MBP 2017

Using MBP

KEYSIGHT
TECHNOLOGIES

Notices

Restricted Rights Legend

Contents

- MBP Overview
- Parameters
- Simulation
- Optimization
- Data Display

MBP Overview

Model Builder Program (MBP) is a model extraction tool that provides a turnkey solution for parameter extraction and model generation. MBP has an open interface that allows you to customize modeling flows and internal functions using scripting. It's core environment includes all the necessary components for data loading, model simulation, parameter extraction, model generation and reporting. It is a complete solution that integrates SPICE simulation, parameter extraction, optimization, and model generation.

MBP supports all advanced models and mainstreamed simulators and enables the engineers to check models acquired from foundries directly.

MBP provides:

- Turnkey statistical modeling flow that enables data dispose, fast simulation, global statistical and local mismatch model generation in a dedicated module

- Equation viewer to view the details of complex equations and debug models as parameter(s) varies

- Model comparison function that enables in-progress model comparison during model extraction process

- Error monitor giving a global view of fitting error

- Model extraction packages for high-voltage, SOI, Multi-gate FinFET, Statistical and mismatch, and SPICE library construction

- Powerful tweaking function to re-target model library according to the design specifications

It includes Equation Viewer, High-Voltage Device Modeling package, Smart Model Retargeting module, and the automated extraction solutions for the latest CMC approved models such as PSP. In addition, it has many built-in features such as Error Monitor, Corner Model, and Lib Parser that improves the usability and ease the model extraction process.

- MBP Design Environment
- MBP Files
- Device Navigation

Device Navigation

This section describes the operation of MBP's device navigator. It includes the following topics:

- Device Navigator Interface
- Graphic Device Selection (G-select)
- Device Roller
- Operation of Device Navigator

Device Navigator Interface

The Device Navigator is located in the left side of main MBP window. It is designed to track all device information. once you load a valid measurement data file, the device navigator window pops up as shown in following figure.

Device Navigation Window
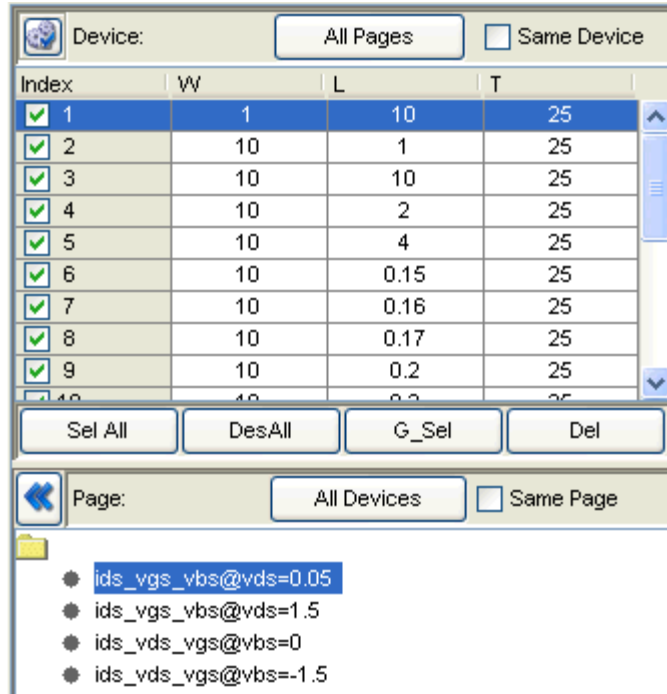
The table on the top side lists device information. An important property of device is device index. In the example above, the device of 10/0.15 has index of 6. You can select or deselect device by checking or unchecking the status box of each row.

Let us use BSIM3v3 model as an example, by default, MBP outputs W, L, and T in device navigator. Click Edit Instance icon to select/unselect the instance parameters from the table, as shown in following figure.

Edit instance

Instance					
Instance	Value	Instance	Value	Instance	Value
W (um)	100	L (um)	100	T (c)	25
NF ()	1	m ()	1	SA (m)	0
SB (m)	0	SD (m)	0	SCA (m)	0
SCB (m)	0	SCC (m)	0	SC (m)	0

To edit model instance and the corresponding device instance, click Extraction > Instance Setting from the main menu, the pop-up window is shown in following figure.

Instance setting

A sample device table with all parsed instance parameters in is shown in following figure.

Device Navigator with all instance parameters

The Page window is decked at the bottom of the device navigator window. It refers to certain device characteristics. In the measurement data file, there can be many devices and each device in turn can have several device characteristics. We call each characteristic a page. For example, in Bsim3v3 MOS measurement, typically there are four pages:

- Page 1: Ids vs. Vgs @ Vds = 0.1V (or 0.05V) with different Vbs.

- Page 2: Ids vs. Vgs @ Vds = Vdd with different Vbs. (Vdd is the maximum drain voltage).

- Page 3: Ids vs. Vds @ Vbs = 0V with different Vgs.

- Page 4: Ids vs. Vds @ Vbs = Vbb with different Vgs. (Vbb is the maximum body bias).

The All Pages button shows all the characteristics of the highlighted device. And the All Devices button shows the highlighted device characteristic of all the selected devices. Once the Same Device option is checked, only the pages of the highlighted device are displayed. As for which pages of the highlighted device gets displayed, it depends on whether you choose to show All pages or merely the highlighted device characteristic. This function is mainly to facilitate observation of all the characteristics of each device from one to another.
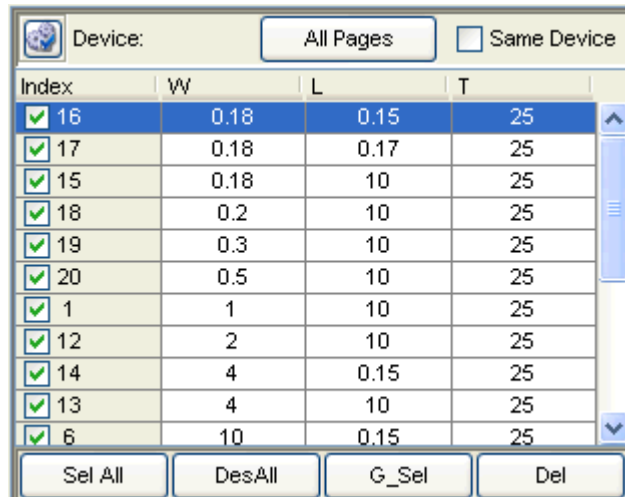
In the upper left corner of each plot, there is a plot index in the format of device index - page index. For example, as shown in following figure, the plot has an index of "4-3", it means that it is the third plot (Ids vs. Vds @ Vbs = 0V with different Vgs) of the device with index of 4 (W/L=10/2 T=25C).

Sample MBP plot
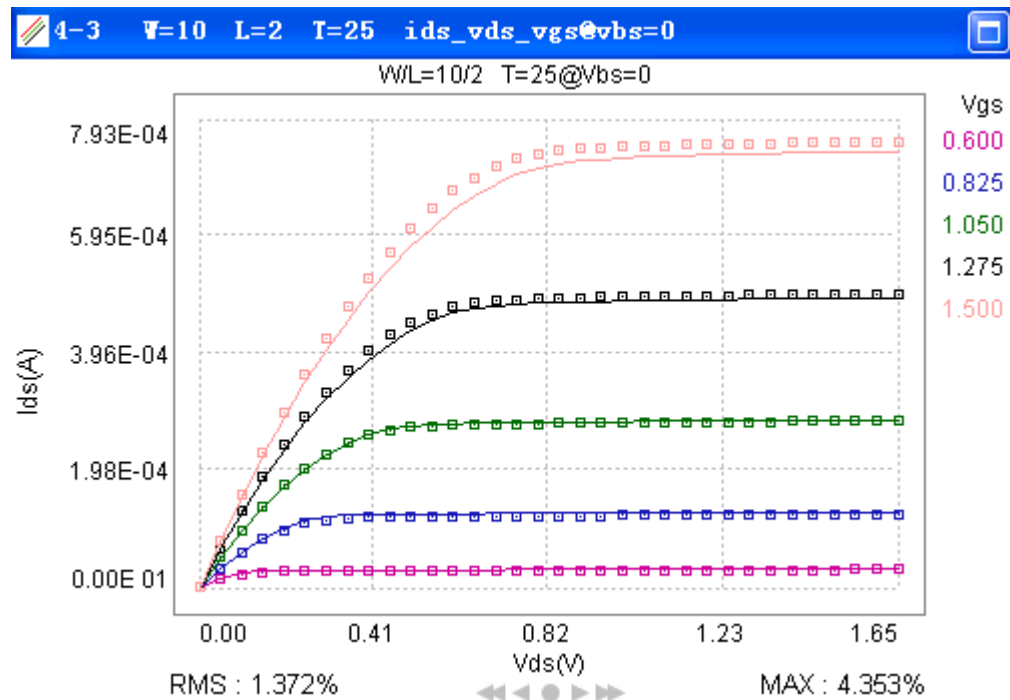
Another useful function is that in the table window, you can sort the devices according to the index, W, L, T or any other instance parameters by clicking the title row of the table. It gets sorted in either descending or ascending order. As illustrated in following figure, devices are in ascending order to W. It helps you to select devices with the same width, length, or temperature. You can also select devices in a graphic mode which is described in Graphic Device Selection (G-select) section.

Devices sorting according to W

By clicking icon ◀◀ near Page, you can minimize the Device Navigator to a bar. The operation remains the same, with acknowledgement of every button, as shown in following figure. To be noticed, in this case, device scroll plays a role of switching between devices. Correspondingly, page scroll plays a role of switching between pages.

Minimized Device Navigator
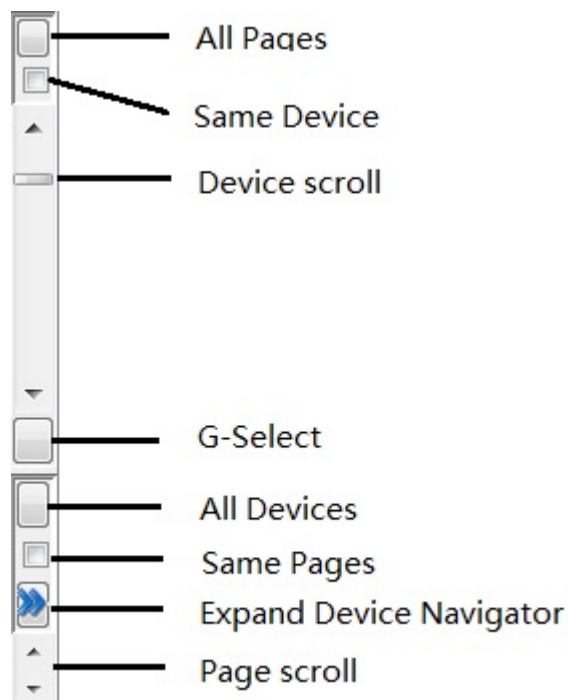
Graphic Device Selection (G_Select)

MBP offers to select devices in a graphic mode. Click G_Sel on device navigator, the graphic selection window pops up as shown in following figure. View the device selection on the graph. You can zoom in on the graph by drawing rectangle covering the selected region. To return the original scale, double click on the graph. You can also change the scale of both X and Y axis by double clicking the two axes. To select a device, hold Ctrl and use cursor (press and hold the left button) to draw a square to include the desired points. You can clear all the selections by clicking the Clear button. After the selection is done, press Apply or OK.

