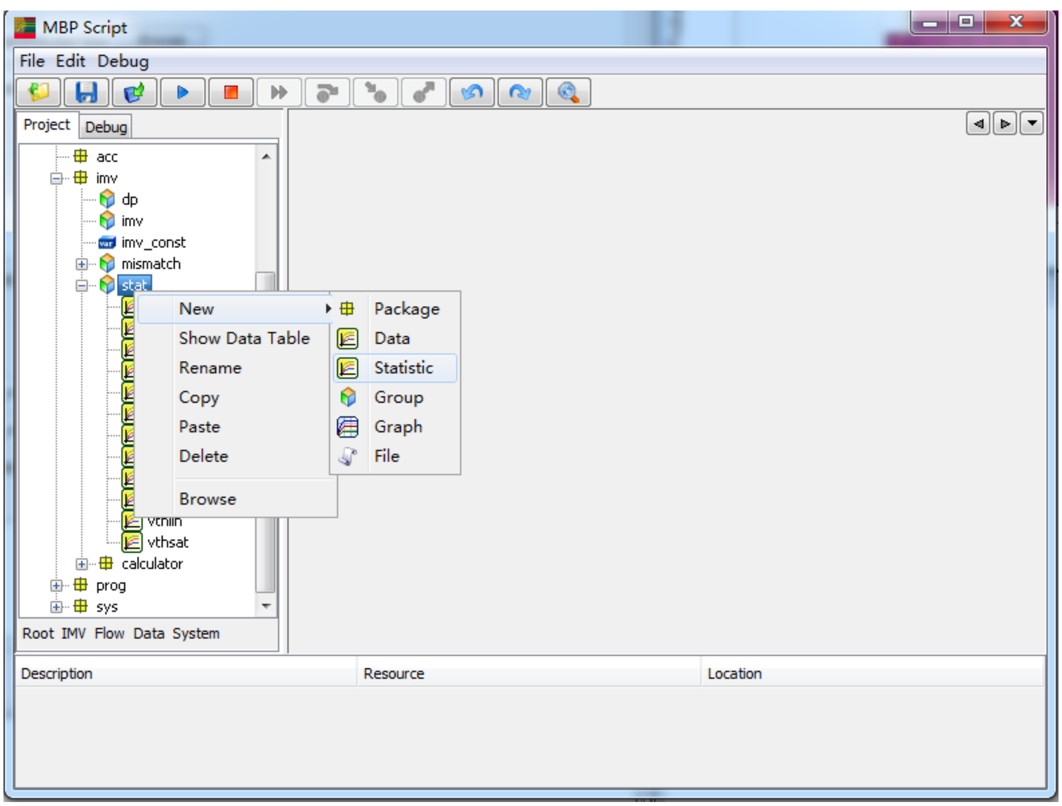




Mismatch Expression: Absolute and Relative

Statistical target

Select from menu Script > Script Project, MBP Script Windows will pop. Select default > imv > stat in the left panel, right click and select new > Statistic to define new statistical targets.



Set the new target's name in the popped window. It can be edit in the right panel. How to edit target in MBP Script can reference MBP's help on IMV setting.

Setup

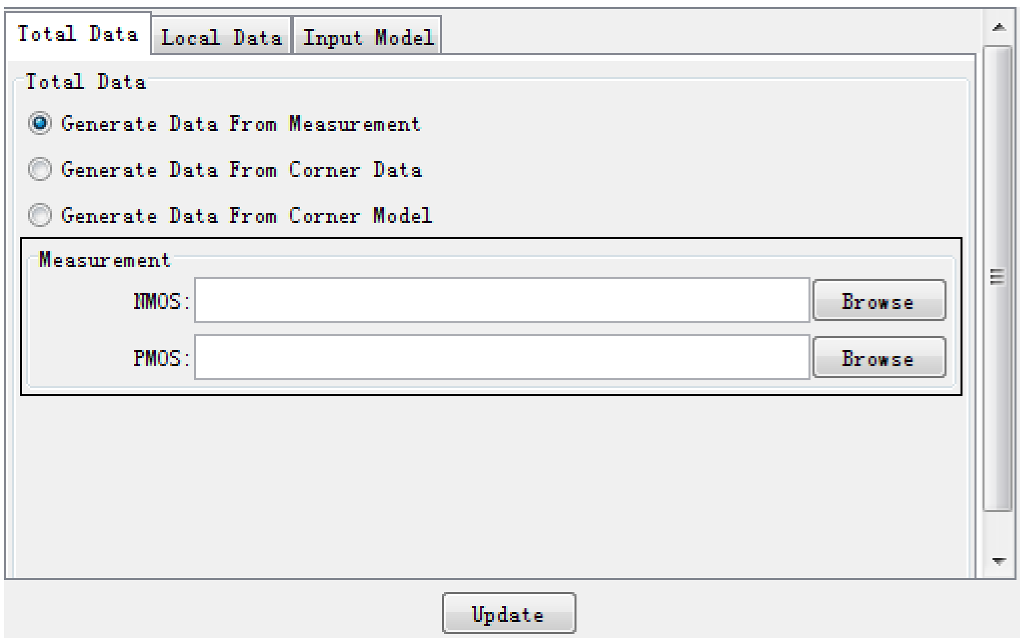
In "Setup" step, user will setup the input information. Three parts of this step:

- Total Data (refer to statistical data)
- Local Data (refer to mismatch data)
- Input Model

Total Data

User can select one from these three options

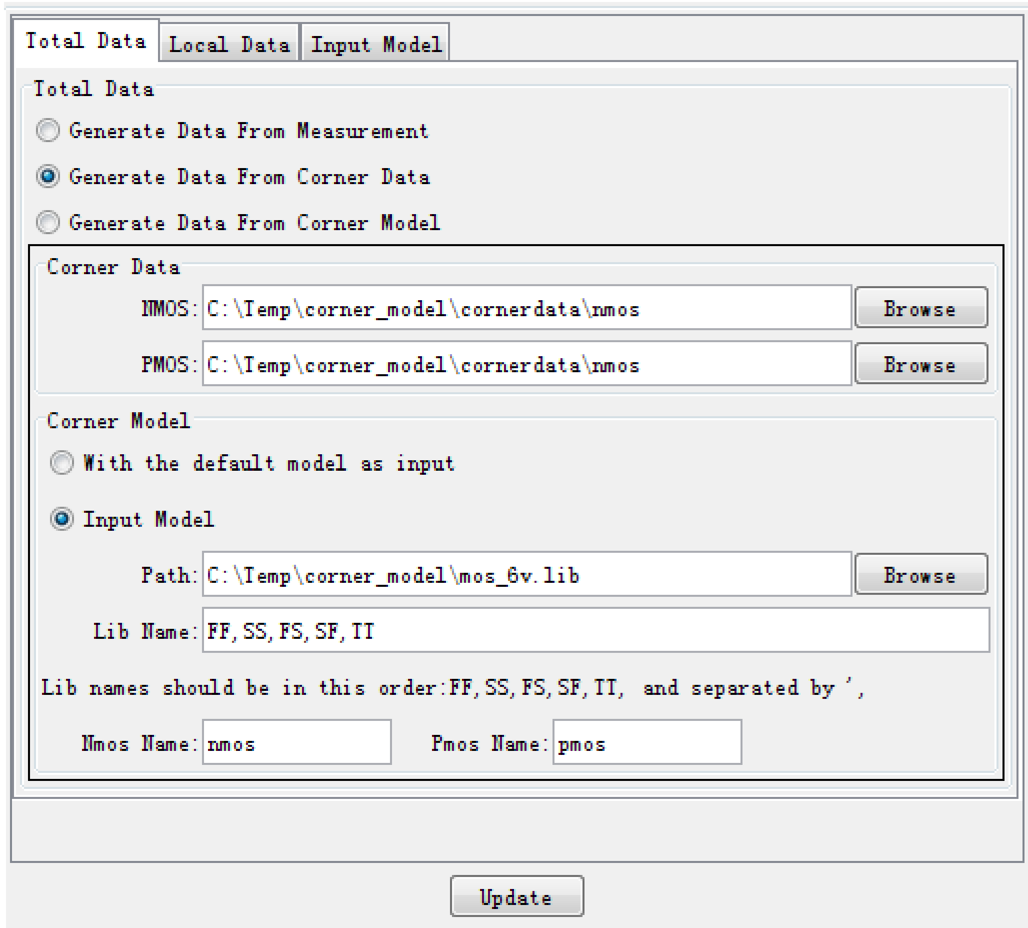
- Generate Data From Measurement



NMOS:	Click the browse button to select measurement data for nmos;
PMOS:	Click the browse button to select measurement data for pmos;

- Generate Data From Corner Data

Generate dummy statistical data from input corner data. The Corner/Statistical Scale parameter will be used in the dummy data generation.



Corner Data: the corner data file contains corner spec information;

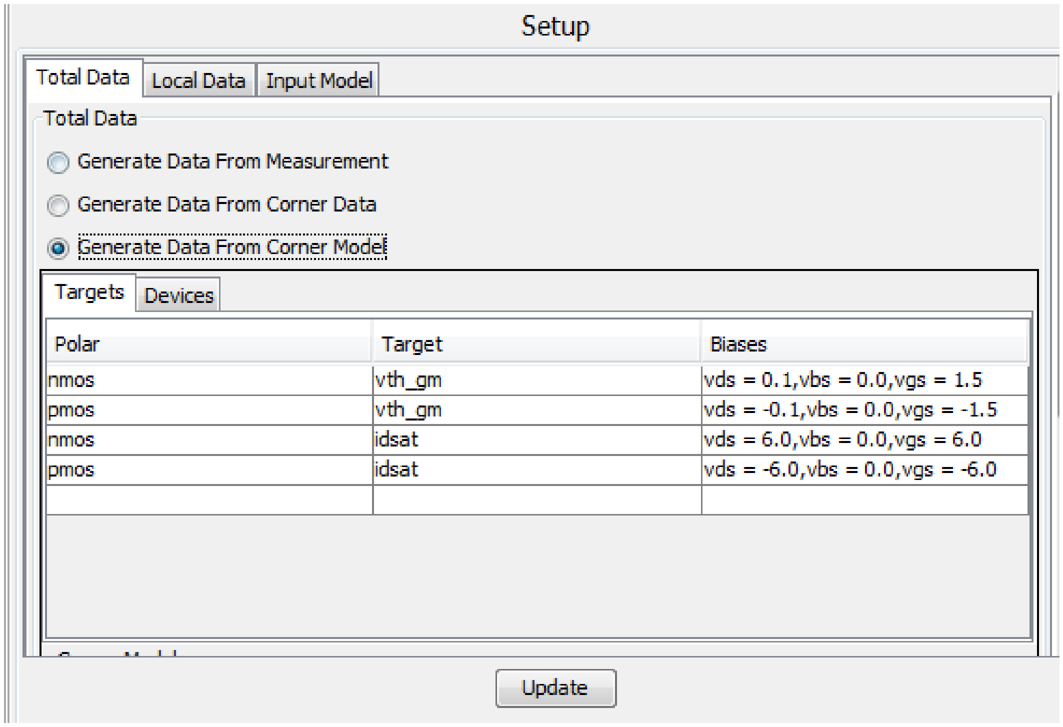
NMOS:	Click the browse button to load corner nmos data file;
PMOS:	Click the browse button to load corner pmos data file;

Corner Model: is not used for the model extraction, only for simulation result plot

With the default model as input:	If customer has no their own corner model, MBP's default corner model will be used in the following work;
Input Model:	Click browse button to select corner model;
Corner Model Names:	Corner models name for FF, SS, FS, SF, TT, and separated by comma;
NMOS Name:	nmos model name
PMOS Name:	pmos model name

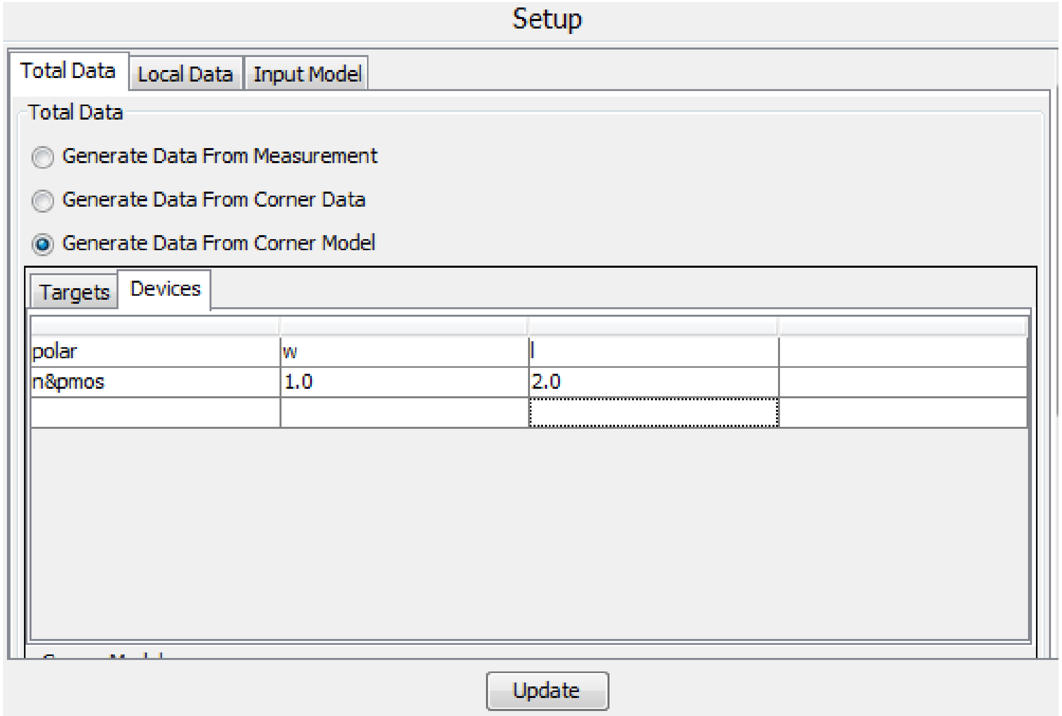
- Generate Data From Corner Model

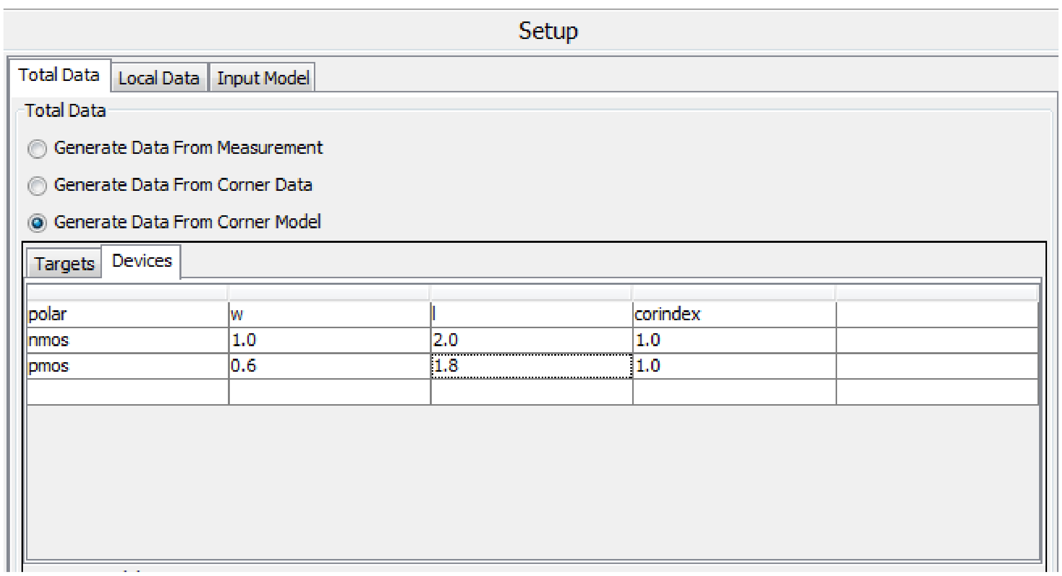
Generate dummy statistical data from input corner model's simulation result. The "Corner/Statistical Scale" parameter will be used in the dummy data generation.



Targets: statistical target definition

Polar:	nmos or pmos;
Target:	Targets Name;
Biases:	bias condition for this target;



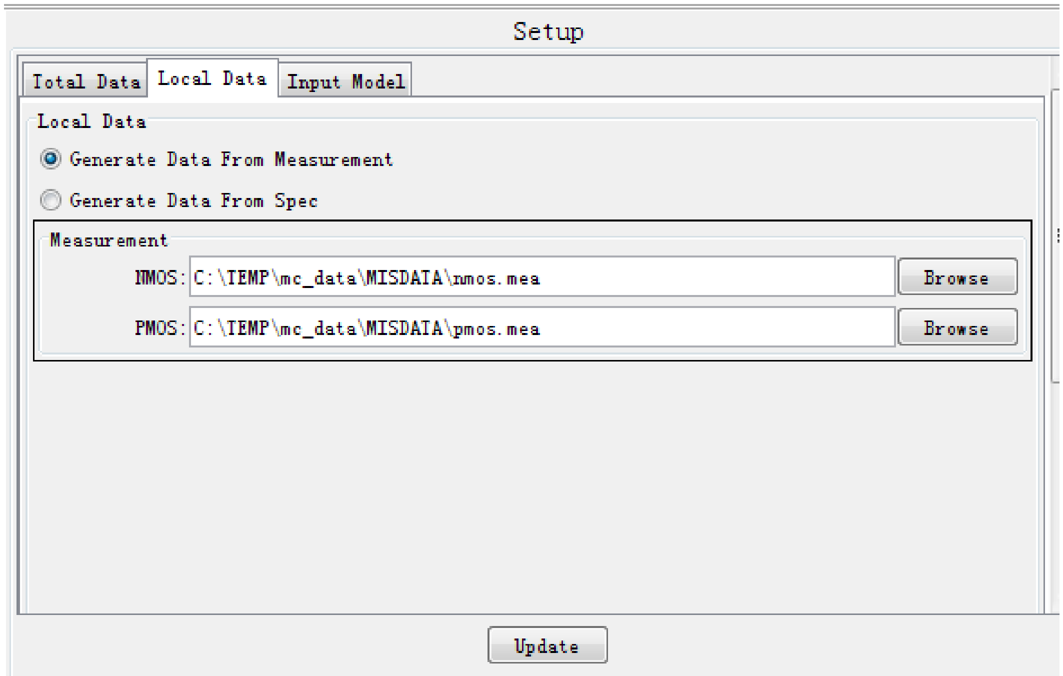


Devices: definition for devices instance parameters (w, l, temp, etc.)