If loaded data are with two or more temperatures, switch between devices with different temperatures through T column at the left. And any operation is to the devices under highlighted temperature.

Graphic Selection Window
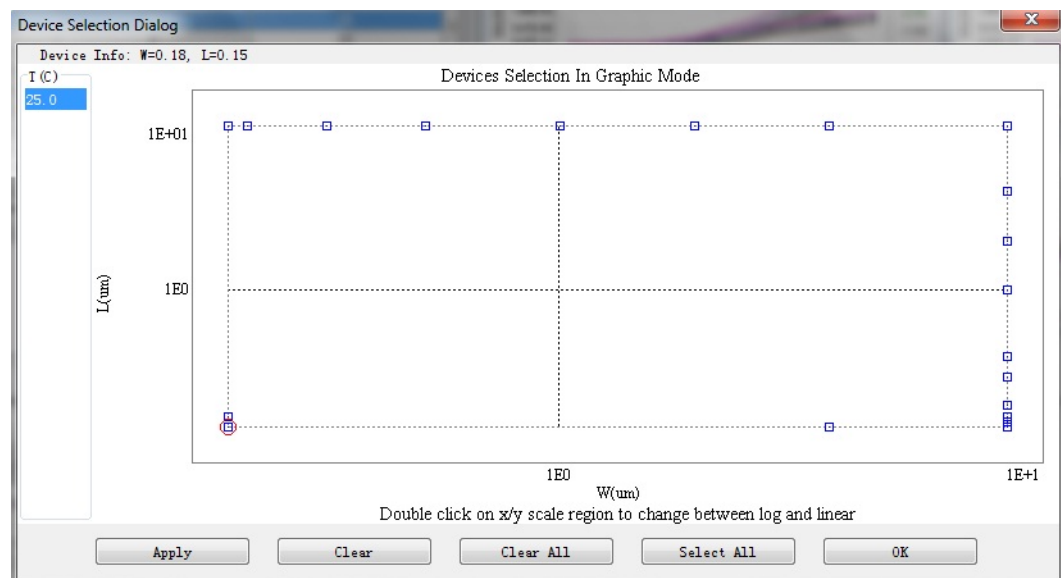
Device Roller

Device roller is a device selector which performs most of functions as the Device Navigator. Since it is at the bottom of each plot, you can browse different devices and pages without leaving the plot. The function that each button performs is explained as following:

- ◀◀ All Previous Device: All pages are replaced by the corresponding ones (with the same page index) of previous device.

- ◀ Previous Device: Current page (active window, see Active window section for details) is replaced by the page (with the same page index) of previous device.

- 🟢 Select Device: Current page is replaced by the page selected from the device list, as shown in Figure: Clicking Select device.

- ▶ Next Device: Current page (active window, see Active window section for details) is replaced by the page (with the same page index) of next device.

- ⏩ All Next Device: All pages are replaced by the corresponding ones (with the same page index) of next device.

> **NOTE** By device roller, you can only browse the devices that already selected in the Device Navigator.

The switching sequence through device roller follows the same sequence in the Device Navigator. For example, if the device is sorted by W (channel width), this order is also applied in device roller. If no device is available for the next or previous , when you point the cursor to the device roller, it turns red, like icon ⏪ .

> **NOTE** If math functions (like dx/dy) or scale transform (like x-lin, y-log) are applied to the current plot, such settings are kept when you switch between devices and pages by the device roller.

Clicking Select device
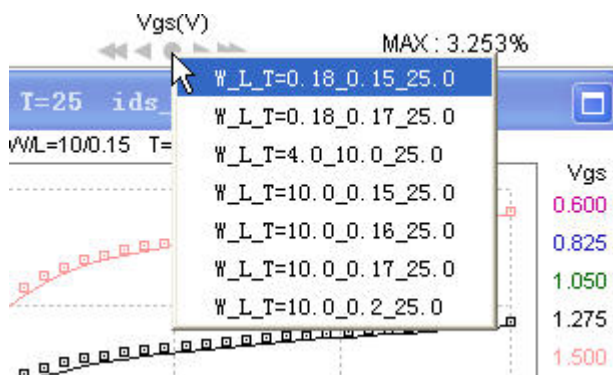
Operation of Device Navigator

In this section, we are introducing the definitions and settings regarding to the operation of Device Navigation.

Active Window

Clicking any of the plot window makes it active, which means it reflects the change made on the device navigator. If you want to display the certain page of certain device in a window, you should make the window active first (by clicking the plot), then use the device navigator to locate the desired device and page. In the sample window shown in following figure, the upper right window (i.e. index 6-2) is active with the title bar brightened.

Sample Active window
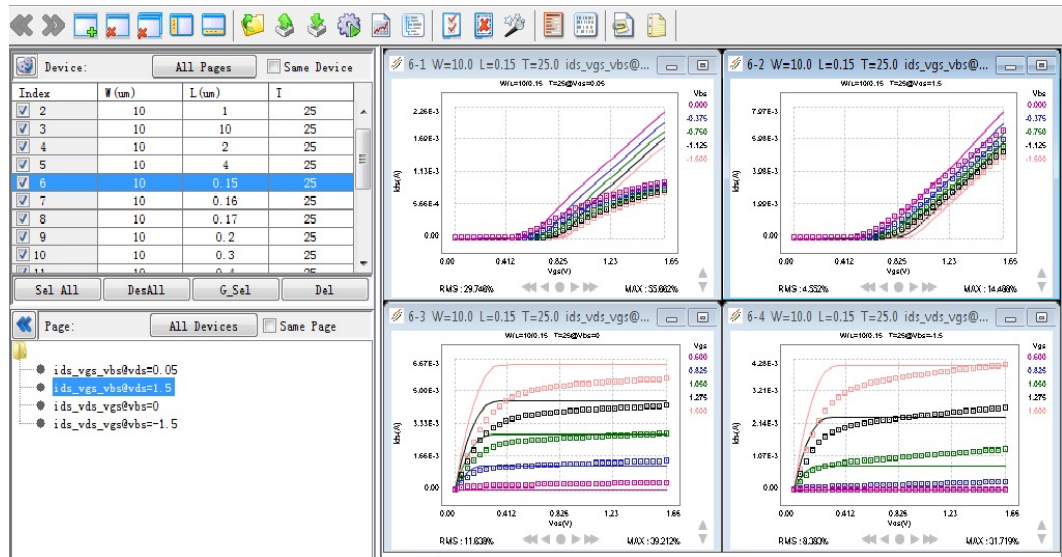
Next/Previous Screen

When multiple devices are selected, once clicked All Devices or All Pages button, MBP often could not display all the plots in one screen. Then icon Next Screen gets activated. You can use the Next Screen or Previous Screen buttons highlighted in following figure to switch between screens. For both All Devices and All Pages, MBP arranges plots in the order the same to that in Device Navigator. You can look through pages with desired order by sorting in Device Navigator.

Switching Between Devices
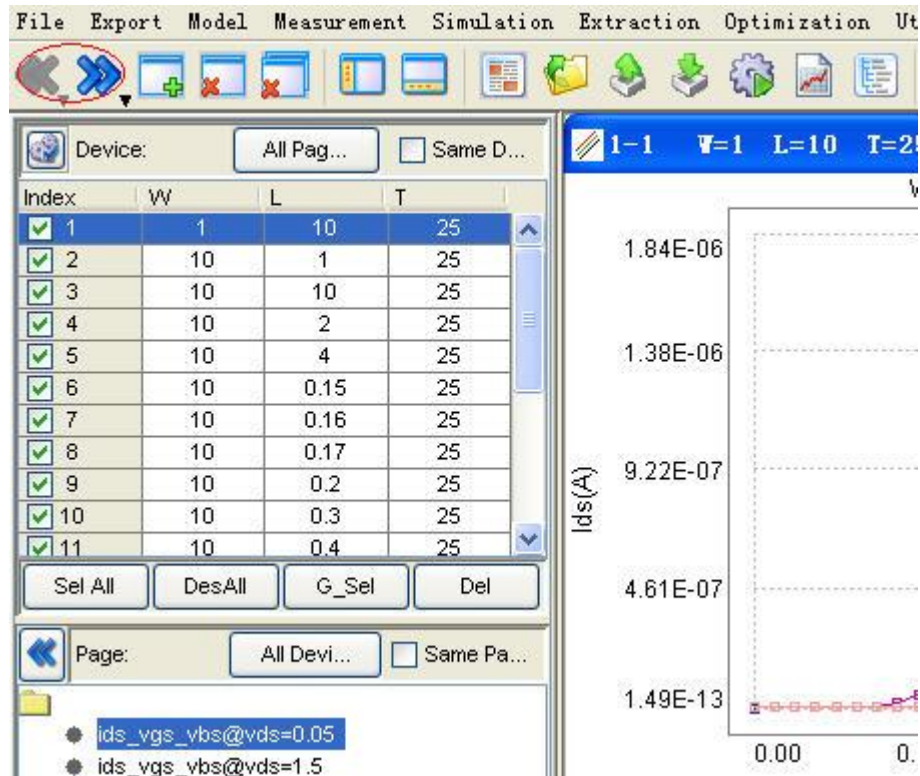
Layout Configuration

In GUI, select Tool > Layout to access layout configurations. By default, MBP displays four graph panels. If you want to display other styles of the plot window, select the configuration of the plot from the list as shown in following figure.

Graph Option Window
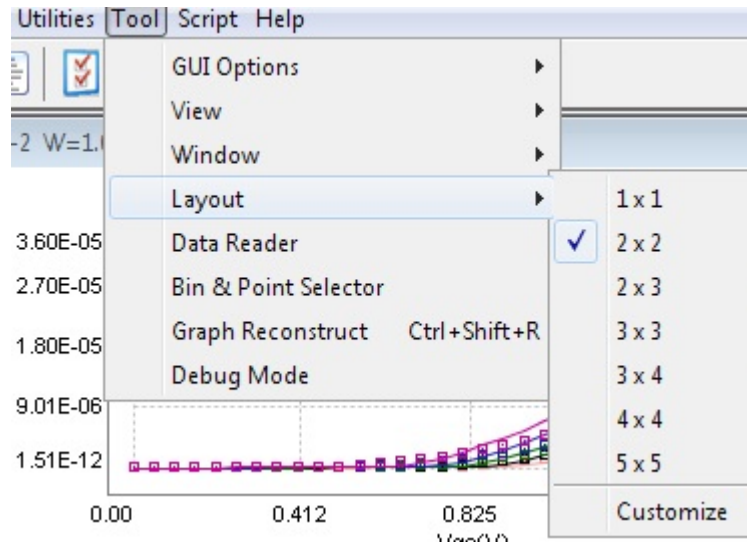
Or customize layout definition by clicking Customize. The layout menu setup window gets displayed and you can create new layout freely, shown in following figure.

Layout setup Window
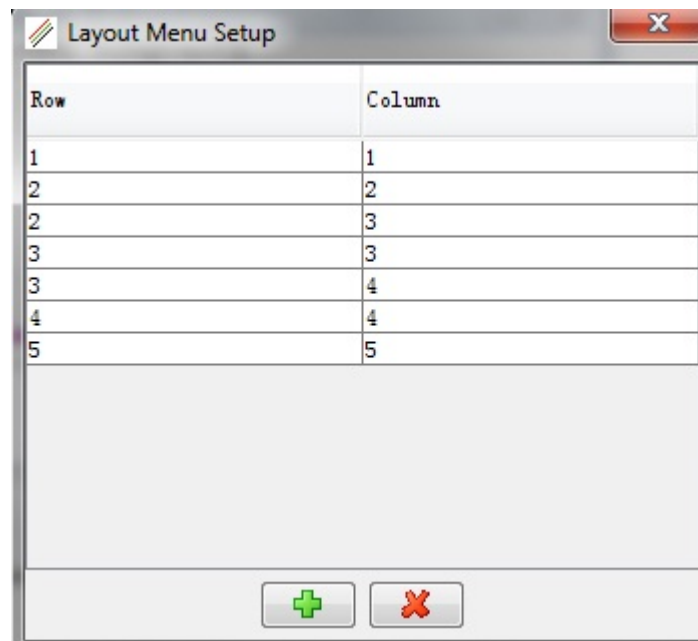
MBP Design Environment

This chapter describes the MBP GUI provided by MBP. It covers the following topics:

Main Window View

The Main window appears after MBP program starts, as shown in Figure: Main window view of MBP. It consists of six major parts:

Status Bar

Displays the path and name of the current project, compatible simulator and model information.

Main Menu

The central control part of MBP. It comprises eleven menus. Each of them has its own drop down sub-menu.

Tool Bar

Provides an easy way to carry out MBP's functions.

Control Pane

Device Navigator enables you to access and manage measurement data in data-based MBP. The data are listed in the format of tables in control pane. Other modules including Task Tree, IMV pages, and Lib Parser are also organized in tree structure in this pane. Switch among modules by clicking the corresponding tags.

NOTE	The tags besides **Device Navigator** are only available after corresponding modules are activated. For example, after you click **Task Tree** icon in the tool bar, you can see the **Task Tree** tag at the bottom of control pane.

Control pane can be docked by clicking the Hide Left Pane icon in the tool bar.

Plot Panel

All plots are displayed in the plot panel. If none of page is selected, the pane remains empty.

Bottom Pane

It contains two switchable windows:

- Message window: You can read the information such as the executed operation, error information, etc., from the Message window. Switch to this window by clicking the Message tab.

- Parameters window: All the parameters available are listed in the left and the selected parameters for tuning are listed in the right. For more information about parameter window, see Defining Parameters. Bottom pane can be docked by clicking the Hide Bottom Pane icon in the tool bar.

Main window view of MBP
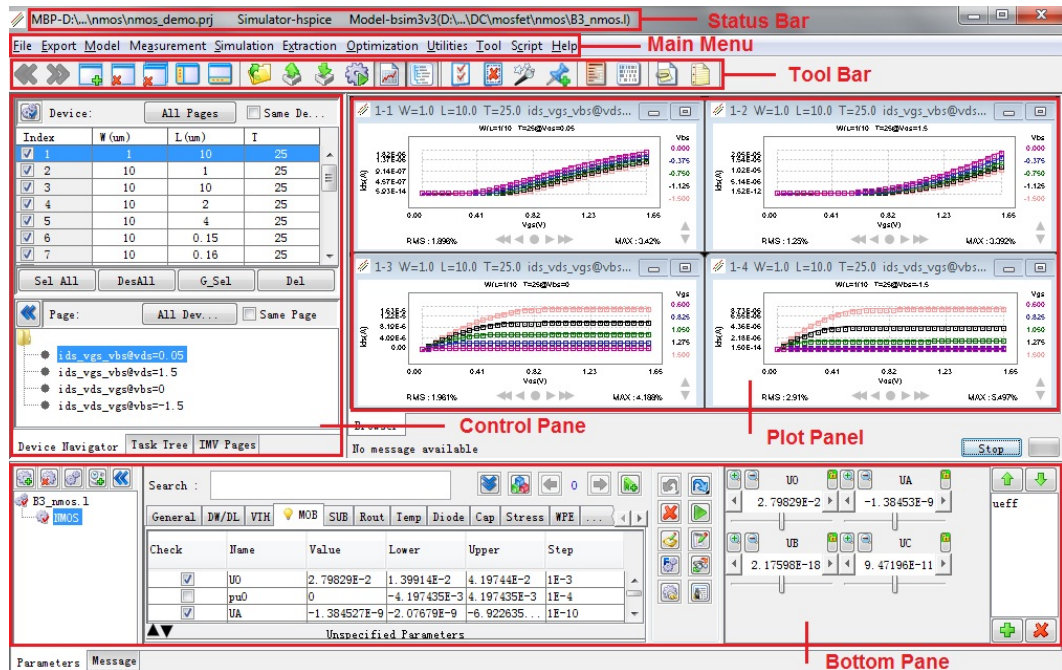
Plot Operations and Options

You can view the data using plot options like zoom in/out on the plot, set data range, change scale, etc. For more details on plotting operations and options, see Data Display section.

Other GUI Options

Layout Config

Using Layout tools, you can configure the way device pages are displayed. For more information on Layout tool, see Layout config section.

Data Reader

Data Reader can be used to determine the XY coordinates of a data point on a plot. For more information on data reader, see Data Reader section.

For details on other GUI tools, see Data Display section.

MBP Files

This chapter describes the file management in MBP. It includes the following topics:

- Overview
- Project Files
- Measurement Files

- Model Files
- Files under etc
- Files under usr
- Files under defaultproject
- File recentpath

Overview

Model extraction process involves many files including HSPICE/SPECTRE models, device characterization results, and system setting files. You can save, load, edit, or retrieve these files during modeling process. To give a clear expression of MBP files organization, a tree structure as well as the descriptions is shown in following figure.

Tree structure of MBP files

Tree structure	Description
$MBP_Home\	
project\, data\, models\	Save default project, data and model files
hspice\, spectre\, spice3\, ads\	Call **external simulator** settings
lic\, res\, help\	Save license file, logo and manual
dcp\	Measurement program module
demo\	Save **demo cases**
etc\	Default model, **parameter classification**
hspice\ (spectre\)	Instance parameter definition
...\model_core\	Files regarding **MOSRA**
...\inst_core\	Main declarations
mosra\, config.ini	Default **process parameter** value
analysis.xml	Default **plot style**
parameter.ini	Parameter **convertor** mapping
graph.ini	Model configuration
paramconvert.xml	RF graph setting
modelconf.xml	HV/RF template
rfgraph.ini	Math transfer functions
jet\usr\	Error monitor target
hvtemplate\, rftemplate\	Files regarding task tree
math\	Files regarding measurement
target\	Files regarding IMV pages
task\	Files regarding script
extras\	Recent path history
imv\	Main config file effect many settings on GUI
script\, jython.jar	RF project
recentpath.ini	Statistical project
mbuilder.ini	Script extraction flow by default configuration
defaultproject\	
rf\	
statistical\	
program\	

$MBP_Home stands for the directory where MBP has been installed.

NOTE

File recentpath

MBP shows your latest file (as shown in following figure) when you load data, model, or project. All the history records are saved in the file `recentpath.ini`. Clear this file after long time's usage since, there may be too many records here.

Quick Combo Box
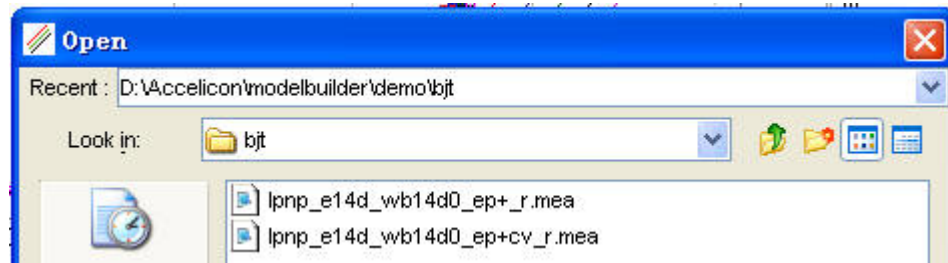
NOTE You can delete the unnecessary lines in this file to make the quick combo box clean.

Files under defaultproject

The configuration files regarding RF and statistical are stored under the folder \defaultproject.

\defaultproject\rf

There are several subdirectories under \defaultproject\rf which represent for different device types. All the model types supported by RF module are included in here. They can also be viewed by clicking Model > Select Model >RF from the main menu, as shown in following figure.