Polar:	nmos or pmos or n&pmos;
	If select n&pmos, it means a pair of devices (both nmos and pmos) will be defined whose w=1 and l=2 as above picture.
	the nmos and pmos which has the identical corindex value are a pair of devices, where w=1 and l=2 for nmos, w=0.6 and L=1.6 for pmos;

Local Data

- Get Data From Measurement

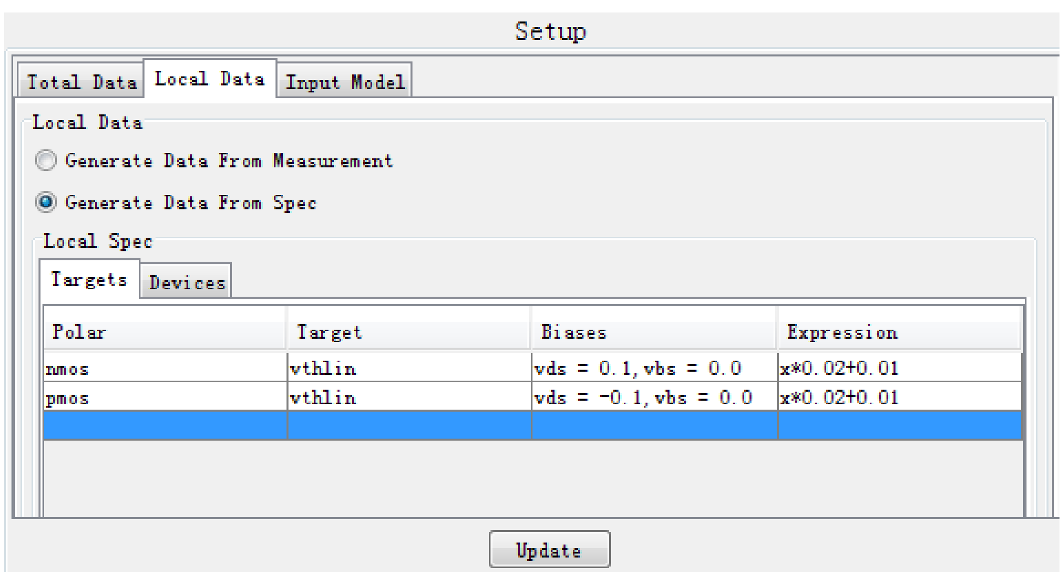


Measurement

NMOS:	Click browse button and select nmos mismatch measurement data;
PMOS:	Click browse button and select Pmos mismatch measurement data;

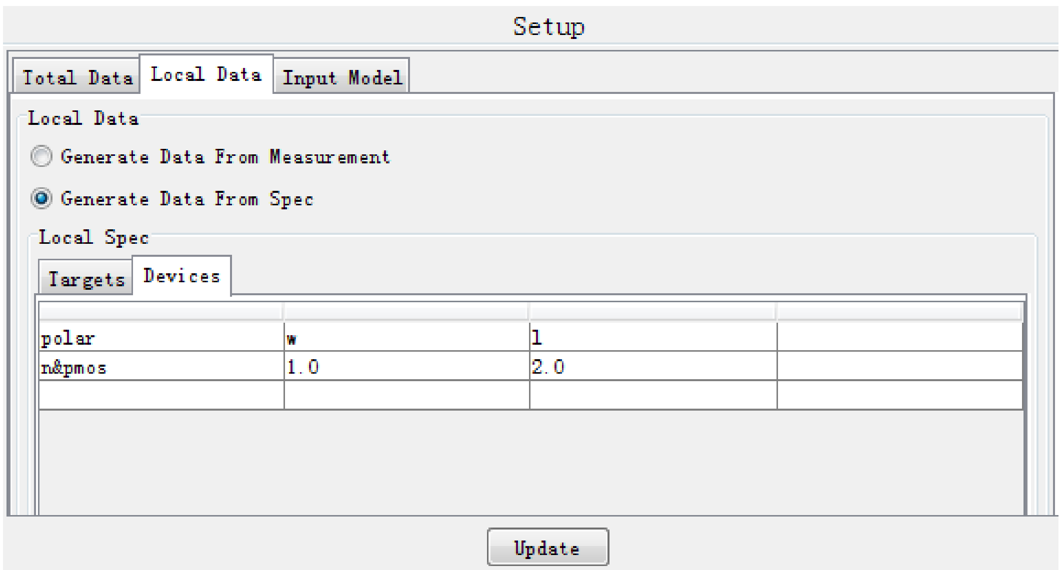
- Generate Data From Spec

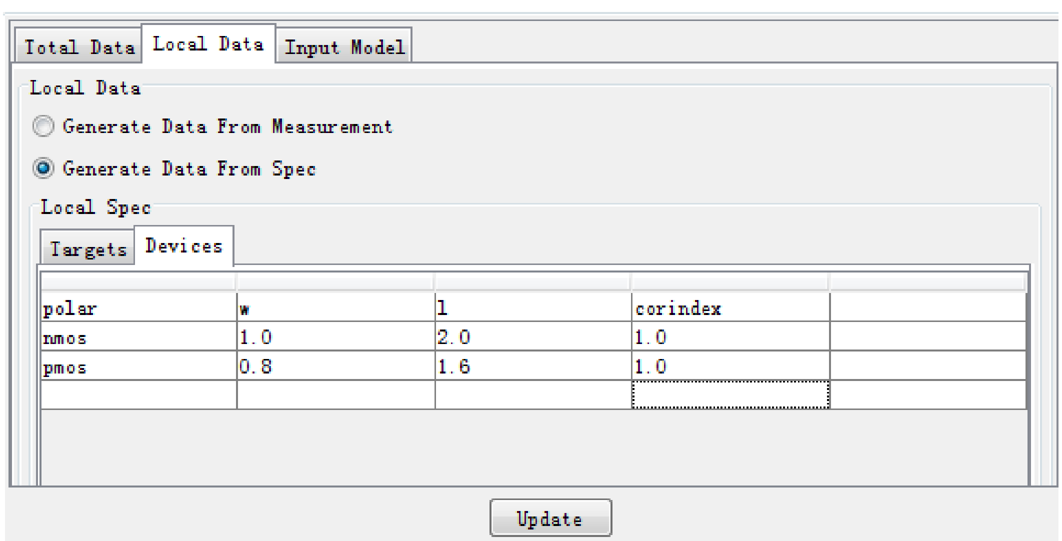
Generate dummy mismatch data from the information in the following table.



Targets: mismatch target definition

Polar:	nmos or pmos;
Target:	targets name;
Biases:	bias condition for this target;
Expression:	Target's sigma value= a*x + b; where, x=1/sqrt(w*l); a is slope coefficient; b is intercept value; we will use this express to calculate target value for every devices defined in "Devices" tab



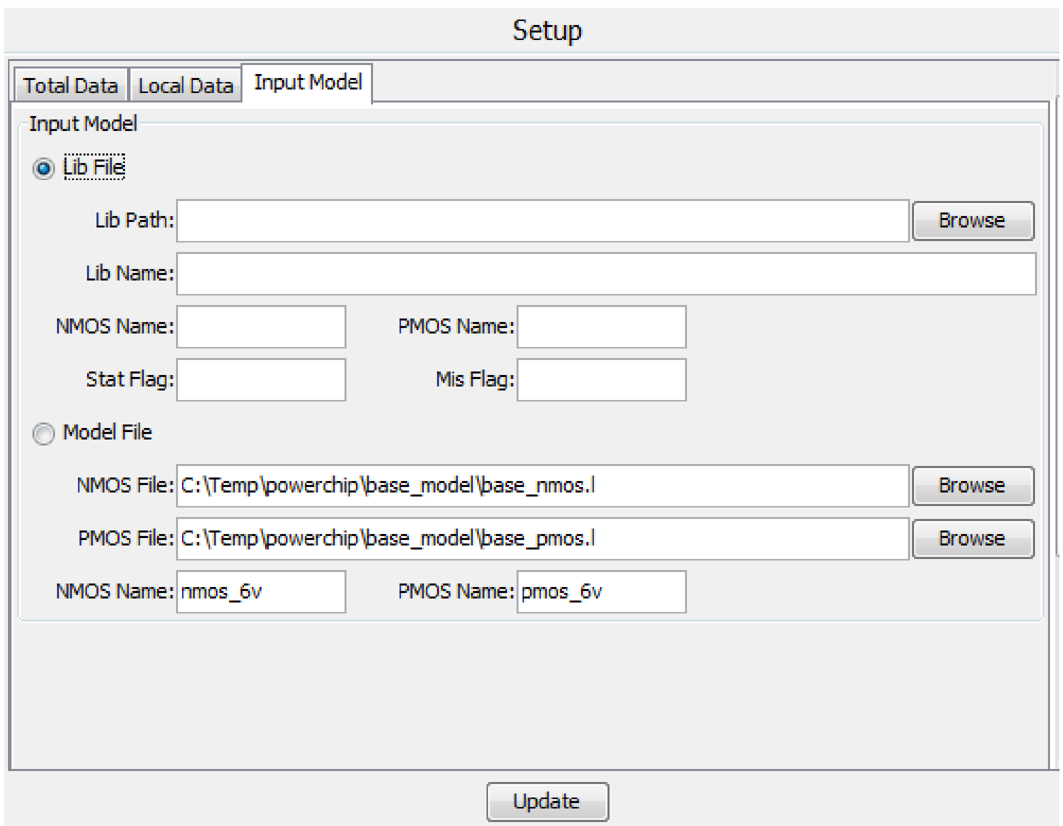


Devices: definition for devices instance parameters (w, l, temp, etc.)

Polar:	nmos or pmos or n&pmos;
	If select n&pmos, it means a pair of devices (both nmos and pmos) will be defined whose w=1 and l=2 as above picture.
	the nmos and pmos which has the identical corindex value are a pair of devices, where w=1 and l=2 for nmos, w=0.8 and L=1.6 for pmos;

Input Model Information

MBP statistical module accepts either model library or single model card.



Lib File: By using model library, please input library path, library name, NMOS model name, PMOS model name, stat flag, mis flag.

Lib Path:	Click browse button, and select the library model;
Lib Name:	Library's name;

NMOS Name:	nmos name;
PMOS Name:	Pmos name;
Stat flag:	The switch parameter in the model card, to switch *on/off* the statistical function of the model.For example, we set *Stat flag* as "staflag", if staflag=1, all statistical parameters works; if staflag=0, all statistical parameters disabled.
Mis flag:	The switch parameter in the model card, to switch *on/off* the mismatch capability of the model.For example we set *Mis flag* as "misflag", if misflag=1, all mismatch parameters works, if misflag=0, all mismatch parameters disabled.

Model File: By using baseband model, please input nmos file, pmos file, nmos name, pmos name.

NMOS File:	Click browse button, and select nmos baseband model;
PMOS File:	Click browse button, and select pmos baseband model;
NMOS Name:	Nmos model name;
PMOS Name:	Pmos model name;

In the setup, all generated data can be found at "project_folder\statistical\data".

Data Analysis

There are five steps in this section, four of them are optional.

Data Pruning

This option is to eliminate the data points beyond 3-sigma.

Data Sampling

This option is to minimize the data with "Group Size". For example, if "Group Size" is set to 10, MBP will select data every ten sites. If the entire data amount is too huge, user can speed up the progress by using this option.

Global Data Generation

This option is to generate global data from total and local data.

Mean Value Check

In case measurement total data's mean value is not equal to simulation total data's mean value, user could use this option to do data tweaking.
Two methods for the tweaking:

- *Tweak model*: Tweak the baseband model to fit the measurement, user need go to next step "Baseband Model" to select parameters for tweaking.
- *Tweak measurement total data*: MBP will adjust measurement data to fit model simulation. The tweaked data will be used in following steps.

Create Mismatch Dummy Data

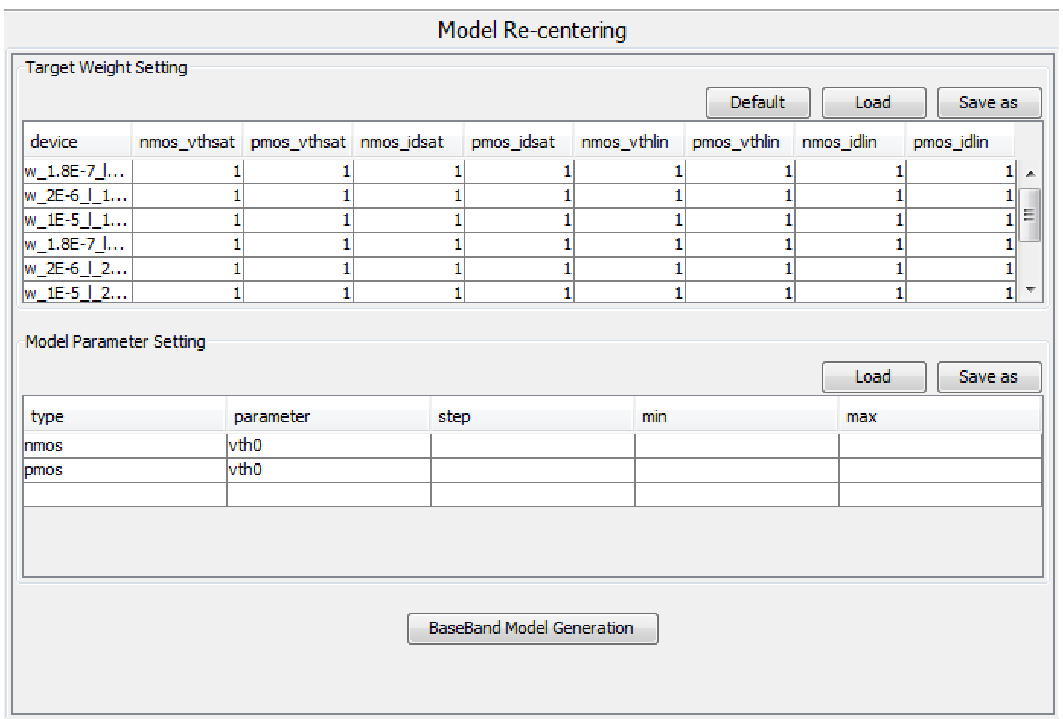
This step is to create mismatch dummy data:

- The dummy data is a straight line, replace the original point data.
- User can choose the straight line through origin point or not before the dummy data generation.
- The dummy data line will be the optimization target during following steps; it can give better trend fitting result than original data points.

Click "Generate Data" button after all the steps to execute data dispose.
In the Data Analysis, all generated data can be found at "project_folder\statistical\data".

Re-centering

Re-centering is used when "Tweak model" is selected at "Mean Value Check" steps, if user select "Tweak data to fit model", user could skip this step. The step is used to select parameters for model tweaking, as below.



Target Weight Setting

Click "Default", a weight table will appear. The left side of the table is device column, the upside of the table is target row, and all the default weights are set to 1.0.
User can set one value to the entire column or the entire row. By set the first

column (or the first row)'s weight and then right-click on this column (or row) select "Set value to the whole column" or "Set value to the whole row".
User can load pre-defined configuration file by "Load", also can save the configuration file by "Save as".