RF module supported model type

\defaultproject\statistical
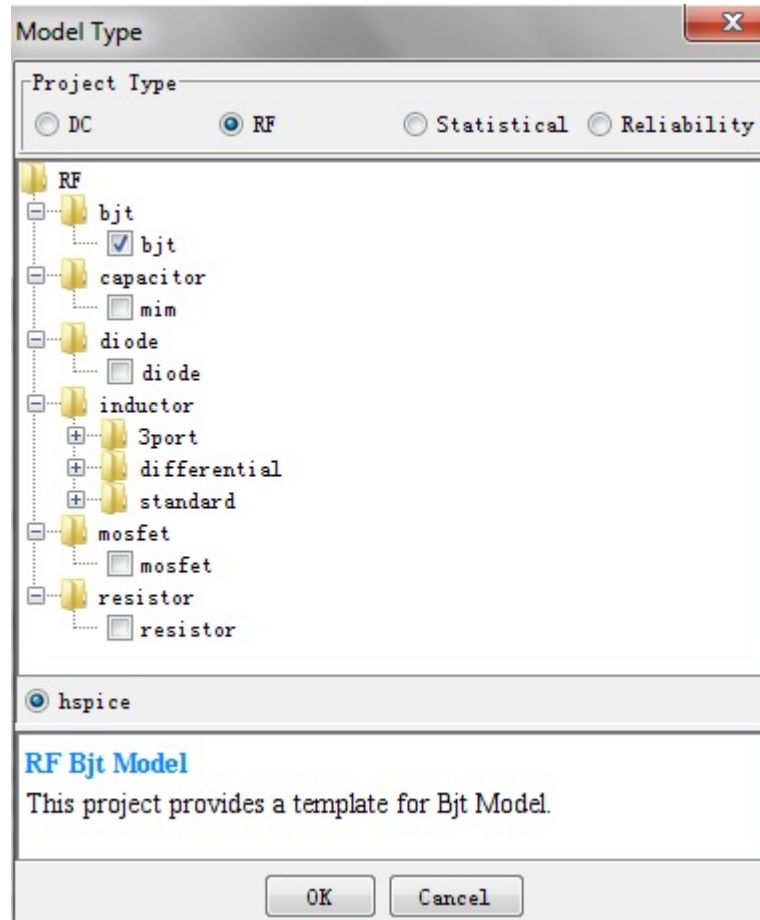
Similar with RF, there are also several subdirectories under
\defaultproject\statistical which represent for different device types. They can also
be viewed by clicking Model > Select Model > Statistical from the main menu, as
shown in following figure.
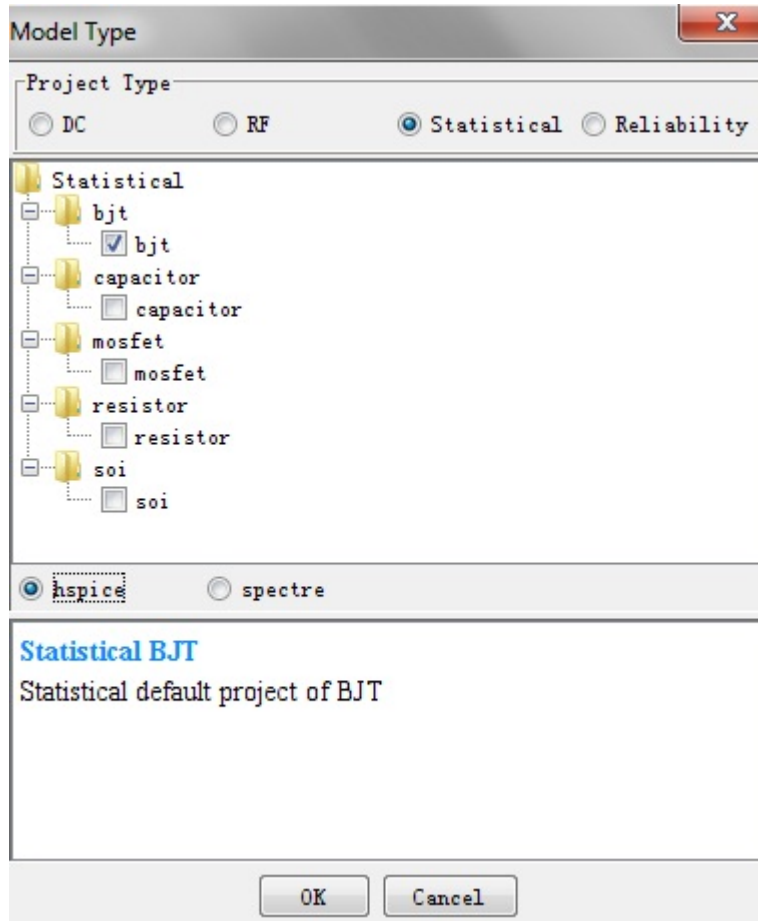
Files under etc

From this section, the customizable files, including the configuration files for many MBP features settings are introduced. There are many configuration files in its directory structure. These files define the display property, files operating paths, and so on. The modifications in these files customizes the tool as per your requirements. Files ending with *** in title can be modified in GUI. It is also recommended to modify the files in GUI directly. However, in some particular cases, it might be required to edit these files instead.

First we introduce the files under \etc directory. An example of directory structure is shown in following figure. The directory structure under \etc folder is complex. However, the hierarchy of this folder is straight-forward. Different simulator type is the first level while device type is the next. Under the model level folder, there are three folders named inst_core, model_core, and options, respectively.
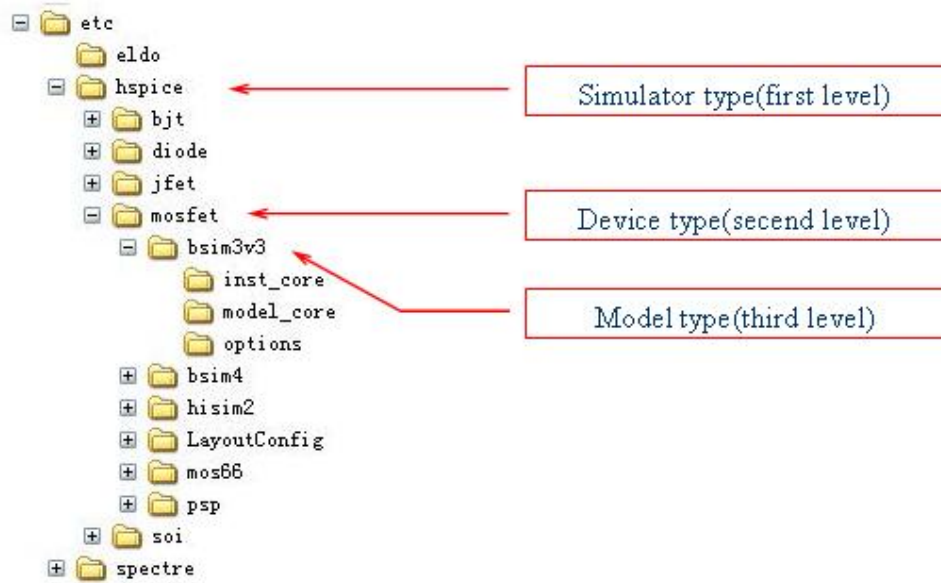
Files in \etc\hspice\mosfet\bsim3v3\model_core

- default_nmos.ini (or default_pmos.ini)

This file defines the default value of the parameters, soft/hard boundary, and the sweep step for the model corresponding. If model card is not used to define the parameter value, MBP takes the default value from this file. The default value is identical with the default value which BSIM3V3 defines. When a parameter is out of the default hard boundary, MBP prompts the warning message.

NOTE The hard boundary of the parameters could be only modified in this file.

The syntax of definitions is shown below:
Syntax:
Parameter_name = {para_type; defaultvalue; soft_boundary; hard_boundary; step}
Example:
lmin = {type=0; default=10E-6; soft=[1e-8,1e-4]; hard=(0,inf); step=1e-7}

- model_nmos.l (or model_pmos.l)
  When creating a new project in MBP, this model file is loaded by default.
  The parameters in this model file can be a subclass of those in default_nmos.ini (or default_pmos.ini) .

- paramclassify.conf

By configuring both parameter classification file (paramclassify.conf) and model settings file(default_nmos.ini), you can organize the parameters in the optimization window. The usage of parameter classification files is shown in following figure.

Usage of parameter classification file


```
#File: paramclassify.conf      #File: default_nmos.ini

[General]                      *** Flag Parameter ***
value = 1                      level      = {type = 3;   default = 49;
value = 2                      version    = {type = 3;   default = 3.3;
value = 3                      binunit    = {type = 3;   default = 1;
[DW/DL]                        update     = {type = 3;   default = 0;
value = 4                      binflag    = {type = 3;   default = 0;
[VTH]                          paramchk   = {type = 3;   default = 0;
value = 5                      mobmod     = {type = 3;   default = 1;
[MOB]                          capmod     = {type = 3;   default = 0;
value=6                        tempmod    = {type = 3;   default = 0;
[SUB]                          nqsmod     = {type = 3;   default = 0;
value=7                        acnqsmod   = {type = 3;   default = 0;
[Rout]                         noimod     = {type = 3;   default = 1;
value=8                        stimod     = {type = 3;   default = 0;
[Temp]                         wpemod     = {type = 3;   default = 0;
value=14                       igcmod     = {type = 3;   default = 0;
[Diode]                        igbmod     = {type = 3;   default = 0;
value=13
[Cap]                          *** Geometry Range Parameter ***
value=12                       lmin       = {type = 0;   default = 0;
[Stress]                       lmax       = {type = 0;   default = 1;
value=15                       wmin       = {type = 0;   default = 0;
[WPE]                          wmax       = {type = 0;   default = 1;
value=16
[Gate Current]                 *** Process Parameter ***
value=9                        tox        = {type = 2;   default = 1.5E-8;
[GIDL]                         toxm       = {type = 2;   default = tox;
value=17                       xj         = {type = 4;   default = 1.5E-7;
[other]                        gamma1     = {type = 4;   default = 1.2;
value=10                       gamma2     = {type = 4;   default = 0.2;
value=11                       nch        = {type = 4;   default = 1.7E17;
```

General	DW/DL	VTH	MOB	SUB	Rout	Temp	Diode	Cap	Stress

Check	Name	Value	Lower	Upper	Step
☐	level	49			0
☐	version	3.3			0
☐	binunit	2			0
☐	lmin	0	0	1	1E-7
☐	lmax	1	0.5	1.5	0.1
☐	mobmod	1			0
☐	wmin	0	0	1	1E-7

Unspecified Parameters

Besides the files introduced above, there are also two optional configuration files (param.ini and paramclass.ini) enables you to customize model parameters. To realize this function, create these two files in the corresponding \etc . The examples of these two files are shown below. You can define parameters name, boundary, group, etc there.

- param.ini

```
[Test]

*** P   ***
  p1          =  {type = 0;    default = 1;
soft = [ 0,              10 ];           hard = [ 0,
10 ];              step = 1                }
```

```
   p2          =  {type = 0;    default = 1;
 soft = [ 0,              10 ];              hard = [ 0,
 10 ];                step = 1                  }
   p3          =  {type = 0;    default = 1;
 soft = [ 0,              10 ];              hard = [ 0,
 10 ];                step = 1                  }
   p4          =  {type = 0;    default = 1;
 soft = [ 0,              10 ];              hard = [ 0,
 10 ];                step = 1                  }

 *** q ***
   q1          =  {type = 0;    default = 2;
 soft = [ 0,              10 ];              hard = [ 0,
 10 ];                step = 1                  }
   q2          =  {type = 0;    default = 2;
 soft = [ 0,              10 ];              hard = [ 0,
 10 ];                step = 1                  }
 *   *paramclass.ini*

 [demo1]
   value = 1
   [demo2]
 value = 2
```

So, after the following lines are added in the model file:
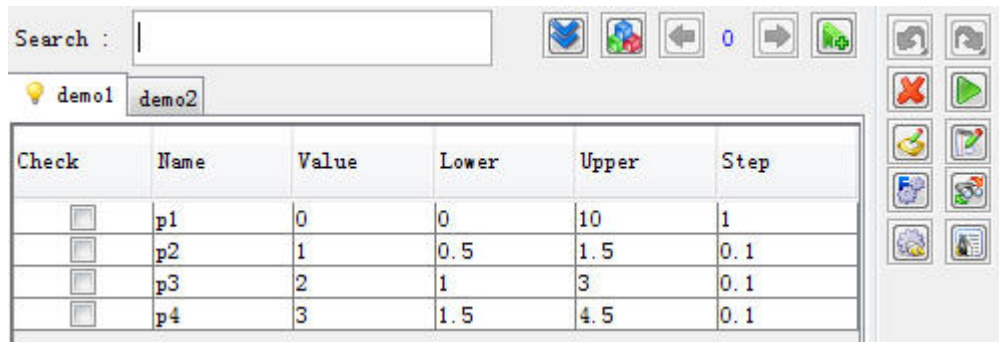
.param
+p1=0 p2=1 p3=2 p4=3
+q1=0 q2=0 q3=0

The model card window in MBP appears as shown in following figure.

User-defined parameters in group

Files in \etc\Hspice\MOSFET\BSIM3v3\inst_core

- inst.ini

This file claims the model name and instance parameter of the certain device. MBP creates netlist according to this file. Modify this file to add the instance parameters.

Syntax:
Model_name, ports name, device type, instanceparameter1=<default_value, parameter_type, unit_scale, offset_value, unit_name>;
Example:
m1 d g s b nmos w=<1e-6,W,1e-6,0,um> l=<1e-6,L,1e-6,0,um> temp=<25,T,1,0,>;

In some cases, you can use instance parameters with different names. For example, you can define subcircuit model as:
mcore d g s b width=1e-6 length=1e-6

To make it compatible to MBP, change model definition to fit its format:
mcore d g s b w=1e-6 l=1e-6

or add the definition to inst.ini as:
...w=<1e-6,width,1e-6,0,um > l=<1e-6,length,1e-6,0,um>...

- instdisplay.ini

This file records the instance parameters that displayed on the device navigator, which is the top left corner. Click the button Edit Instance 🖼 to add or delete the instance parameters displayed. The Instance Parameter window is shown in following figure.
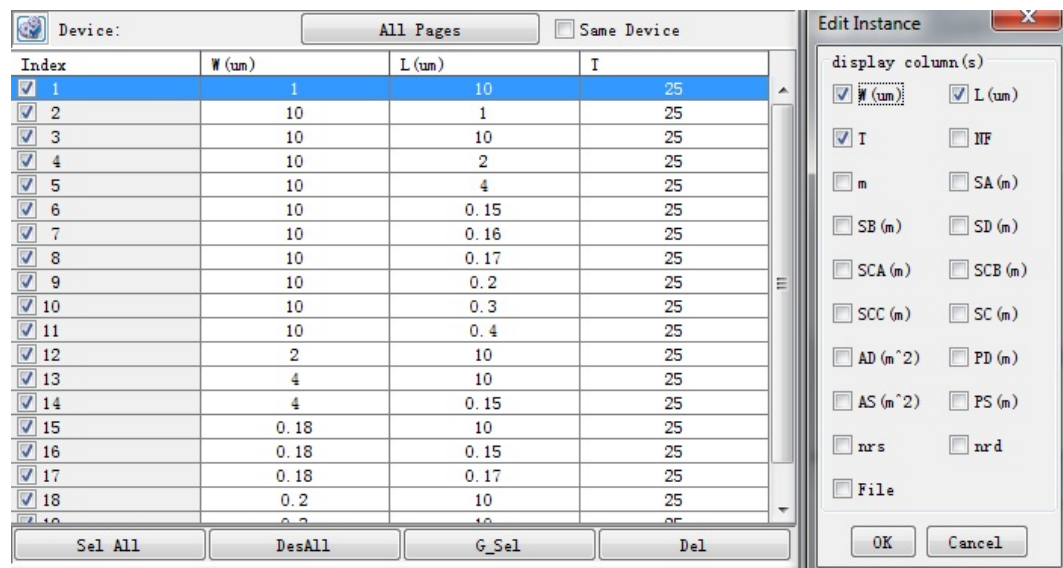
NOTE If option **File** in the Edit Instance window is checked, the file name of the measurement data is displayed in a new column. Hence, you can easily separate the data from different files by name.

- meainst.ini

NOTE The file meainst.ini is only applied to the old measurement module which is stopped being developed now. Since, DCP has replaced the old measurement module, refer to DCP user manual ($MBP_HOME\dcp\doc) for more powerful and flexible feathers.

Different from the file inst.ini, file meainst.ini defines the instance parameters that is recorded during the measurement. The basic syntax is just the same as inst.ini. The Measurement Instance Parameters window is shown in following figure.


\etc\hspice\mosra
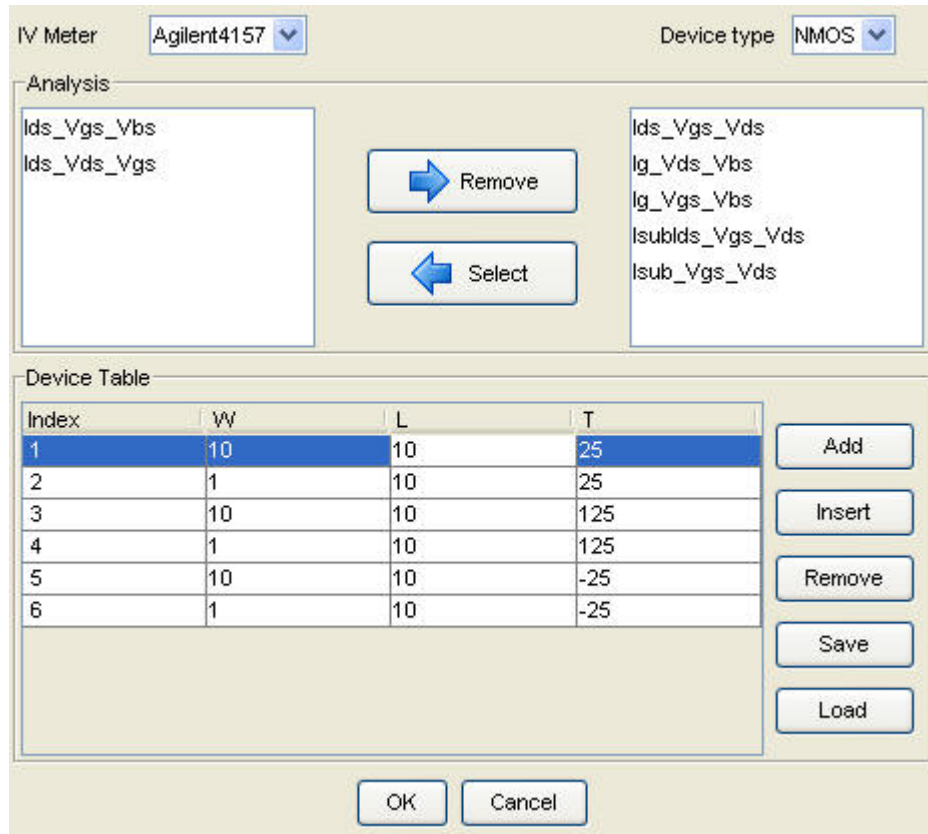
The folder includes the setting files regarding MOSRA. The file \etc\hspice\config.ini lists the device type which MOSRA supported: mosfet and soi.

\etc\analysis.xml

The file analysis.xml is critical to the simulations and outputs. After loading the measurement data, MBP establishes the netlist according to its counterpart in analysis.xml and passes the netlist to internal or external engine. In summary, it works as a bridge between the measurement data and the outputs of MBP.

The analysis.xml is divided into several sections. Each section corresponds to a certain device type. Let's use the bjt section in the file analysis.xml for instance, as shown figure below. First we get variable declarations. After name is declared, decide its source type. The "i" stands for current source and "v" stands for voltage source. And the sweep mode should be either 'dc' or 'cv'. The tonode or fromnode determines how you connect this source, or say that it determines the polarity. Then all these variables are allowed for reference in the page declaration.

Analysis
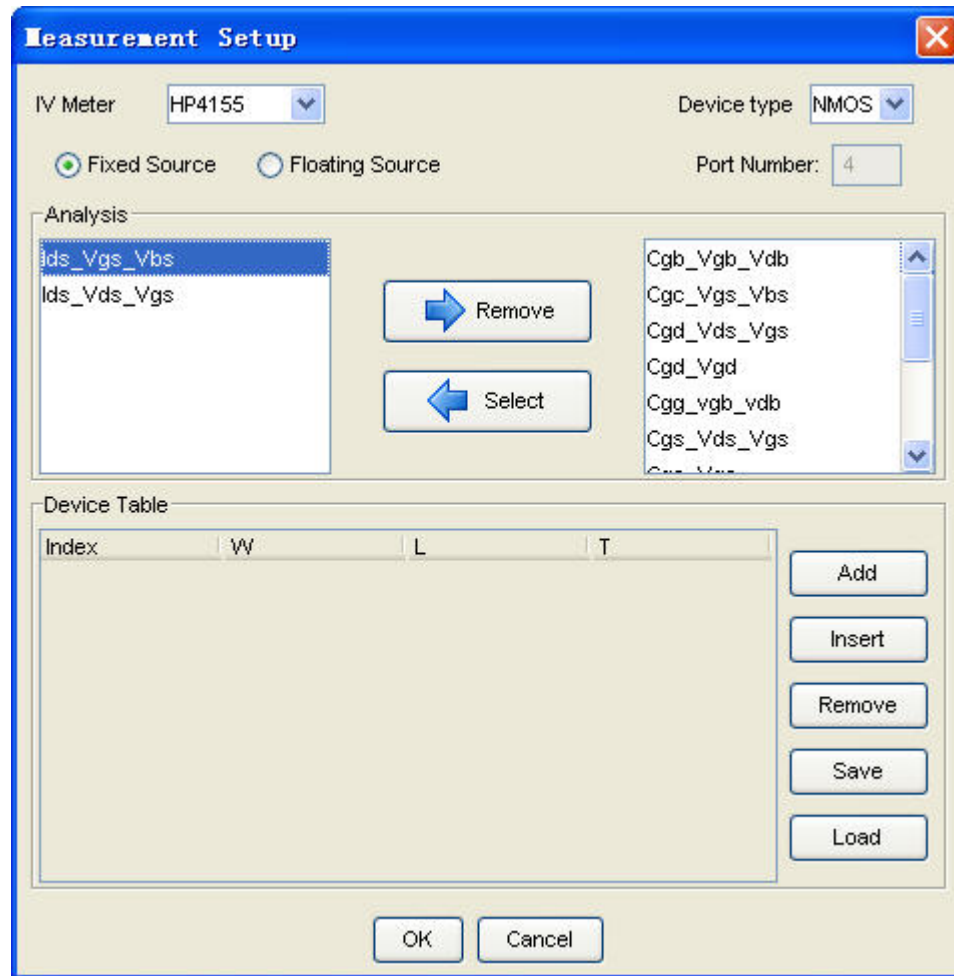
In the page declaration, ensure that circuit name should match the page name properly, as in the following. The ground represents reference node for this circuit. Here, "ib" and "ic" are the outputs. In terms of the element items, describe the connection of the sources to the available nodes.

```
<circuit name=Ic_Vbe type=dc ground=e>
<output name=ib unit=a/>
<output name=ic unit=a/>
<element name=vbe type=v/>
<nodes=b,e def_value=0/>
<element name=vbc type=v/>
<nodes=b,c def_value=0/>
<element name=vse type=v/>
<nodes=s,e def_value=0/>
<circuit/>
```

The corresponding output is illustrated below in following figure.