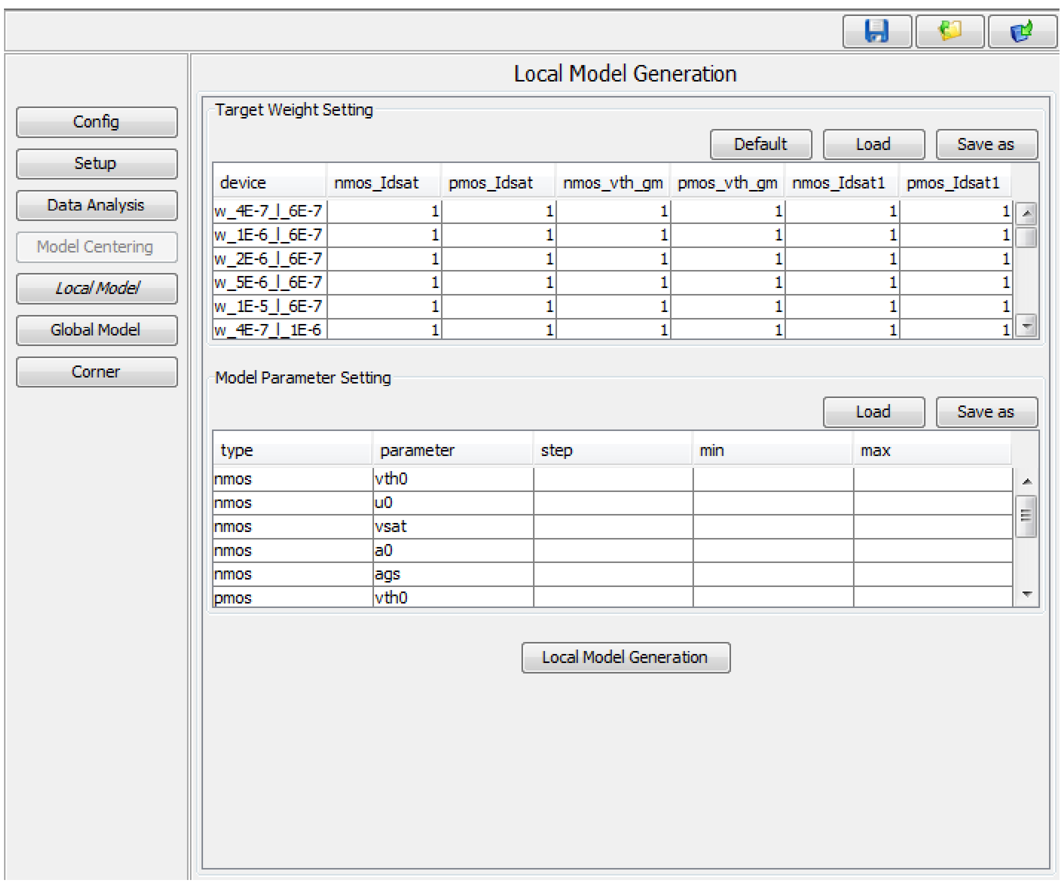
Model Parameter Setting

Fill in the parameters for model tweaking in this table, together with parameter's type ("nmos" or "pmos"), step, min/max value. Pre-defined configuration file can be loaded by "*Load*". A new configuration file can be saved by "*Save as*". Click "*Baseband Model Generation*", then the model will be tweaked automatically to fit measurement data
In the Re-Centering, the model can be found at "project_folder\statistical\output\baseband".

Local Model

Local Model contains two parts:

- Target Weight Setting
- Model Parameter Setting



Target Weight Setting

You can load your pre-defined configuration file by clicking "Load". And you can save the configuration file by clicking "Save as".

Target Weight Setting

Default | Load | Save as

device	nmos_Idsat	pmos_Idsat	nmos_vth_gm	pmos_vth_gm	nmos_Idsat1	pmos_Idsat1
w_4E-7_l_6E-7	1	1	1	1	1	1
w_1E-6_l_6E-7	1	1	1	1	1	1
w_2E-6_l_6E-7	1	1	1	1	1	1
w_5E-6_l_6E-7	1	1	1	1	1	1
w_1E-5_l_6E-7	1	1	1	1	1	1
w_4E-7_l_1E-6	1	1	1	1	1	1

Model Parameter Setting

Two different table formats will appear depend on what format of model we start with: model library or single model card.
For model library, the table shows as below:

Model Parameter Setting

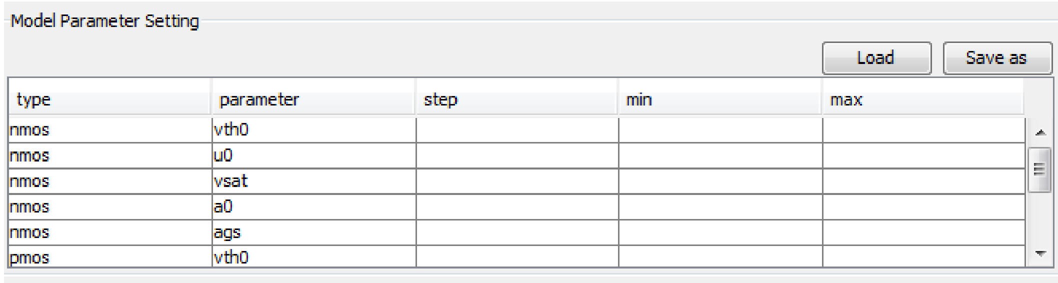
Load | Save as

random	mean	sigma	parameter	step	min	max
misrad_misnmos1	0	1	vth0_misnmos_k0			
misrad_misnmos2	0	1	u0_misnmos_k0			
misrad_misnmos3	0	1	toxe_misnmos_k0			
misrad_mispmos4	0	1	vth0_mispmos_k0			
misrad_mispmos5	0	1	u0_mispmos_k0			
misrad_mispmos6	0	1	toxe_mispmos_k0			

Fill in this table with user-defined mismatch model parameters and corresponding random parameters, the mean and sigma value of the parameter are "1" and "0" by default respectively.
Setup your model library as the MBP's demo example. Below are some useful tricks in the model parameter setting:

- Add ":" after parameter name, it means this parameter's input value will be the initial value for next step
- Add "=" after parameter name, it means this parameter's value will always keep constant
- If the input lib is complex, and the relation between the random and parameter is not very clear, please add "?" at first line and add the tuning parameters in the following lines.

If single model card is used, the table is as below:



The table is very similar to "Baseband Model". User need fill this table with parameters for tweaking, and specify parameters' polar type; polar type can be "NMOS" or "PMOS". User can also assign parameter's step, min value and max value.
After setting up all the target weights and model parameters, click "*Local Model Generation*" button, then the mismatch model will be generated automatically.

Key points in the local model extraction

Firstly, selecting right parameters for the local model extraction is most important. Parameter selection depends on the mismatch targets and device's dimension. If simulation result is failed to converge, please reset parameter's min/max value and try again.
Secondly, set weight matrix to tradeoff between different optimize targets.
Finally, switch extraction methods or tweak optimization parameter to help optimization converge at better result.
Moreover, open debug mode and read log file can help trouble-shooting.
The extracted local model can be found at "project_folder\statistical\output\local".

Global Model

Before go to this step, make sure "Data Dispose" is done. This section is used to generate global statistical model automatically. Three steps to complete a "Global Model" table:

- Target Weight Setting
- Correlation Weight Setting
- Model Parameter Setting

Target Weight Setting

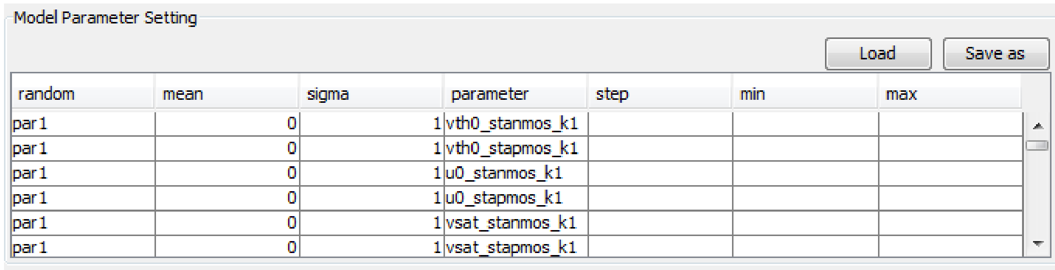
User is suggested to use the default setting by click "*Default*" button. Different weight can be set to different target, MBP will put more effort on the target with high weight Configuration file can be saved and load by "*Save as*" and "*Load*" button.

Correlation Weight Setting

If both NMOS and PMOS data is loaded within one extraction progress, user are enabled to set weight for NMOS and PMOS correlation.MBP put more effort to the correlation with higher weight, may have better fitting accuracy normally.

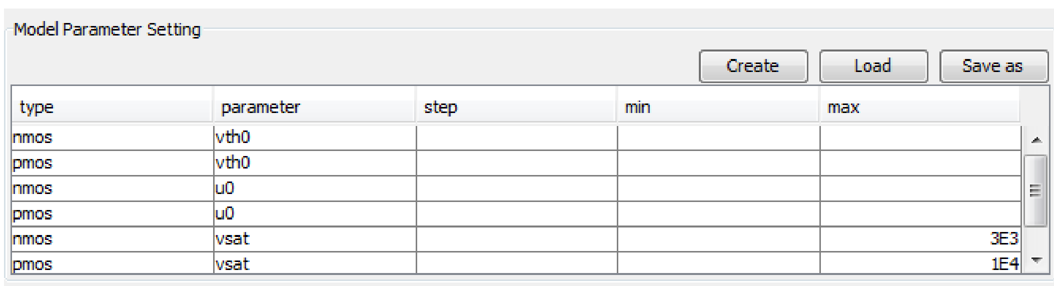
Model Parameter Setting

The table has two forms depending on the input model format.
Library model, the table is as below:



random	mean	sigma	parameter	step	min	max
par1	0	1	vth0_stanmos_k1			
par1	0	1	vth0_stapmos_k1			
par1	0	1	u0_stanmos_k1			
par1	0	1	u0_stapmos_k1			
par1	0	1	vsat_stanmos_k1			
par1	0	1	vsat_stapmos_k1			

User need fill in this table with user-defined statistical model parameters and corresponding random parameters. Parameters' mean and sigma value are "1" and "0" by default respectively. User can also assign parameter's step, min/ max value. The configuration file can be save and load by "*Save as*" and "*Load*" button.
If single model card is used, the table is as below:



type	parameter	step	min	max
nmos	vth0			
pmos	vth0			
nmos	u0			
pmos	u0			
nmos	vsat			3E3
pmos	vsat			1E4

User need to fill in this table with parameters for tweaking, and specify parameters' corresponding polar type. User can also assign parameter's step, min/max value. The configuration file can be saved and load by "Save as" and "Load" button.

- Click "*Global Model Generation*" button after completed all setups, then the global statistical model will be generated automatically. There are some different algorithm will be used when the global model generation.

1. a. No device is selected in the target weight setting table

All devices will be taken into account in the statistical model extraction. And "global" statistical model will be extracted.

1. a. If only one device is selected in the target weight setting table

Only selected device will be taken into account in the statistical modeling extraction. And "point" statistical model will be extracted.

1. a. All devices are selected in the target weight setting table

All extracted "point" statistical model will be merged into a "global" statistical model

The extracted statistical model can be found at "project_folder\statistical\output\global".

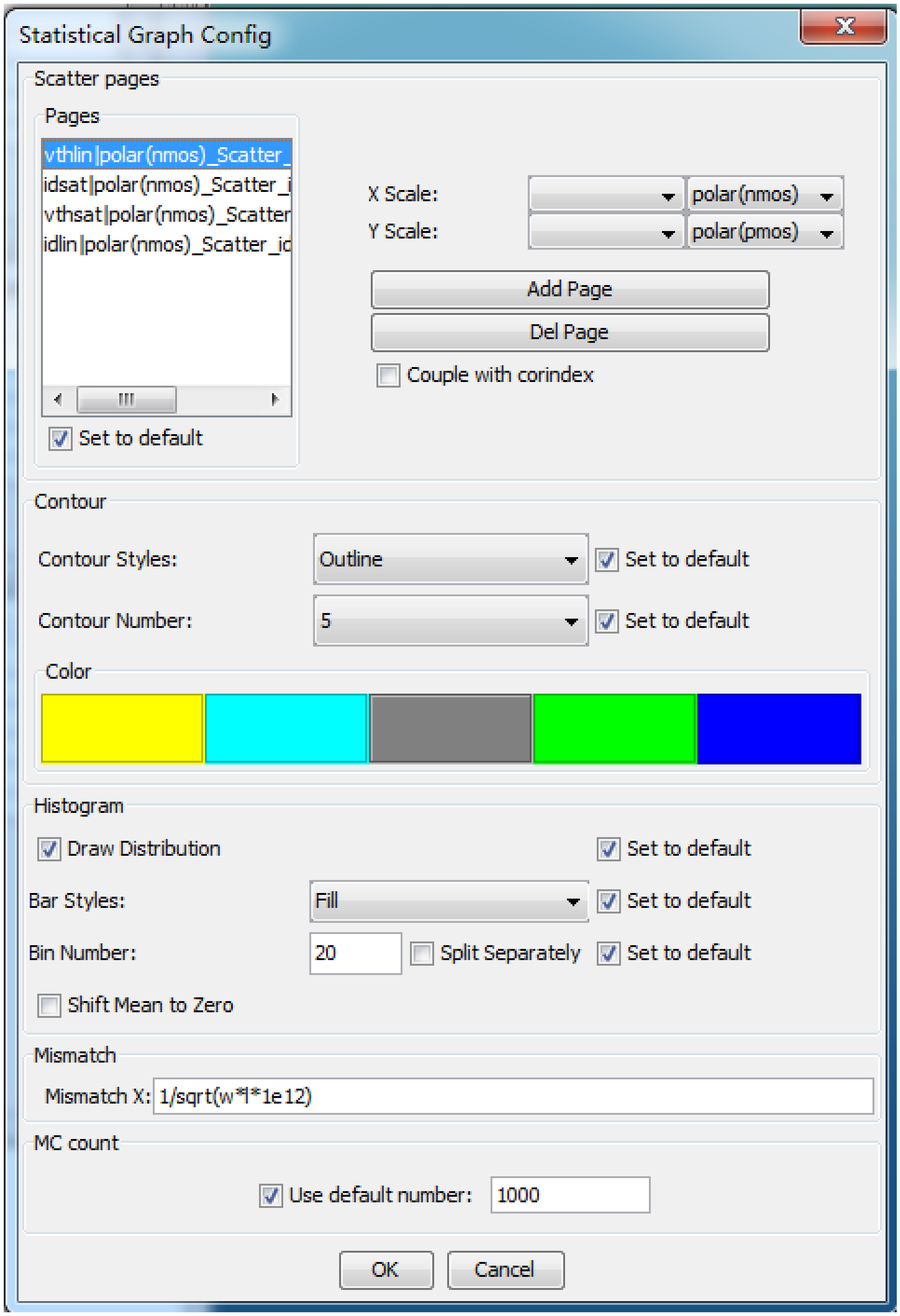
Key points in global model extraction

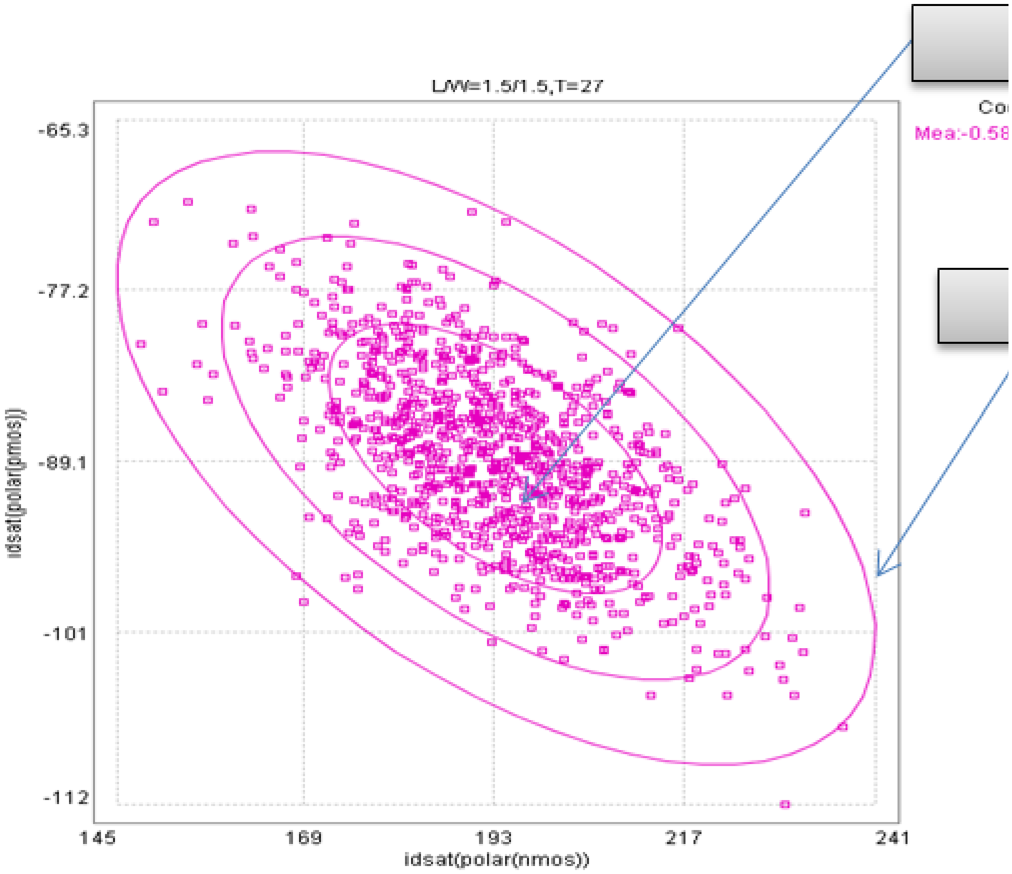
Firstly, selecting right parameters for the global model extraction is most important. Parameter selection depends on the statistical targets and device's dimension. If simulation result is failed to converge, please reset parameter's min /max value and try again.
Secondly, set weight matrix to tradeoff between different optimize targets.
Finally, switch extraction methods or tweak optimization parameter to help optimization converge at better result. For example, user can select one device and extract point model one by one. It is a good way to find the cause for the problem. Moreover, open debug mode and read log file can help trouble-shooting.
Ps: how to de-select the device? Please press "ctrl" button and right click mouse to de-select the specific devices.

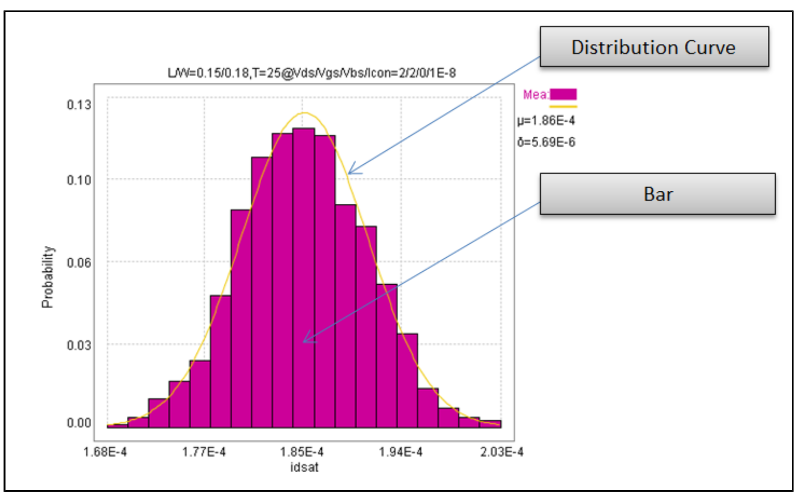
Graph Configure

Statistical Graph Configure

Select from menu Tool > GUI Options > Statistical Graph Config, and a setup window is popped.


Statistical Graph Config
Scatter pages
Pages
vthlin|polar(nmos)_Scatter_
idsat|polar(nmos)_Scatter_i
vthsat|polar(nmos)_Scatter
idlin|polar(nmos)_Scatter_id
X Scale:
polar(nmos)
Y Scale:
polar(pmos)
Add Page
Del Page
Couple with corindex
Set to default
Contour
Contour Styles:
Outline
Set to default
Contour Number:
5
Set to default
Color
Histogram
Draw Distribution
Set to default
Bar Styles:
Fill
Set to default
Bin Number:
20
Split Separately
Set to default
Shift Mean to Zero
Mismatch
Mismatch X:
1/sqrt(w*l*1e12)
MC count
Use default number:
1000
OK
Cancel





Scatter Pages:

X Scale:	Target and polar selection for X-axial;

Y Scale:	Target and polar selection for Y-axial;
Pages:	Click Add Page, the plot defined by X Scale and Y Scale will be add into page; Select item in the Page List and click Del Page, the selected plot will be deleted from list.
Couple with corindex:	Nmos and Pmos device will be grouped by corindex;

Contour:

Contour Styles:	Contour Curve Plot Styles;
Contour Numbers:	Contour Curve number, default number is 5;
Color	Contour Curve Plot Color;

Histogram:

Draw Distribution:	Draw theoretical distribution curve or not
Bar Styles:	
Bin Number:	Bin Number in the histogram plot
Shift Mean to Zero:	

Mismatch:

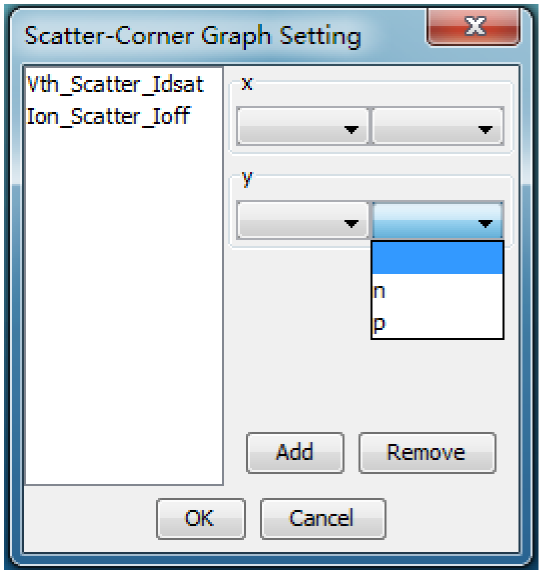
Mismatch X:	Calculation expression for x axial in mismatch plot

MC Count:

MC Count:	The random numbers in Monte Carlo Simulation

Scatter-Corner Graph Config

Select from menu Tool > GUI Options > Scatter-Corner Graph Config, and a setup window is popped.

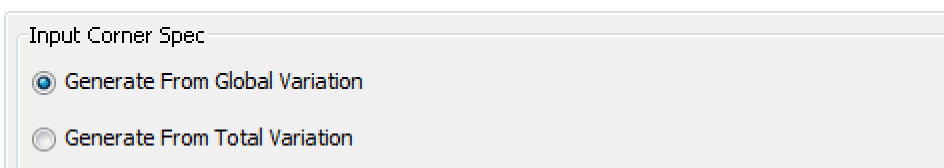


X:	Target and Polar for X-axial
Y:	Target and Polar for Y-axial

Corner Analysis

Make corner analysis for the customer input to generate corner spec and corner model template.

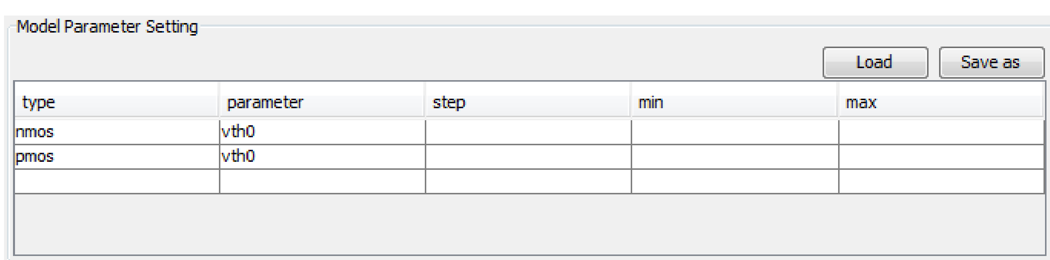
Generate corner spec



Generate corner spec from global variation data;
Generate corner spec from total variation data;

Model parameter setting

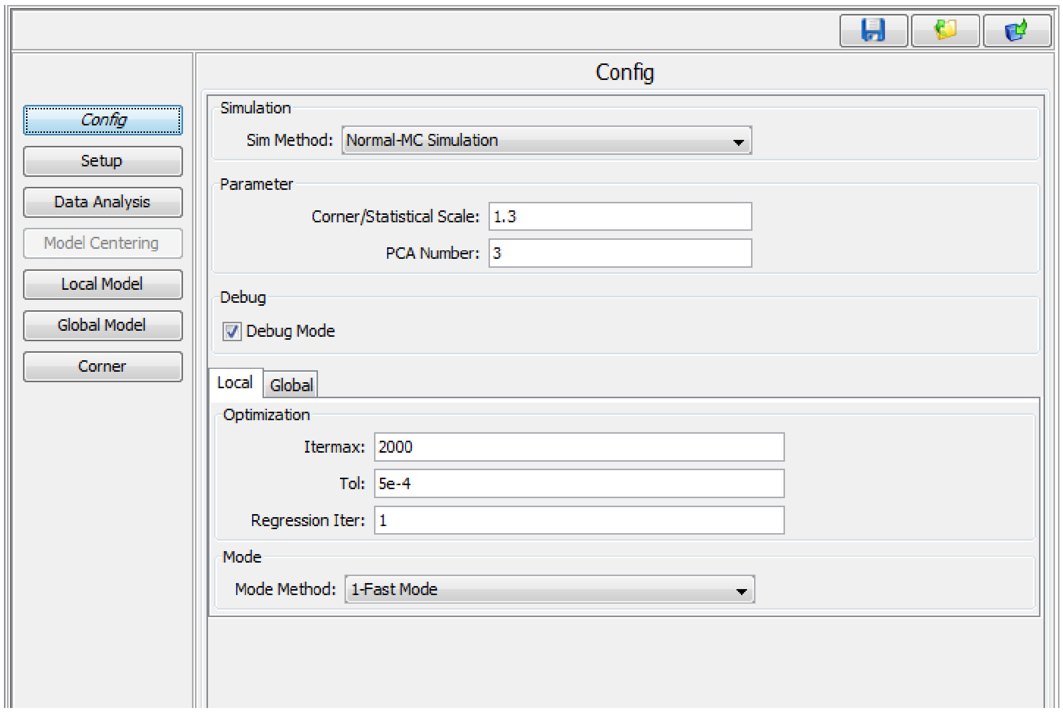
Model parameter setting for corner model template generation.



Mismatch and Statistical model extraction from data – Baseband Model.

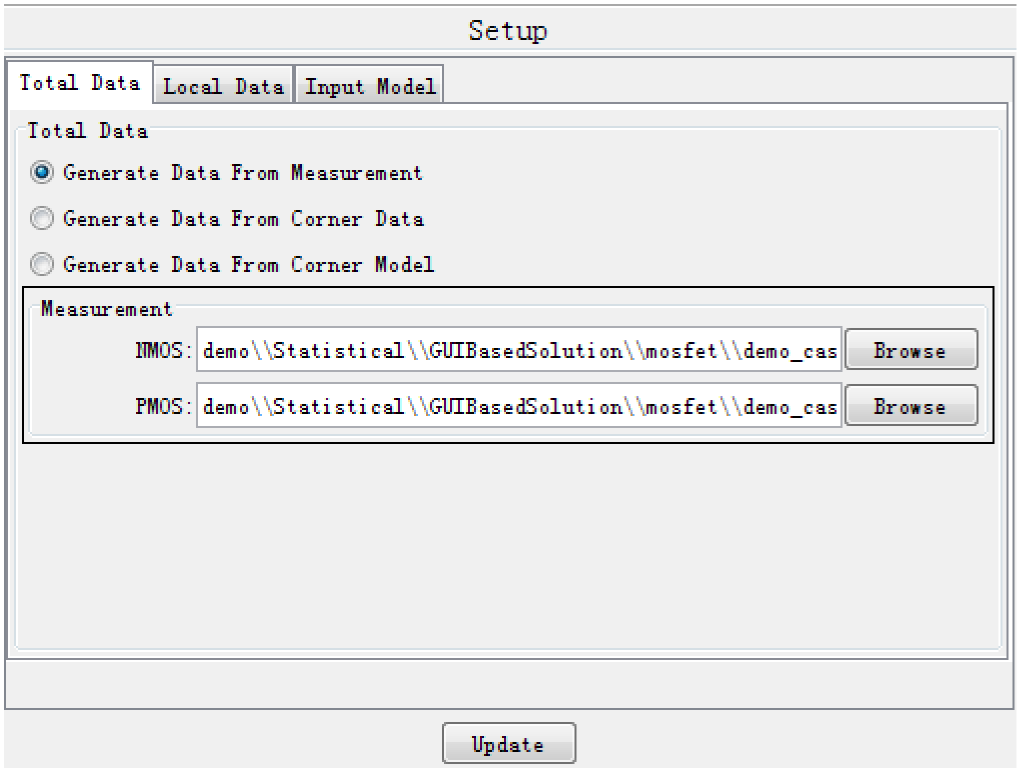
Open the project from "MBP_home\demo\Statistical\GUIBasedSolution\mosfet\demo_case_baseband"

1. a. Config

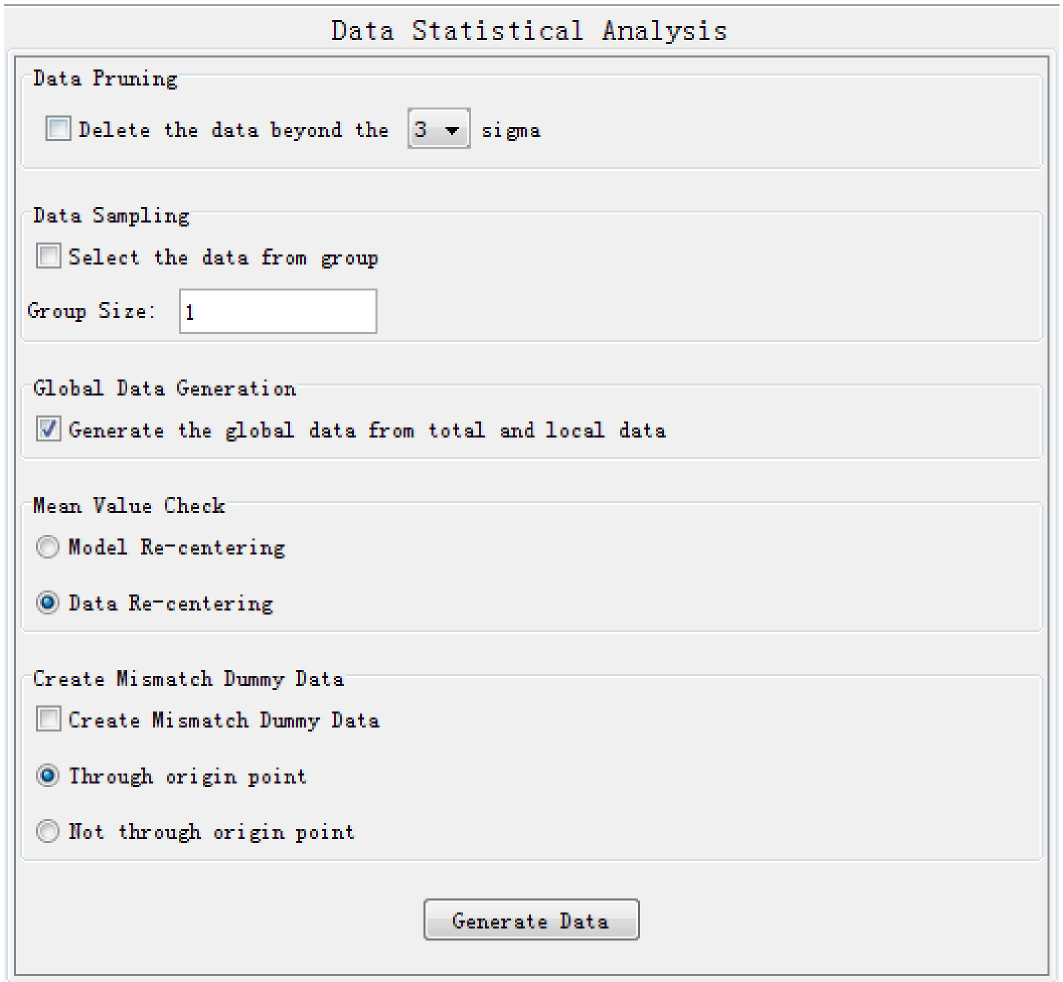


1. a. Setup

The measurement for this demo case can be found "project_path\MISDATA" and "project_path\STADATA"