Analysisoutput
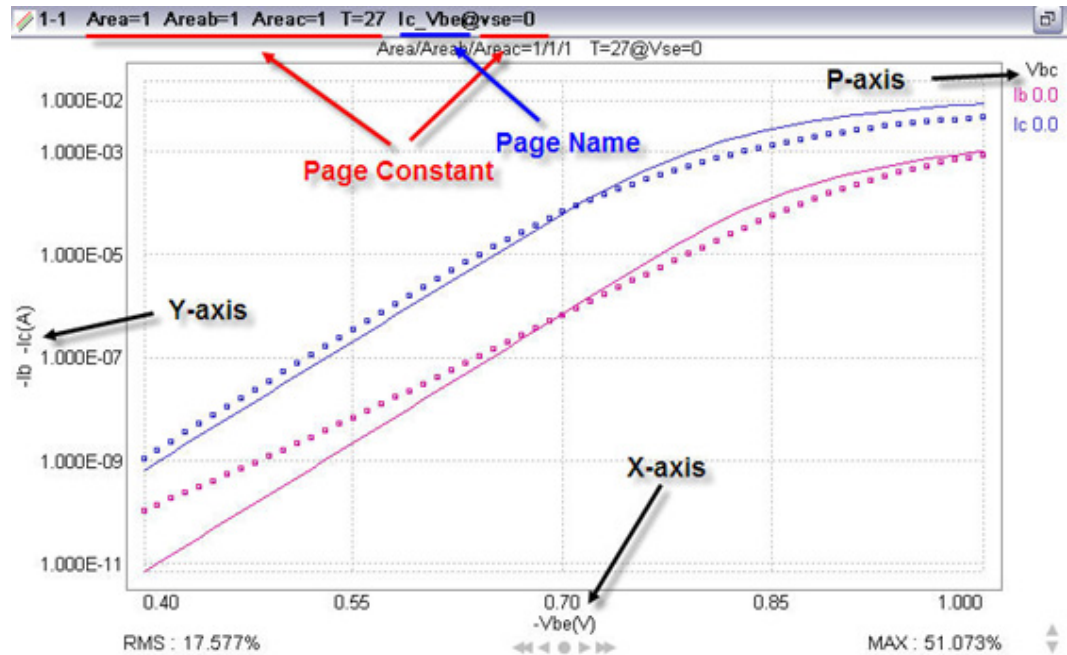
\etc\parameter.ini

Default value for process parameters is saved in this file. For each section, the syntax is as shown below:
entry_name = entry_value

Here, the entry_name is model parameter name. And the entry_value is surrounded by <> and comma-separated to five parts.

Take Tox for example,
Tox=<70.0,Tox,1e-10,0,A>

In the order from left to right, means: 70.0(default value), Tox(GUI display string),1e-10(scale),0(offset value),A(unit symbol)

You can modify the parameter value by clicking Model-> Set Model Options, shown in Figure 2.21. The detailed items in the window are different for different models types such as HSPICE_Bsim3, HSPICE_GP etc.

Model_option

> **NOTE** If the value of process parameter defined in model card is different from the default, MBP chooses the value from the model card. In other word, the parameter value defined in model card has a higher priority than the default.

\etc\graph.ini

This file defines the default plot style. There are four sections here: The first section is for plot type definition; the second section defines the scale of x-axis; the third part saves the y-axis scale; and the last one records the math-transform information. Detail information is shown in the following:

Syntax:

PageName=GraphStyle, XLogScale, YLogScale, MathTransform

1. graph style: 0 for CARTESIAN.(Default=0)
   a. for Smith Chart
   b. for Polar Chart
   c. for STAC_HISTOGRAM
   d. for STAC_SCATTERPLOT
   e. for MISMATCH

1. X-axis log scaled: true/false (Default=false)
2. Y-axis log scaled: true/false (Default=false)
3. math transform code: 0 for Y_AXIS_VALUE_RAW (y)(Default=0)
   a. for Y_AXIS_VALUE_RECIPROCAL (1/y)
   b. for Y_AXIS_VALUE_FIRST_ORDER_DERIVATIVE (dy/dx)
   c. for Y_AXIS_VALUE_SECOND_ORDER_DERIVATIVE (y*)
   d. for Y_AXIS_VALUE_THIRD_ORDER_DERIVATIVE (y*')
   e. for Y_AXIS_VALUE_FIRST_ORDER_DERIVATIVE_ PECIPROCAL (dx/dy)

For example:

[BJT]
Ic_Vbe=0,false,true,0
Ie_Vbc=0,false,true,0
Beta_Ic=0,true,false,0

\etc\paramconvert.xml

MBP supports model loading from HSPICE and SPECTRE format. The parameter conversion mapping is saved in this file. The configuration in the file and the corresponding comments are shown below:

```
<Hspice> //The original SPICE format, to be  converted
from.
```

```
<to_Spectre>// The target SPICE format, to be converted
to.
<MOSFET> // Device Type.
<allModels> // for all model types under
MOSFET.
<Delete/> // parameters to be deleted.
<Exchange/> // paramaters to have name
exchanged.
<DeleteByValue/> // parameters to be deleted in case of
specified  value.
<ExchangeNameAndValue/> //parameters to have name
exchanged in  case of specified value.
</allModels>
<BSIM3V3/>//for BSIM3V3 model.
<BSIM4/>//for BSIM4  model.
<BJT/>
<DIODE/>
</to_spectre></Hspice>
```

\etc\modelconf.xml

MBP supports different model types. For each type, MBP allows you to define different versions. The configuration in the file and the corresponding comments are shown below:

```
<device name=*mosfet*> // Define MOSFET
<simulator name=*hspice*> // Define HSPICE format
<condition> // condition of HSPICE MOSFET
<and>
<or>
<polar value=*nmos*/> // nmos and pmos
<polar value=*pmos*/>
</or>
<not>
<or>
<parameter name=*level* value=*57* type=*double*/> //
Not SOI devices
<parameter name=*level* value=*70* type=*double*/>
</or>
</not>
</and>
</condition>
</simulator>
```

```
<model name=*bsim3v3*> // Under MOSFET HSPICE, define
Bsim3v3 model type
<simulator name=*hspice*>
<condition>
<or>
```

Using MBP

```
<parameter name=*level* value=*49* type=*double*/> //
Bsim3 level
<parameter name=*level* value=*53* type=*double*/>
</or>
</condition>
</model>
```

\etc\rfgraph.ini

The RF module in MBP supports smith chart and polar chart dealing with the S/Y/H/Z parameters. The plot property is defined in this file as well as in GUI. You can also modify it in GUI (shown in Figure 2.22) by clicking Tool > GUI options > RF Graph Config.
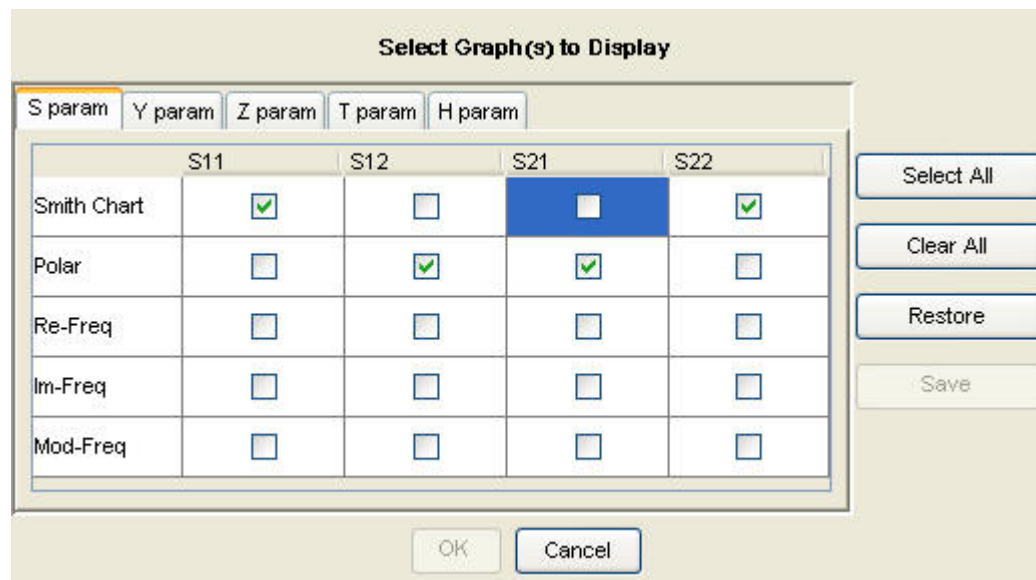
The structure of rfgraph.ini is as following:

[S param]
Smith Chart=1,-1,-1,1
Polar=0,1,1,0
Re-Freq=0,0,0,0

The syntax is:

plot_format=flag_s11, flag_s12, flag_s21, flag_s22

where flag_sxx can equal to 0, 1, -1. 0 means it is not plotted, 1 means reversely, and -1 means that this chart is not selectable.

NOTE Make sure that one RF model is selected first to enable the **RF Graph Config** option in the main menu.
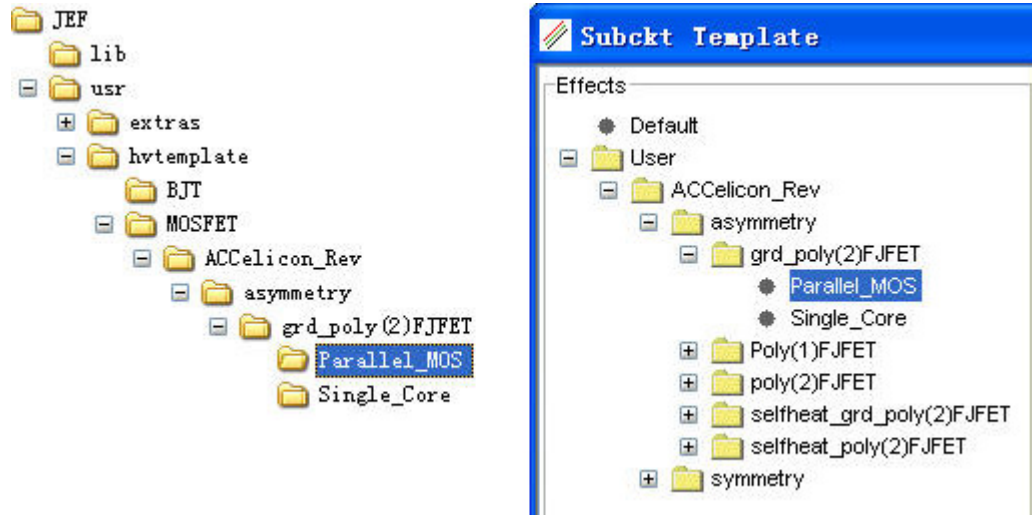
Files under usr

Under this folder, there exist the files which define many MBP features, such as IMV pages, Task tree, Error monitor, etc.

Files under \jef\usr\hvtemplate

You can create or load their own HV templates here. The folder's structure is the same as the tree structure in subckt template. One example is shown in following figure.

Template Tree Structure

NOTE	You can create as many levels of directories as wanted. However there should have four files in the last level of the directory: The two model files (model_nmos.l and model_pmos.l) are necessary for MOSFET while two JPEG files (which for the effect and schematic) are optional.*

NOTE	Refer to the Subcircuit Template section in Utilities for the steps to create templates in MBP.

Files under \jef\usr\rftemplate

Same as HV templates described above. This folder is also fully customizable so you can add your own RF model formats easily.

Files under \jef\usr\math

This folder contains the math-transform functions. Adding these functions needs programming knowledge on JAVA.