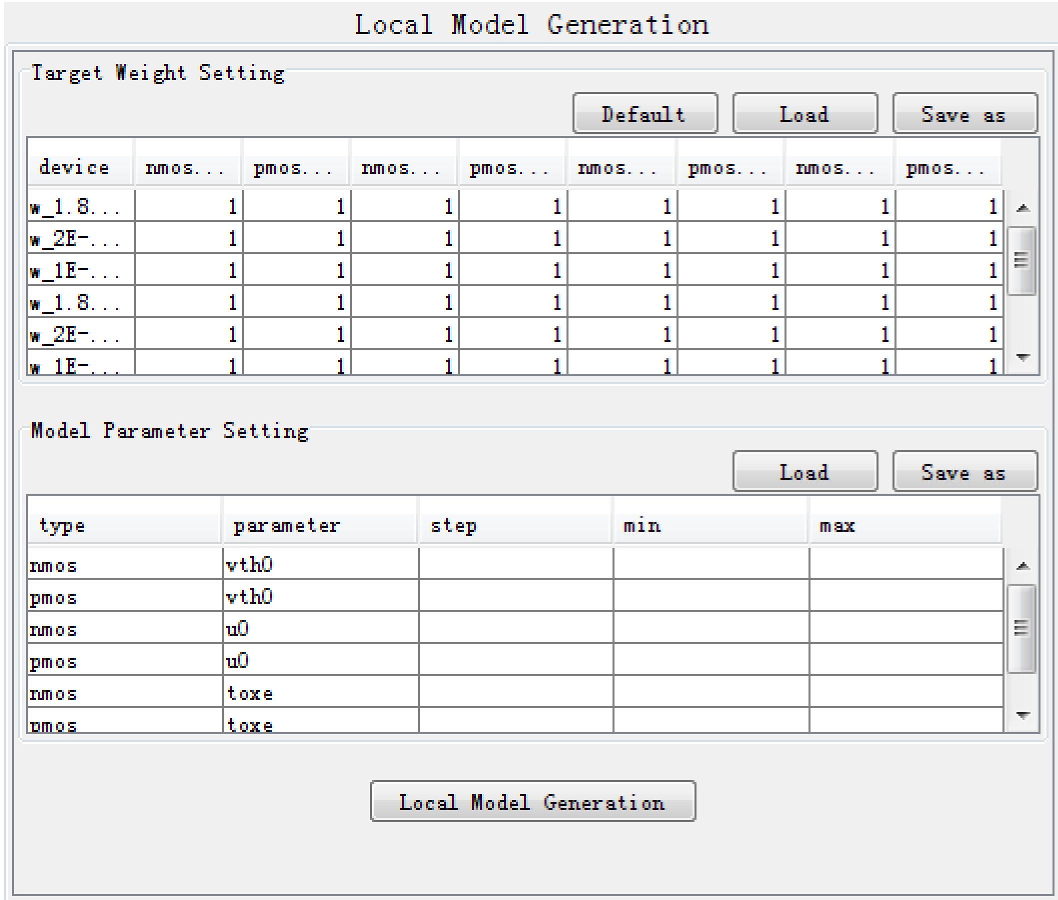


1. a. Data Dispose



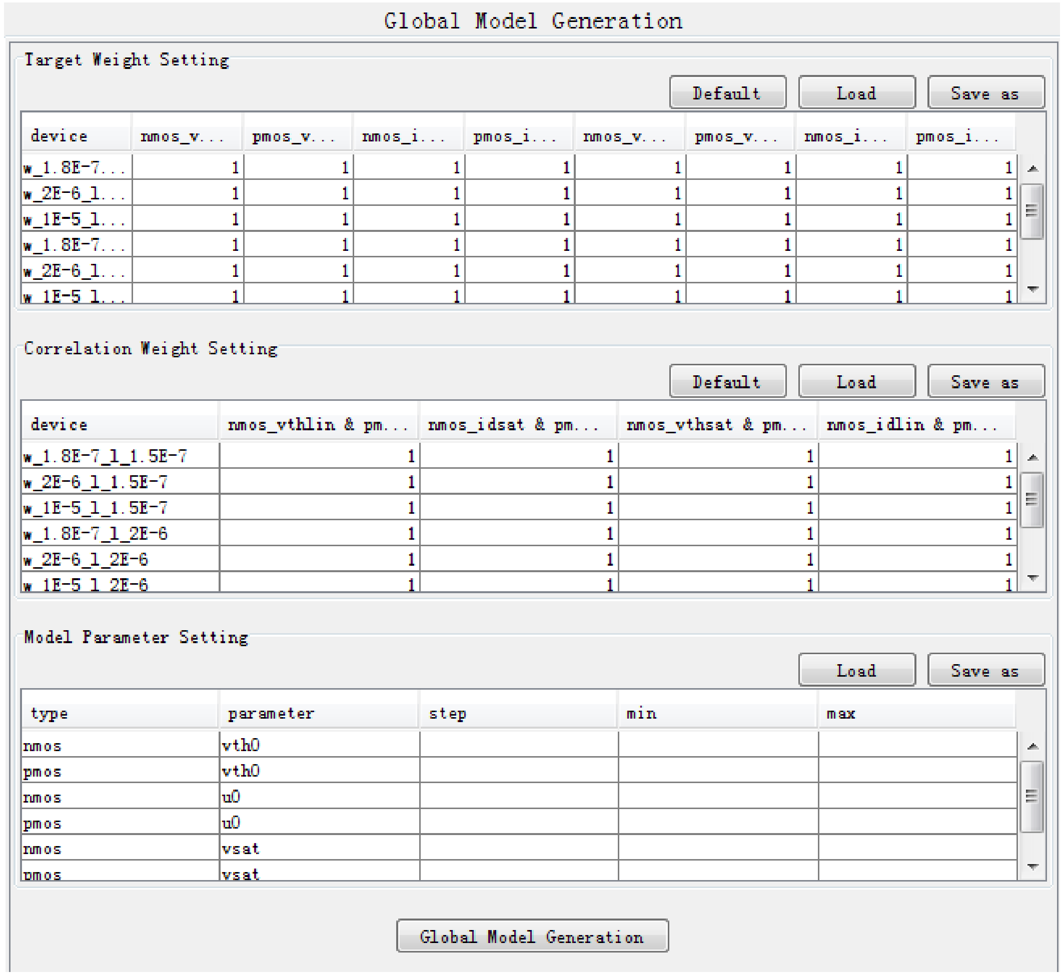
Click the "Generate Data" for data dispose.

1. a. Local Model



Set target weight table and parameter table, and click "Local Model Generation" button to extract local model

1. a. Global Model



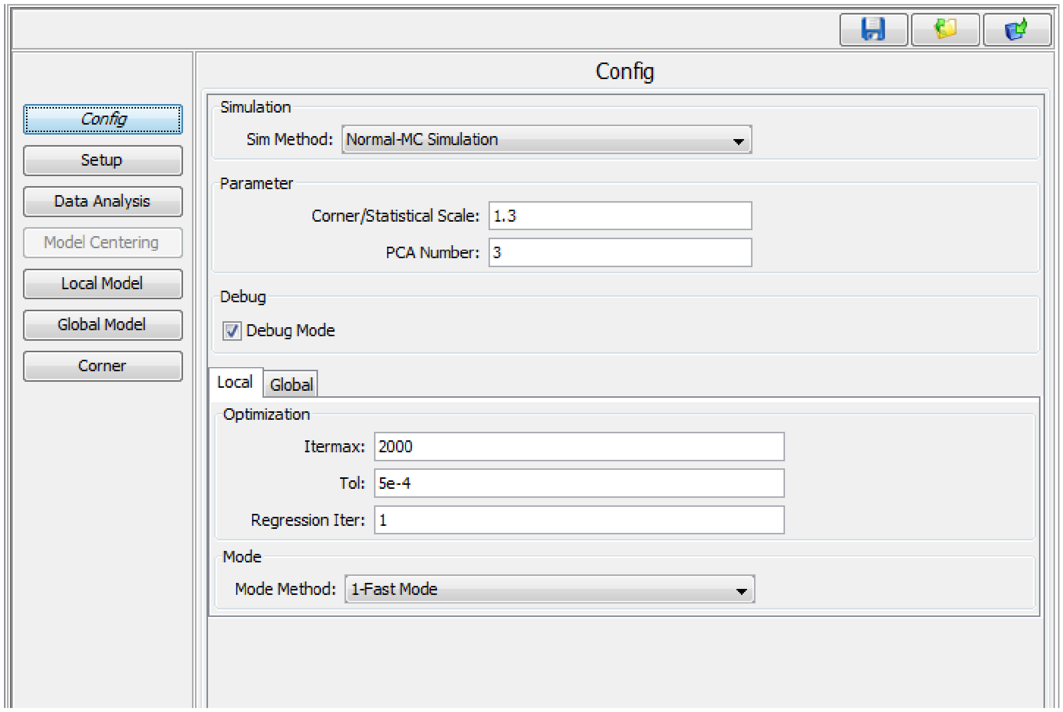
Set target weight table, correlation weight table and parameter table, and click "Global Model Generation" button to extract global model.

The generated dummy data and extracted model are saved in the "project_path\mosfet_demo_2\statistical";

Mismatch and Statistical model extraction from data - Model Library.

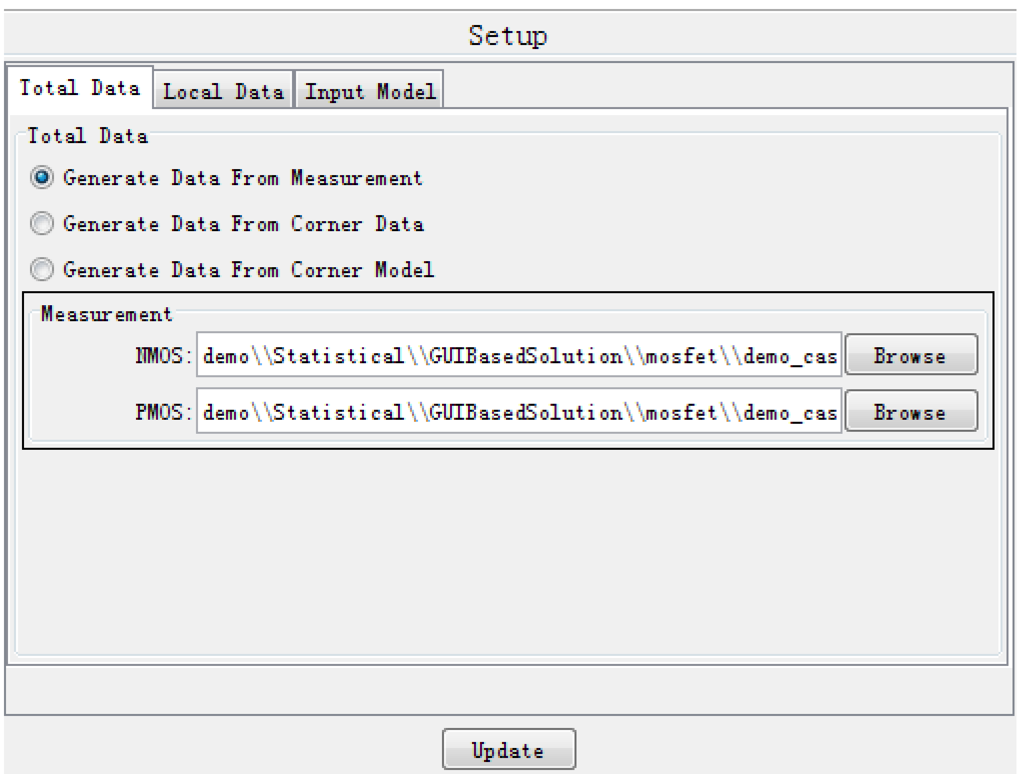
Open the project from "MBP_home\demo\Statistical\GUIBasedSolution\mosfet\demo_case_library"

1. a. Configuration

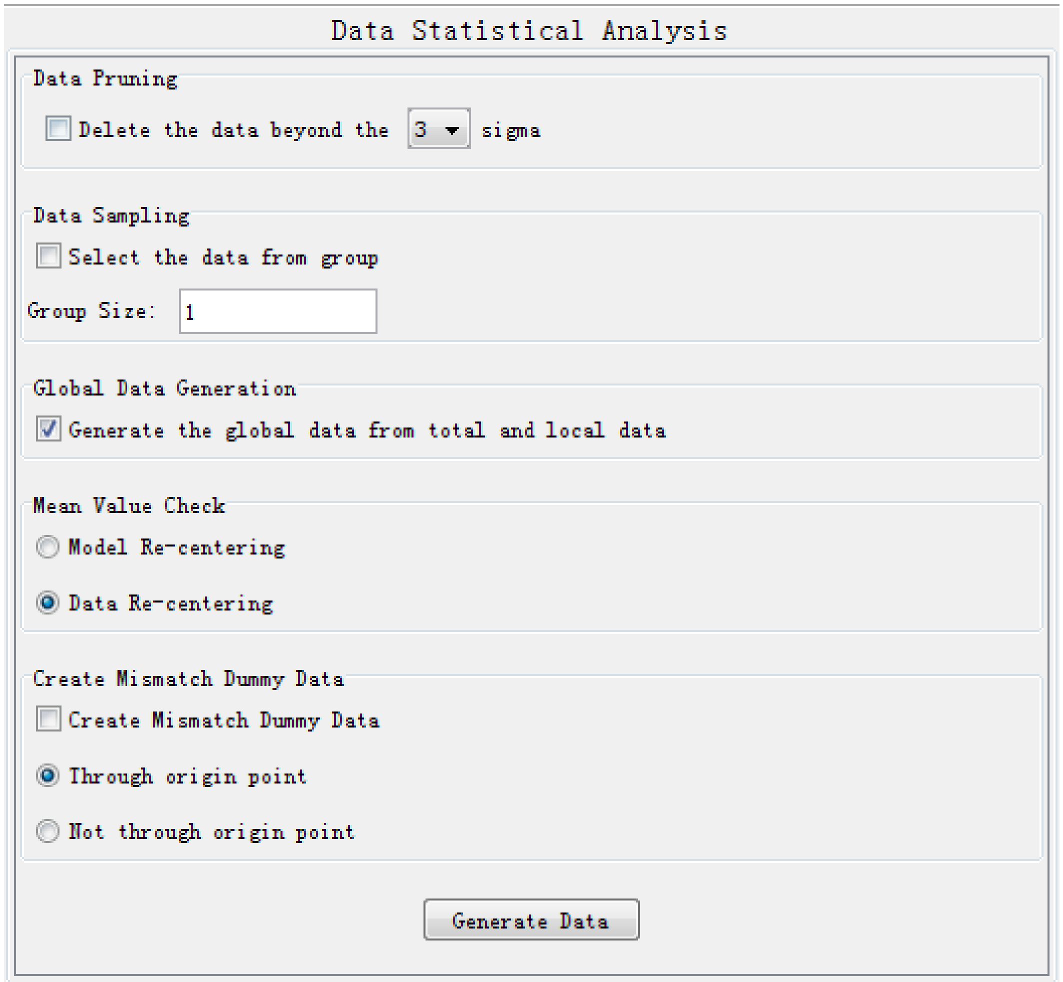


1. a. Setup

The measurement for this demo case can be found "project_path\MISDATA" and "project_path\STADATA"

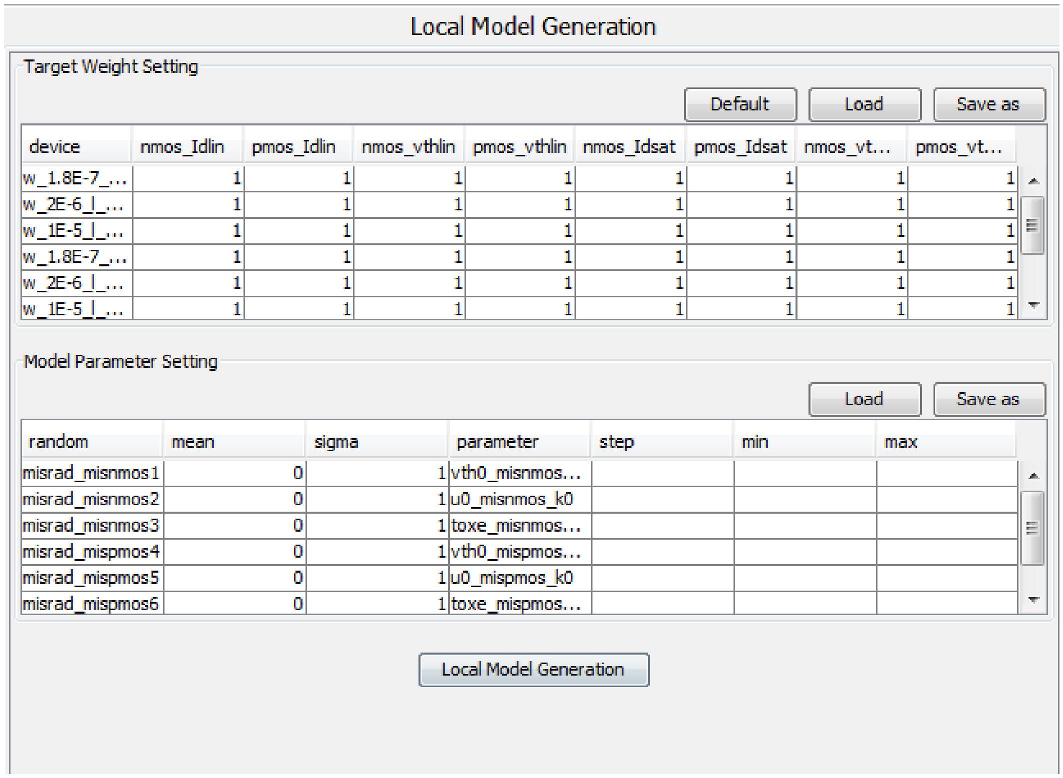


1. a. Data Dispose



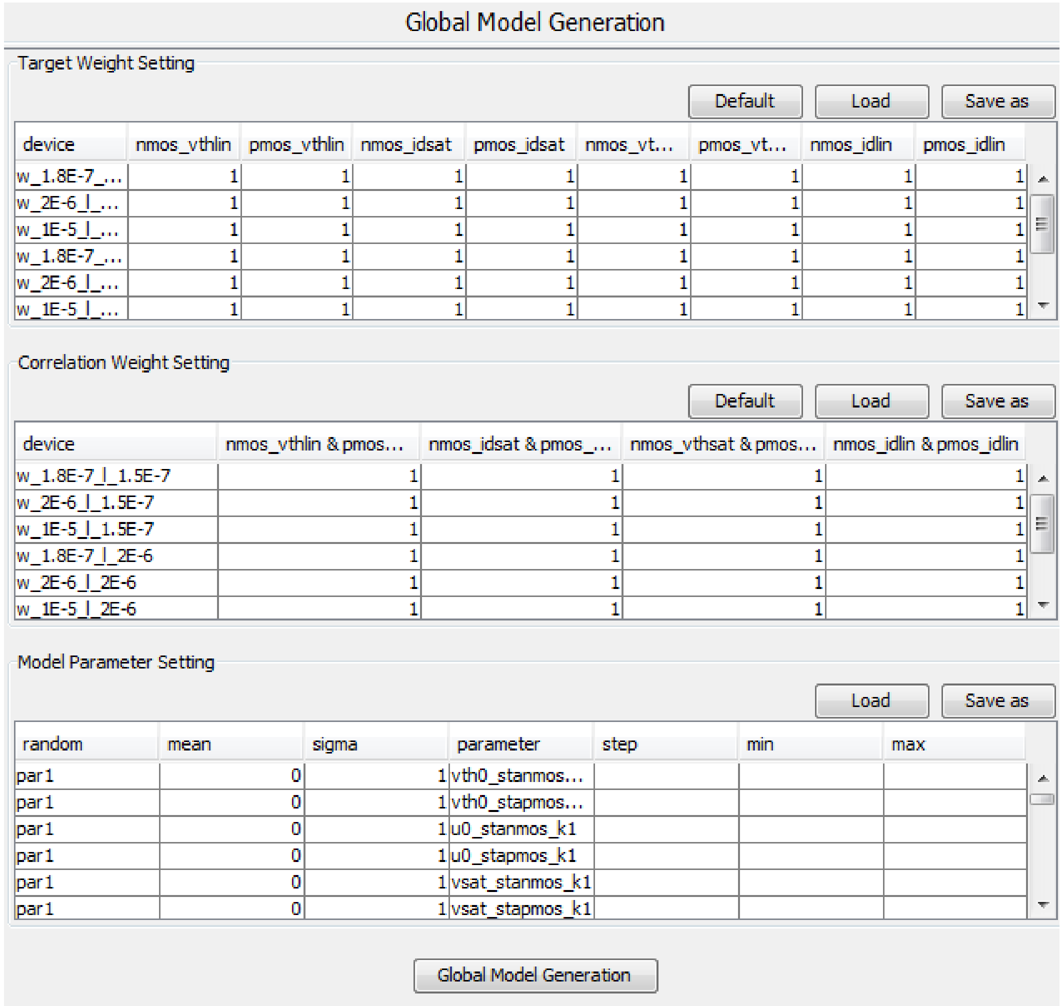
Click the "Generate Data" for data dispose.

1. a. Local Model



Set target weight table and parameter table, and click "Local Model Generation" button to extract local model

1. a. Global Model

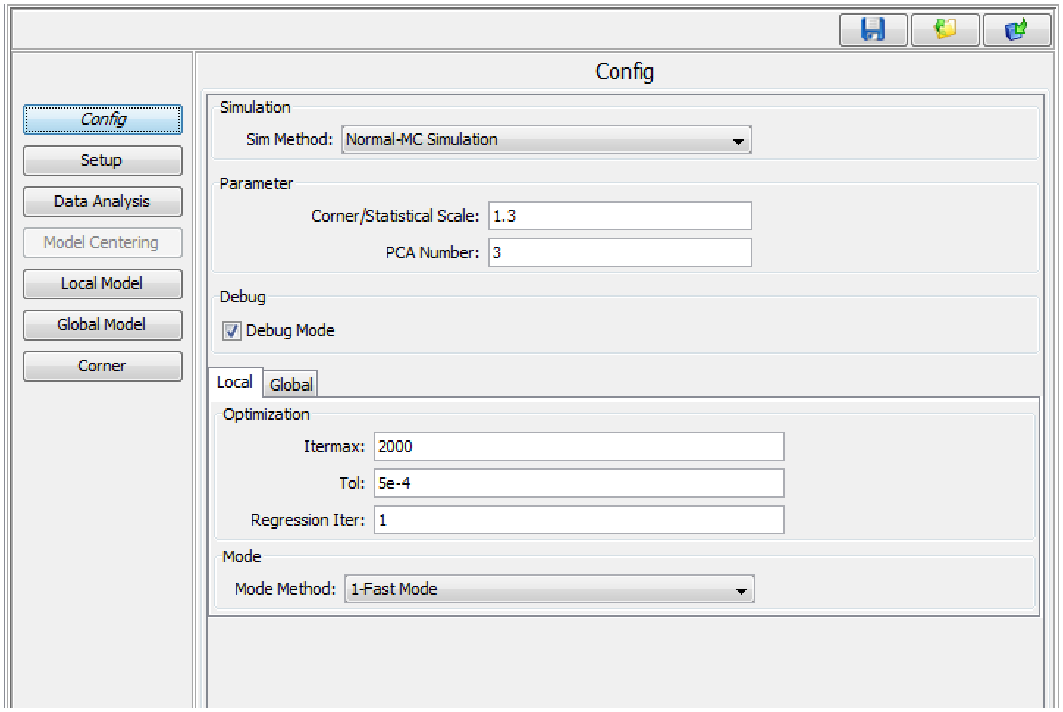


Set target weight table, correlation weight table and parameter table, and click "Global Model Generation" button to extract global model.
The generated dummy data and extracted model are saved in the "project_path\mosfet_demo_2\statistical";

Statistical model extraction from corner spec

Open the project from "MBP_home\demo\Statistical\GUIBasedSolution\mosfet\demo_case_cornerspec"

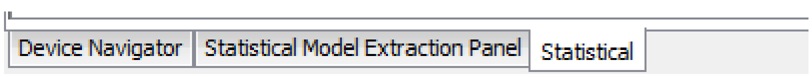
1. a. Config

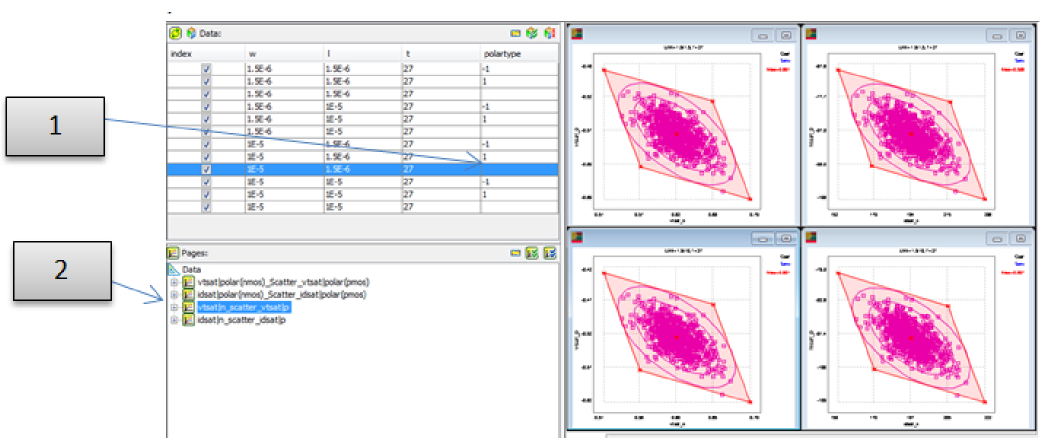


1. a. Setup

The demo corner data for this example can be found at "mbp_folder\demo\Statistical\GUIBasedSolution\mosfet\demo_case_fromcornerspe

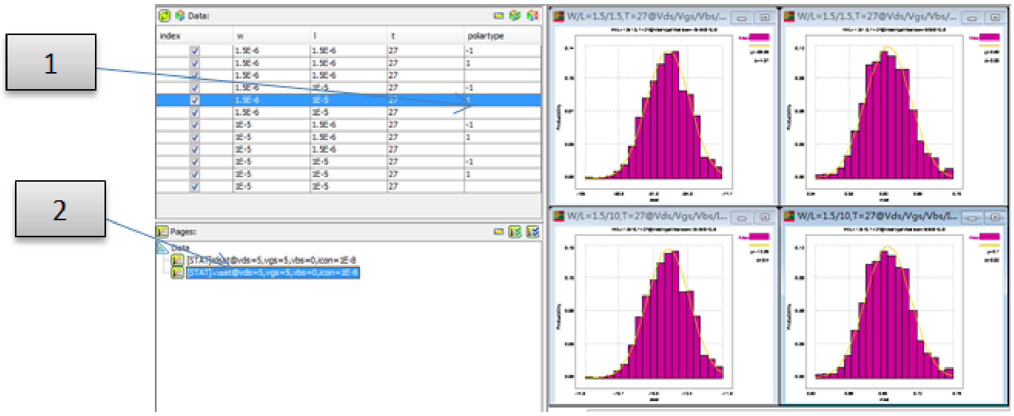
Click setup button, dummy statistical will be generated. And the new generated dummy data can be checked from the plots. Switch to the "Statistical" tab,





Scatter, contour and corner plot checking

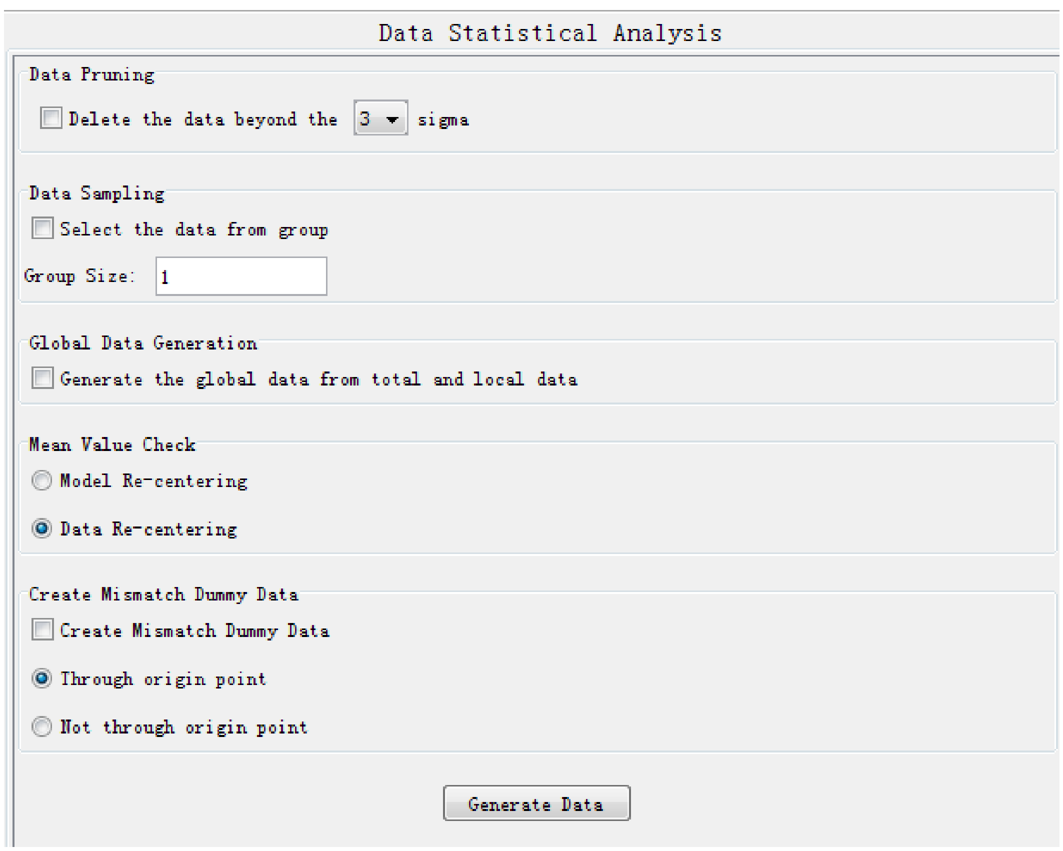
First, in the left table, select device whose polar type is blank. It means a pair of devices (a nmos and a pmos with same instance value) will been selected. Second, select plot from the pages' list. The scatter and corner plot can be plotted in the graph region.



Histogram plot checking

First, in the left table, select device whose polar type is 1 or 1 (1 for nmos and -1 for pmos). Second, select plot from the pages' list. The Histogram plot can be plotted in the graph region.

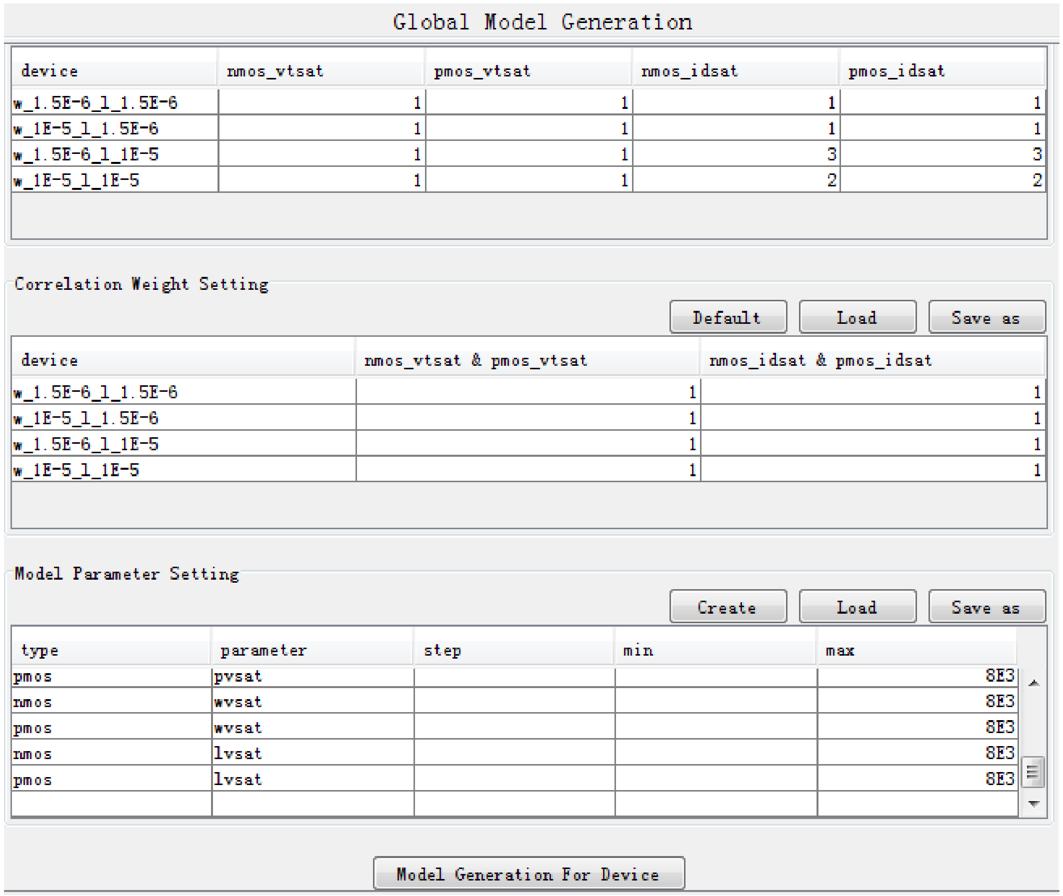
1. a. Data Dispose



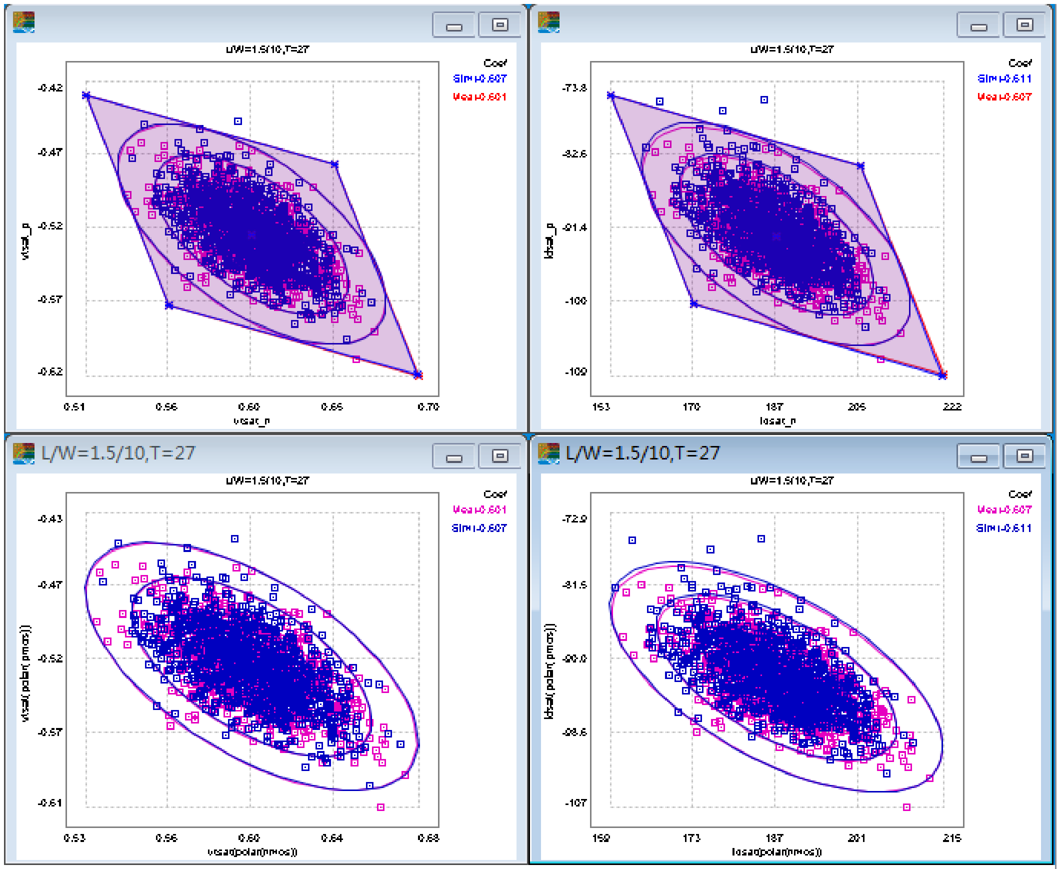
Click the "Generate Data" for data dispose.

1.

a. Global Model Extraction



Click "Model Generation for Device" button for global model extraction. And extracted model simulation result is plotted in the graph.