Files under \jef\usr\target

Files under this directory are for error monitor usage. There are three files in default. The file rule.index contains the activate rule files list. And the criteria rules are recorded in rule files mos.rule and soi.rule. The rule file syntax is shown as below:

```
\[Title\]
  value:vth0\(mv\)
```

```
    \[Definition\]
    value:Vth@vbs=0
    \[Target_type\]
    value:vth
    \[Enabled\]
    value:yes
    \[Warning_criteria\]
    value:0.03
    \[IsAbsolute\]
    value:yes
    \[Scale\]
    value:1000
    \[Range\]
    index:1
    page:Ids_Vgs_Vbs\|Vds=Vdlin    pvalue:0
....................................
```

Each rule contains eight fields, they are: [Title], [Definition], [Target_type], [Enabled], [Warning_criteria], [IsAbsolute], [Scale] and [Range] .

In the field of [Title], specify target name. Avoid using the same title for different target.

[Definition] section defines tips displayed on screen tp help clarify target definition. It gets displayed if pointed to the corresponding rule, the corresponding GUI window in this example is shown in following figure.

Target Title and Tips

[Target_type] section defines the target value in particular rules. It should be set as one of the following:

- A variable like Vth or Idsat

- RMS

- MAXRMS

[Enabled] section enables certain target in error monitor by setting the value to yes. Otherwise this target is not displayed on the error monitor screen.

[Warning_criteria] section defines the criteria value of the target. MBP uses red color (by default) to plot this target if its value exceeds the criteria. If a variable is set as [Target_type], the [Warning_criteria] could be either absolute value or relative value, depending on how [IsAbsolute] is set.

[IsAbsolute] section defines if this value is an absolute value or a relative (ratio) value, like RMS.

[Scale] section is for unit control. For example, the default unit of Vth is Voltage. If the scale value is set to 1000, then, MBP displays Vth in mV as unit. (1mV*1000=1V) However, it does not affect the values in [Warning_criteria].

[Range] section is to select particular page to get the target value. You can define the page name, certain region in page or even the curve with certain P value in the pages. It contains several keywords as following:

- Index: An integer value. More than one index for a rule means that it can be applied under different conditions.

- Page: To select one of available device pages.

- Pindex: An integer value. To identify which curves to select, use "," to separate different curves, like "0,1,4". All curves are selected by default.

- Pvalue: To specify which curve to select. However, it could uses built-in variable, like "0,VBB". By default, all curves are selected.

- Mathfunction: Choose one of these math functions: "y", "1/y", "y' ", "y'' ", "dx/dy". Math function defaults to "y".

- XRegion: To specify x-axis region on selected page. By default, it is (-inf, inf). You can also use other variables. For example: (vth, vth+0.2)

- YRegion: To specify y-axis region on selected page. By default, it is (-inf, inf). Another example: (-inf, 0.8*ymax)

The following lists the variables you can refer in the rule, some rules have been discussed already:

- VGG: max VGS

- VDD: max Vds

- VBB: max Vbs

- VDLIN: minor Vds used in Ids_Vgs_Vbs page

- Vth: subthreshold voltage

- Idsat: saturation drain current

- Xmin: the min x value

- Xmax: the max x value

- Ymin: the min y value

- Ymax: the max y value

- -inf: the negative infinite value

- inf: the positive infinite value.

> **NOTE** Multiple regions are supported by adding multi-range section with different Index values. * *For detailed information on how to customize the error monitor, please refer to the Error Monitor section in Utilities.

Files under \jef\usr\tasks

The files regarding task tree are placed here. You can customize the task tree by clicking Extraction > Task Tree from the main menu, as shown in following figure. The interface of Task Tree can be seen as a tab besides the Device Navigator.
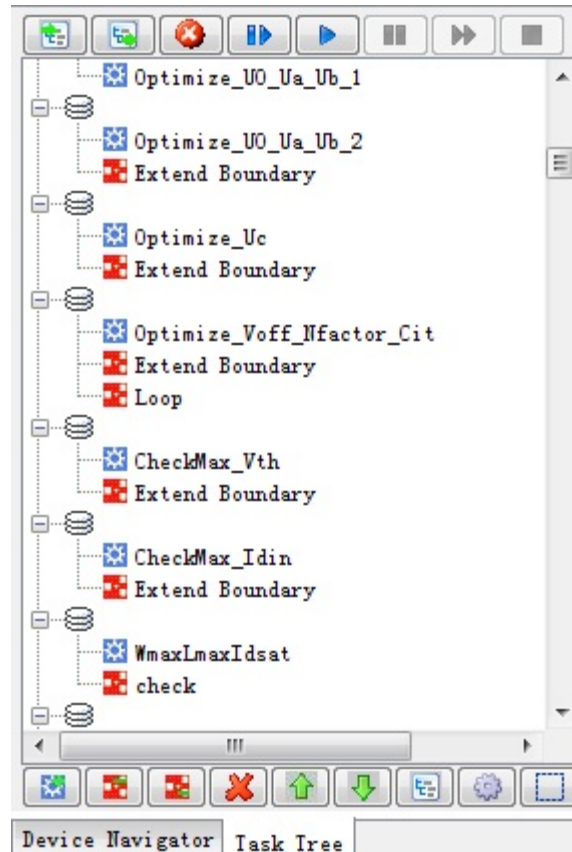
Files under \jef\usr\extras

The two folders (analysis\ and mete\r) in this directory are both for the measurement related operations. Folder analysis\ is for the target definition. There are many files for different curves such as Cgd_Vds_Vgs.class, Ids_Vds_Vgs.class, etc. in this folder. Similarly, measurement instruments can be added in the folder meter. Provide necessary documents (such as programming manual) if new instrument is required to be supported in MBP.

Files under \jef\usr\imv

IMV stands for intermediate variable. Under this directory, all the configuration files regarding IMV pages are stored on the basis of device type. Taking mosfet as example, there are seven files under <MBP HOME>\jet\usr\imv\device\mosfet.

algorithm.ini

File algorithm.ini defines the algorithm for imv target calculation. It is used for target calculation method definition, an example is shown below.

[GM] _name=GM
algorithm=acc/Derivative
parameter=(vgs,ids)
pagename<em>=Ids_Vgs_Vbs_

- name defines the name of the algorithm.

- algorithm defines the method to obtain the target.

- acc/Derivative is its internal function for derivative calculation.

- parameter specifies what parameters to use in the calculation. In this example, Ids, and Vgs is used.

- pagename defines which pages are used in the calculation. Ids_Vgs_Vbs is used in the example.

There are seven internal functions available in MBP. All these functions are related to the curves (or points) chosen by the restrictions (Imv_target_def.ini) from selected page (algorithm.ini). A list of internal functions is shown below:

1. (acc/Current) Get the Y-axis value.

2. (acc/CurrentDivideW) Similar to 1 and normalized by W.

3. (acc/CurrentMultiplyL) Similar to 1 and normalized by 1/L.

4. (acc/CurrentDivedeWL) Similar to 1 and normalized by W*L.

5. (acc/CurrentDivdeWMultiplyL) Similar to 1 and normalized by W/L.

6. (acc/Derivative) Get the derivative value.

7. (acc/GMax) Get the maximum value of gm. *Notice: All the names of these functions are case sensitive.

cal_target_def.ini

File cal_target_def.ini defines the target of imv calculator and the default parameter sweep.

dp_target_def.ini

File dp_target_def.ini defines the target especially for DP data.

imv_const.ini

File imv_const.ini defines the imv constants value, such as Vgg, Vdd.

imv_graph.ini defines the plot of imv pages.

File imv_graph.ini specifies the contents to plot in MBP.
[Plot_Name]
X-axis definition
P-axis definition
Y-axis definition
Hierarchy setup

> **NOTE** When plotting expression targets, parameter should be specified to certain items of the expression, e.g. x=vbs/Idsat

Using MBP

imv_target_def.ini defines the imv target, including the algorithm used and parameter restrictions.

After defining the algorithm of the targets, configure file imv_target_def.ini to define what is used in the plot. There are three kinds of applications in this file:

#the first application:
#refer to other targets, by name
[vth]
name=vth
refer=vth_gm,vth_con,vth_con_LogId
current=vth_gm

You can choose to use vth_gm, vth_con or vth_con_LogId using options on GUI.

#the second application:
#expression of other targets, by name
[vth1]
name=vth1
exp=vth_gm+vth_con

By this way, MBP can calculate certain targets using existing values and simple calculation functions. All the existing targets and the parameters in restrictions could be used for expression.

#the third kind of application:
#define a target, with algorithm and its restrictions
[vth_gm]
name=vth_gm
algor=vth_gm
restrictions=(device,absmax(vds),absmin(vbs))

This indicates that vth_gm value is from vth_gm method in algorithm and it is a saturation threshold voltage (with the maximum vds and minimum vbs).

target_graph.ini

File target_graph.ini defines the plot especially for DP data.

For more information about how to customize IMV, refer to IMV Pages and DP Implementation.

Measurement Files

Measurement files include two types: Measurement Setup Files and Measurement Data Files.

Measurement Setup Files

> **NOTE** Since, DCP has replaced the old measurement module which is stopped being developed now. The content in this section is obsolete and kept here only for the users using the old version of MBP. New users can refer to DCP user manual ($MBP_HOME\dcp\doc) for more powerful and flexible feathers.

Measurement Setup files are to store measurement instrument information and bias conditions. By default, the measurement setup files have the extension of .conf . An example of the Measurement Setup window is shown in following figure. The first label page contains instruments information and node assignments. And the other label pages are bias configurations. Correspondingly, measurement setup files are also consisting of two types of files: the instrument setup file and the analysis setup files.

Measurement Setup window
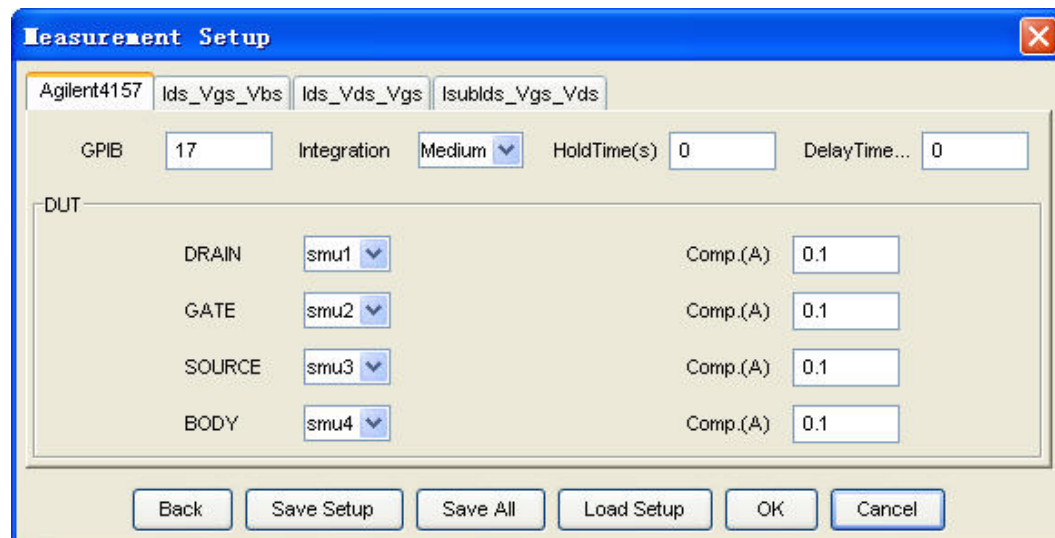
For the analysis setup files, each analysis has its own setup file. As illustrated in Figure: Saved Measurement Setup files, bias condition of analysis Ids_Vgs_Vbs is put into file Ids_Vgs_Vbs_NMOS.conf. To save the setup, you can either save all the setup (instrument setup and analysis setup) to one file by clicking Save All in Figure: Measurement Setup window or save the setup to separated files by clicking Save Setup. Similarly, when to load setup, you can either load all measurement setup files or just one of setup files into measurements.