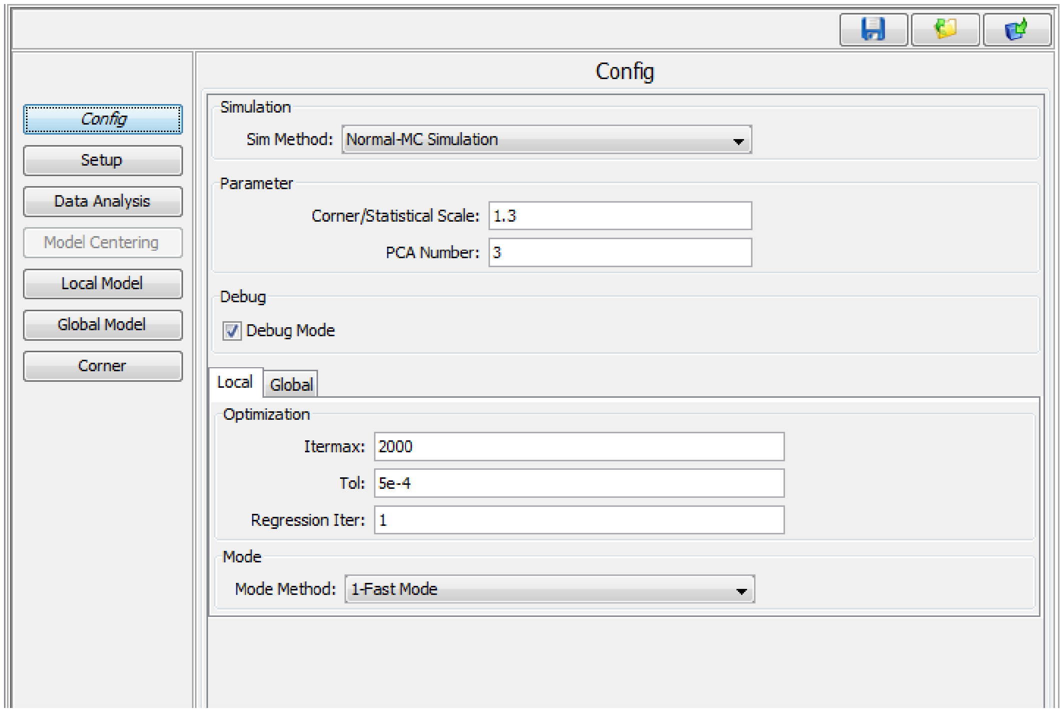


The generated dummy data and extracted model are saved in the "project_path\mosfet_demo_2\statistical";

Mismatch Model Extraction from spec

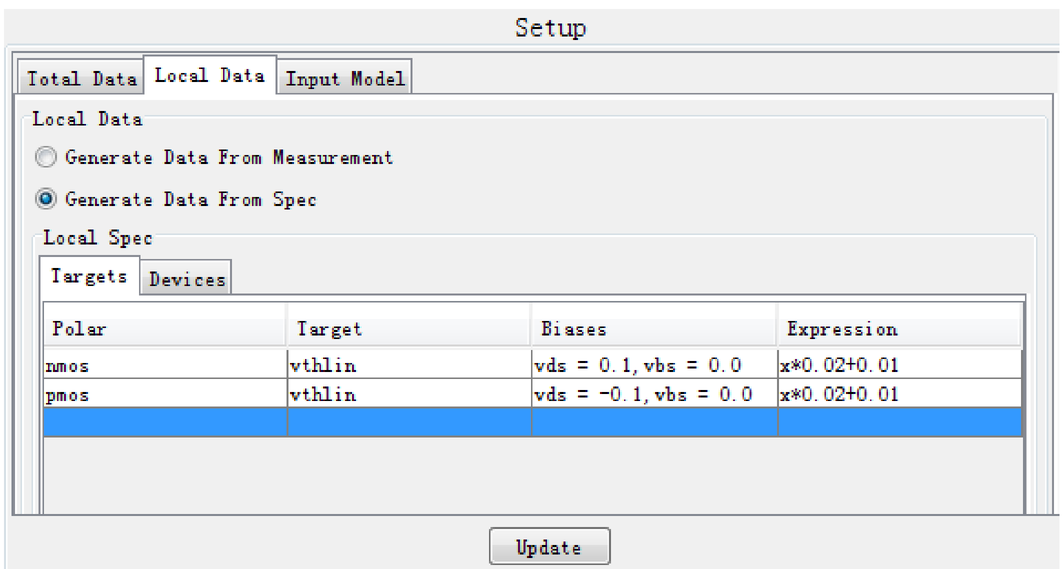
Open the project from "MBP_home\demo\Statistical\GUIBasedSolution\mosfet\demo_case_mismatchspec'

1. a. Config

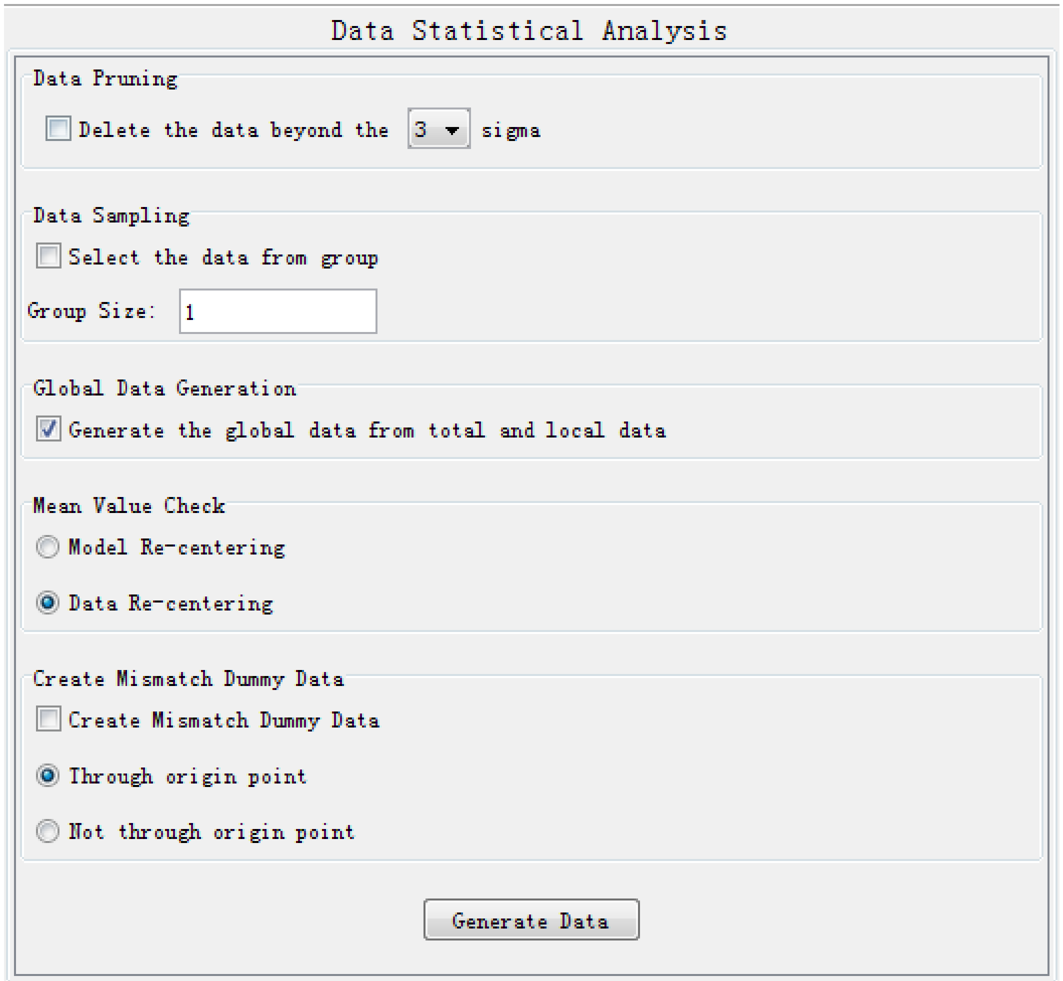


1. a. Setup

The measurement for this demo case can be found "project_path\MISDATA" and "project_path\STADATA"

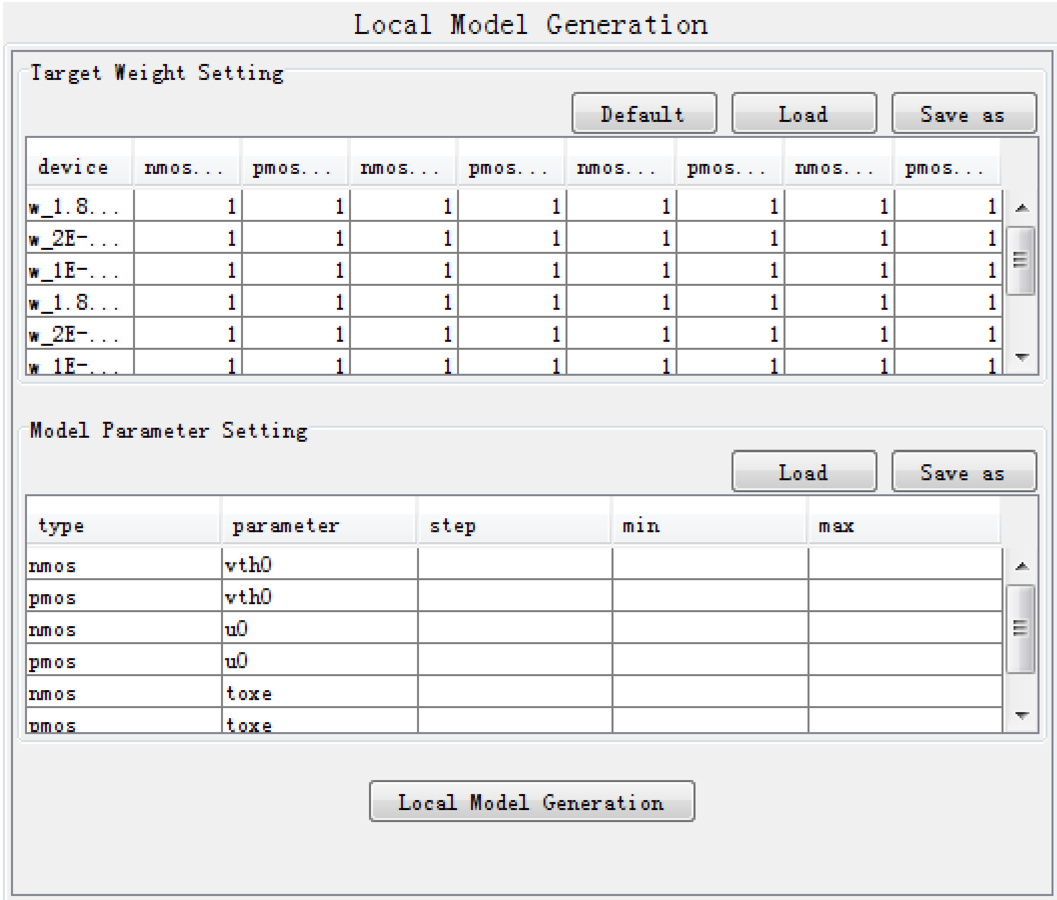


1. a. Data Dispose



Click the "Generate Data" for data dispose.

1. a. Local Model

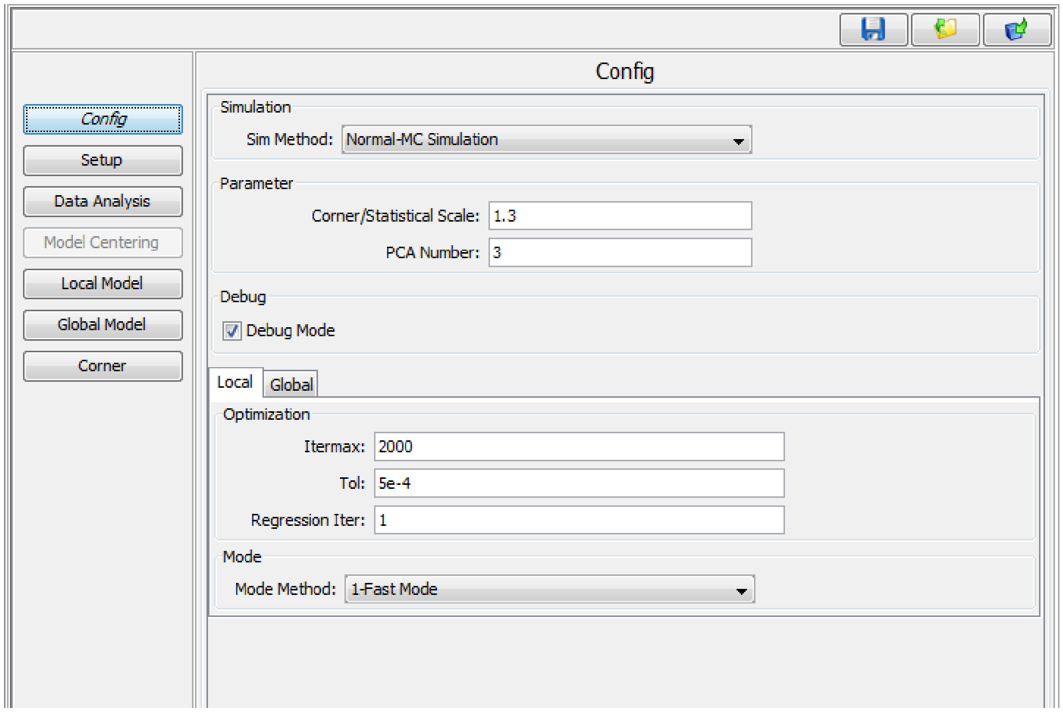


Set target weight table and parameter table, and click "Local Model Generation" button to extract local model

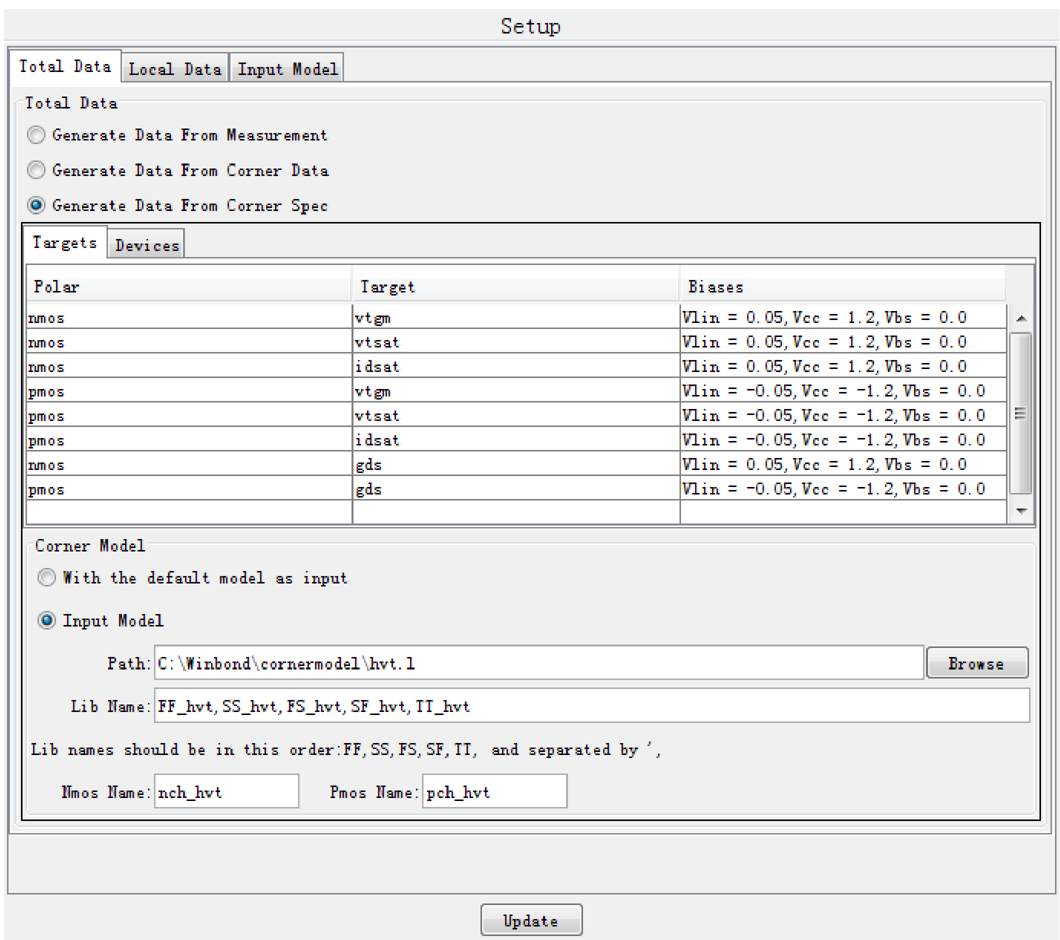
Statistical Model Extraction from Corner Model

Open the project from "MBP_home\demo\Statistical\GUIBasedSolution\mosfet\demo_case_cornermodel"

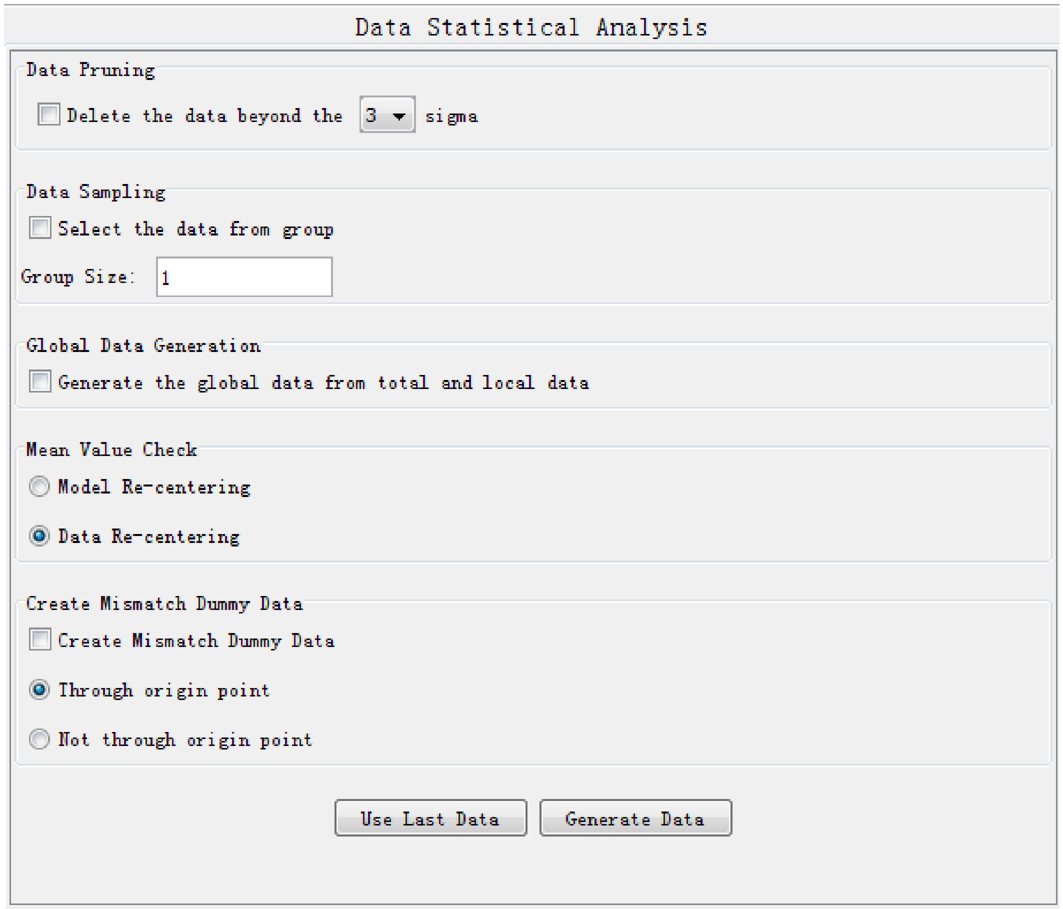
1. a. Configuration



1. a. Setup

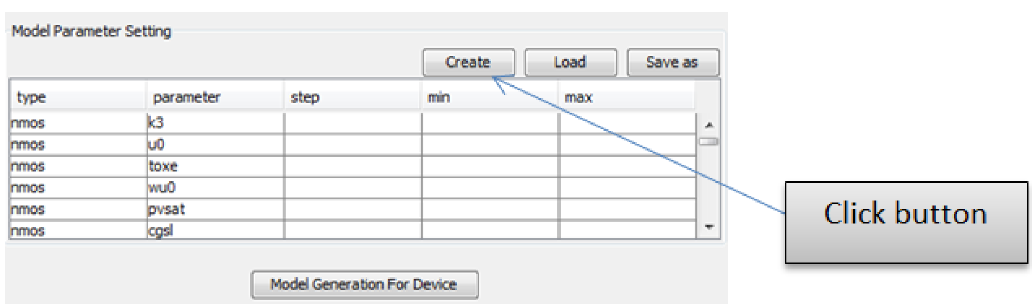


1. a. Data Dispose

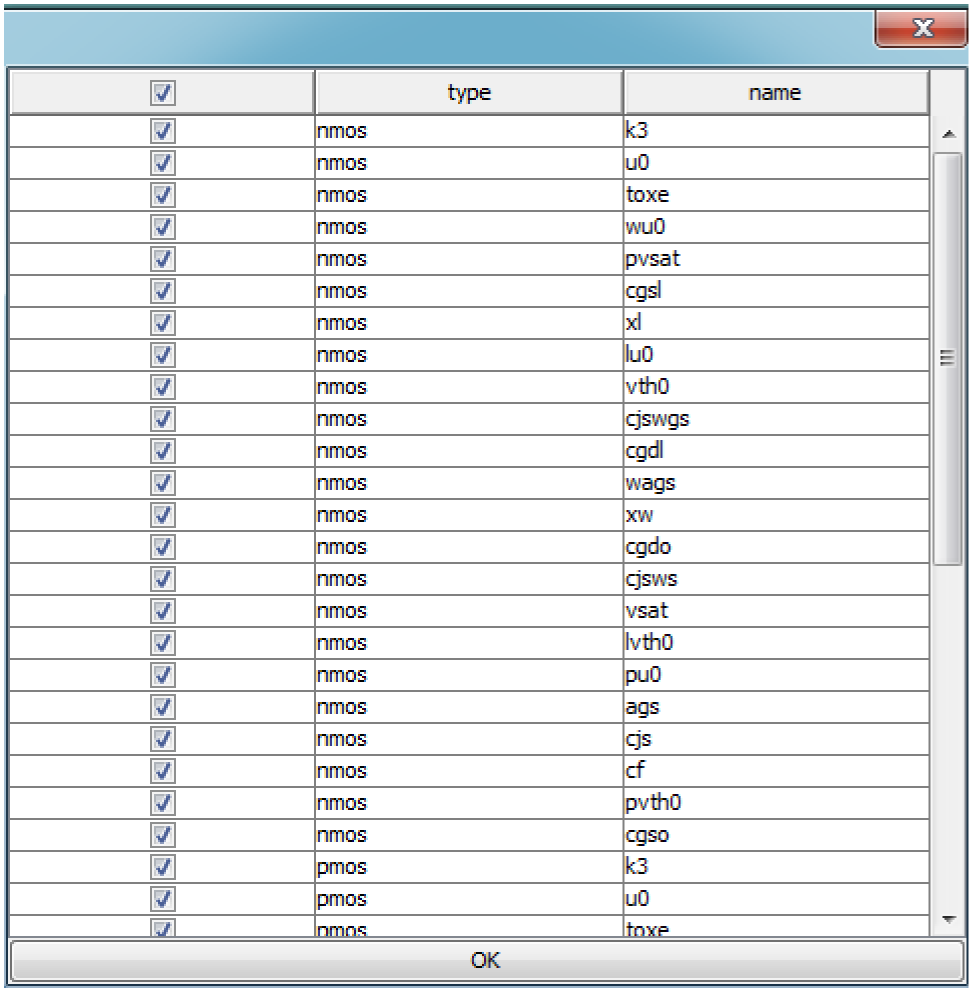


1. a. Global Model

The target weight table is similar as other examples. In the parameter table setting, please click the "create" button at first.

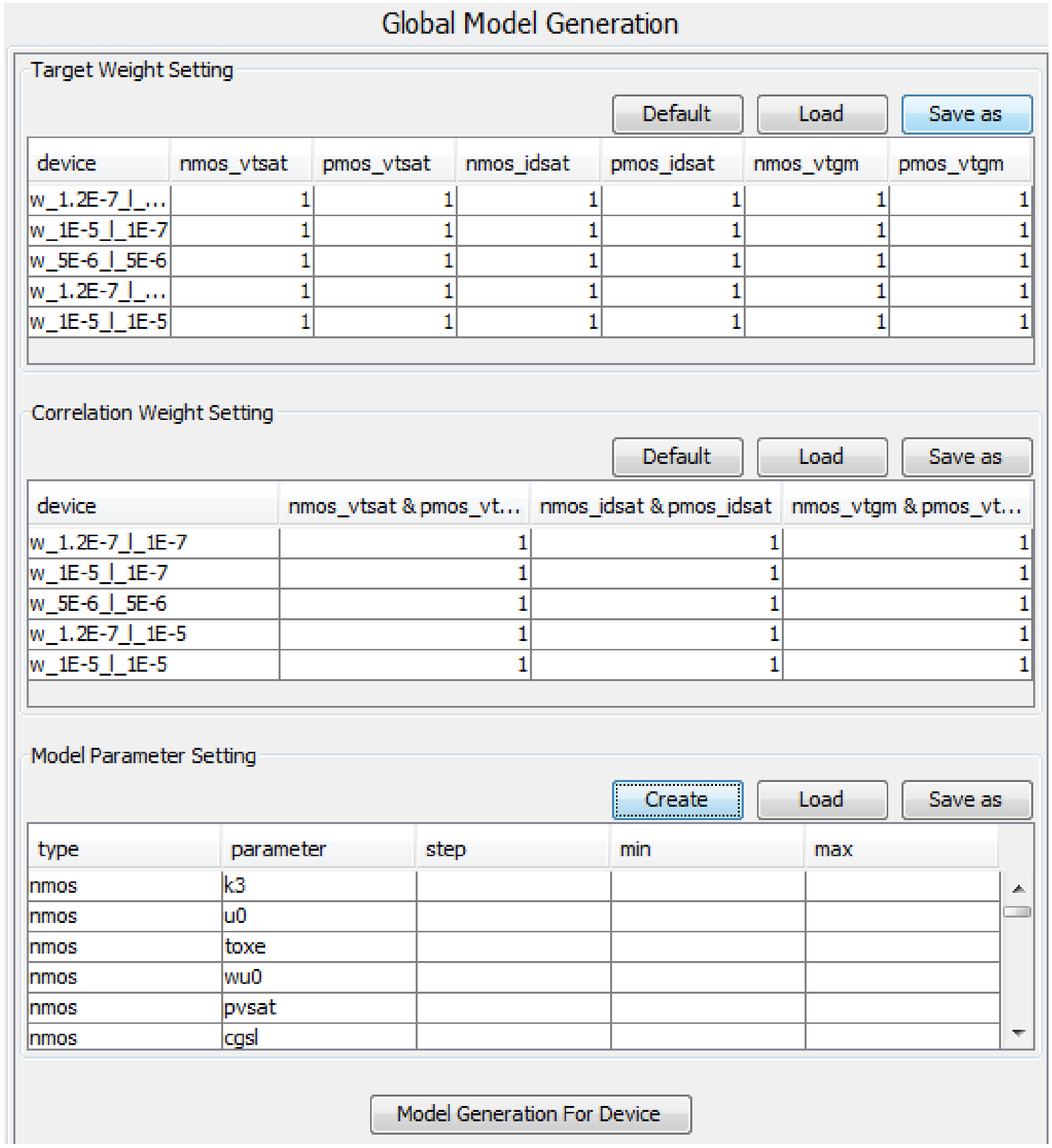


Then, select tuning parameters from the popped table.



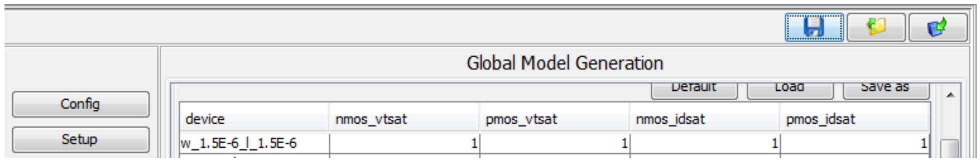
☑	type	name
☑	nmos	k3
☑	nmos	u0
☑	nmos	toxe
☑	nmos	wu0
☑	nmos	pvsat
☑	nmos	cgsl
☑	nmos	xl
☑	nmos	lu0
☑	nmos	vth0
☑	nmos	cjswgs
☑	nmos	cgdl
☑	nmos	wags
☑	nmos	xw
☑	nmos	cgdo
☑	nmos	cjsws
☑	nmos	vsat
☑	nmos	lvth0
☑	nmos	pu0
☑	nmos	ags
☑	nmos	cjs
☑	nmos	cf
☑	nmos	pvth0
☑	nmos	cgso
☑	pmos	k3
☑	pmos	u0
☑	pmos	toxe

Finally, click the "model generation" button to generate global model directly.



Q&A

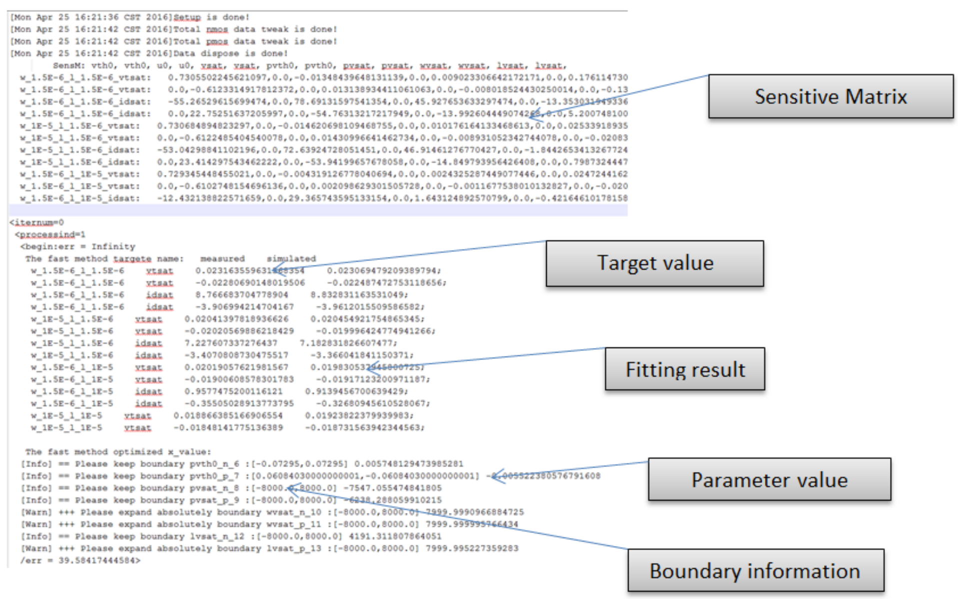
1. What's relation between mismatch model/data, statistical model/data, local model/data, and global model/data?
   In our document, mismatch model/data equals to local model/data. And statistical model/data equals to global model/data
2. How to set Monte Carlo Number?
   Select menu "Tools->GUI Options->Statistical Graph Config->MC count".
3. How to find extracted result?
   Go to current project folder, and all result and setting configuration file can be found in the "statistical" folder.
4. How to import/export configuration files?Save, export, import



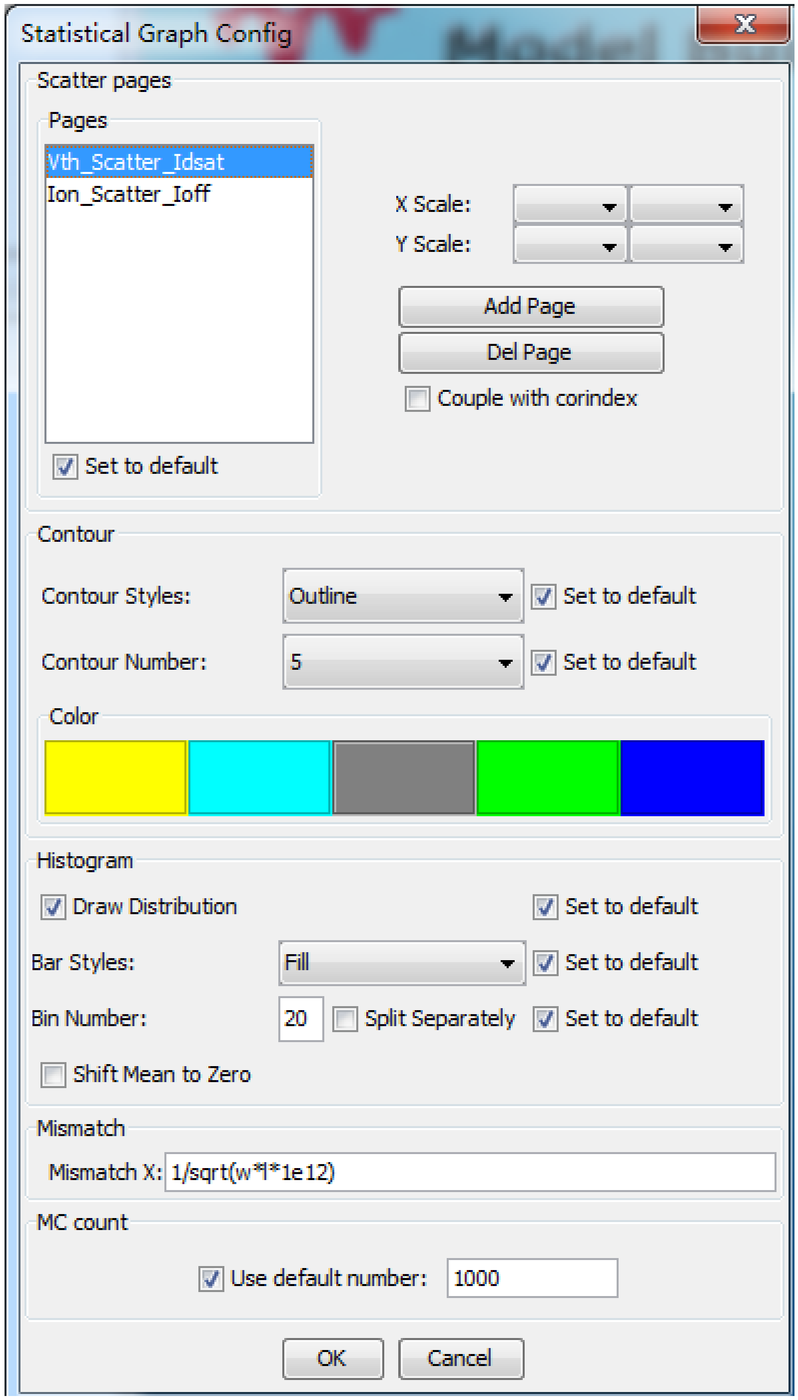
5. How to set contour line's number?
   Select menu "Tools->GUI Options->Statistical Graph Config->Contour Number".

6. Usually, a pair of nmos and pmos which has identical dimension value can be checked with scatter plot and the correlation coefficient is calculated. If a nmos and pmos which has different dimension value also need scatter plot and correlation coefficient, how should we do?
   We can use "corindex" to define a pair of devices. The devices with same corindex will be paired. If you want to check scatter plot for this pair device defined by "corindex", please select "Tools->GUI Options->Statistical Graph Config->couple with corindex".

7. How to understand the debug log file?



8. How to customize the mismatch plot's x-axis?
   From Menu "Tool->GUI Options->Statistical GUI Options"Mismatch X-axis



9. How to configure different type device?
   Script-> prog->defaultsetting
10. For mismatch spec, how to define x-axis?
    Default, we can define mismatch sigma by this way. Here, x=1/sqrt(w*l)

Targets | Devices

Polar	Target	Biases	Expression
r	res	vpn = 1.0	x*0.02

Moreover, we can use instance in the expression for specific applications.

Targets | Devices

Polar	Target	Biases	Expression
r	res	vpn = 1.0	w*0.02

Targets | Devices

polar	w	l	
r	1.0	1.0	
r	3.0	3.0	
r	5.0	5.0	
r	8.0	8.0	
r	10.0	10.0	
r	1.0	5.0	
r	3.0	5.0	
r	5.0	25.0	
r	8.0	40.0	
r	10.0	50.0	
r	1.0	10.0	
r	3.0	30.0	
r	5.0	50.0	
r	8.0	80.0	