Saved Measurement Setup files

Measurement Data Files

Measurement data files are used to store measurement data. By default, the measurement data files have the extension of ".mea". You can open and edited the same using text editor. MBP supports various data types (such as DC, Mismatch, RF, Statistical, etc.). It is described in the following section.

DC Data File

An example of DC data is shown below:

```
condition {mode=forward,type=nmos,modtype=DC}
Page (name=ids_vgs_vps,x=vgs,p=vps,y=ids)  {vds=1.0,ves=
0.0,W=0.18,L=0.15,T=27.0 }
curve { 0.5 }
0.0      1.1024053054394643E-10
0.05      3.2827790989388004E-10
0.1      1.223897310845409E-9
0.15      4.802207520961346E-9
0.2      1.8552352100490592E-8
0.25      6.847457189856461E-8
0.3      2.3744388358213807E-7
0.35      7.582069804395346E-7
0.4      2.1063300093137126E-6
0.45      4.785803862526468E-6
0.5      8.884813678495436E-6

......
```

The first line of the data file contains the information of working mode, device type, and voltage mode. From the example above we know the nmos device is in forward working mode and under DC analysis.

The second line defines all page related information. The information within round bracket ( ) contains Page Name, X variable, P variable and Y variable. The information within brace { } declares Page Constant including bias condition, device geometry.

The latter is the data block information. We know that each page may contain several curves with different P values. For every curve block, it always begins with key word curve and the corresponding P value. All data points are then listed behind it. The first column is for X variable and the second column is for Y variable, separated by either space or TAB. Other delimiters are not recognized. The example data plot is shown in following figure.

Content of measurement data file and corresponding page
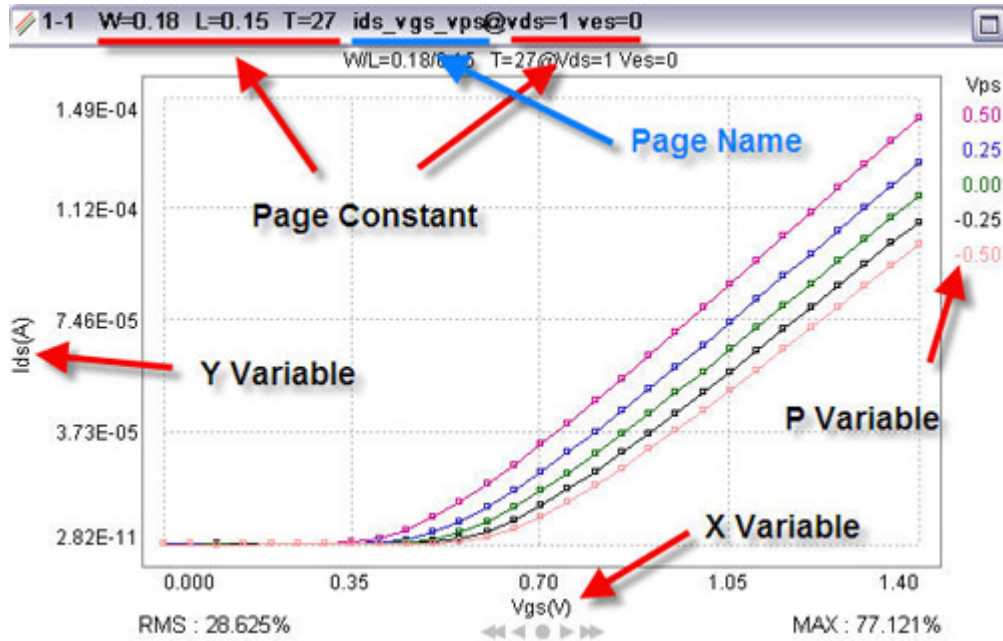
NOTE	The page name and variables are defined in the file analysis.xml under `<MBP HOEM>\etc`. See section *\etc\analysis.xml* for more details.

Mismatch Data File

MBP supports two kinds of data formats for mismatch. The first one is base on the actual measurement data. And the second one allows you to input the mean and sigma value of the target. It is introduced in the following section. Below is an example of the first one:

```
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    -0.001
6386308204789923    0.0012339958926110839    0.008895426
625025848    -0.0048731228537369775    -0.01162406783793
8012
  -0.006426794909106914    0.003820585385617581    -0.00
3906291053092925    -0.005558806143440509    0.003262499
108588224    -0.004310905700422363    -0.004533647758578
008
  -0.005645149082360135    0.005513986383539615    -0.00
12684190995226885    -0.004787090185292664    -0.0136843
2377987483    0.0055176400252267    0.023908953534973687
  -0.0031491340681994995 -0.004776220049452662    -0.010
298545804177328 ...........................
```

The first line of the data file begins with the keyword miscondition and contains the information of time and device type.

The second line defines all page related information. The information within round bracket ( ) contains Page Name, target, scale, and P variable. The information within brace { } declares Page Constants including bias/current condition and temperature.

The latter is the data block information. For every curve block, it always begins with {L W} . All data points are then listed behind it. They are the vth difference between two adjacent devices with the same geometry (in this example it's the device of L=0. 18 and W=2.0). The example data plot is shown in Figure Mismatch data graph

Mismatch data graph
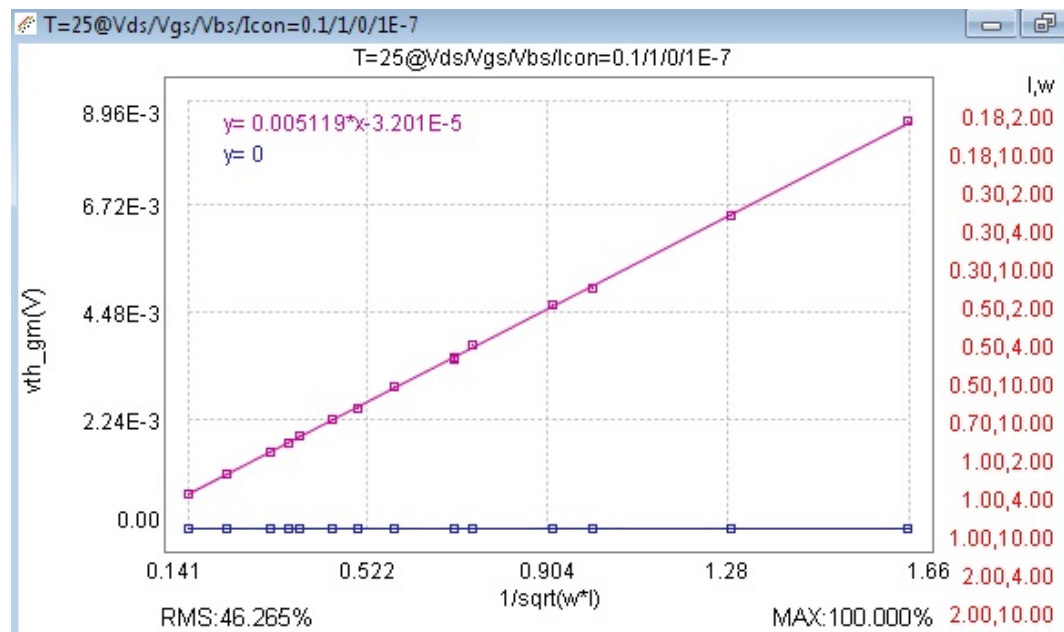
You can also choose the other format for the mismatch data, the example is shown below:

```
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 the information such as corner type, time, 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 are then listed from the second line. It is noted that the last two values in every line are corresponding to the mean and the sigma of the target. In this 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, Ids(mean) =0, and Ids(sigma)=6.448e-4A* .

MOSRA Data File

An example of MOSRA data file is shown below:

```
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      5.0002933308273624E-14
  0.05      5.0016786281530424E-14
  0.1      5.00960618552065E-14
  0.15      5.054972647299803E-14
  0.2      5.3145867705084785E-14
  0.25      6.800237337193422E-14
  0.3      1.5301686407243485E-13
```

The first line of the data file contains the information of corner type, time, measurement instrument, working mode, data type, model version, and device type.

The second line defines all page related information. The information within round bracket ( ) contains Page Name, X variable, P variable, and Y variable. The information within brace { } declares Page Constants including bias, geometry, and temperature. Finally the keyword stress contains the information such as time and bias conditions which applied in the aging test.

The latter is the data block information. For every curve block, it always begins with key word curve and the corresponding P value (in this example it's the stress time). All data points are then listed behind it. There are two values in every line: the first is the value of X variable (Vgs) and the second is the value of Y variable (Ids). The example data plot is shown in Figure Mosra_data

Mosra_data
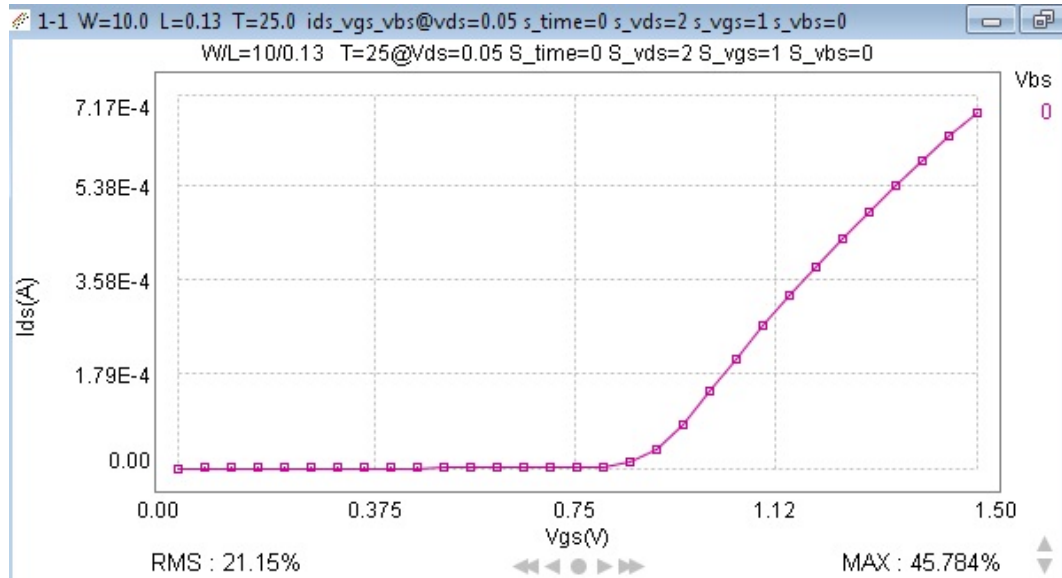
RF Data File

An example of RF data file is shown below:

```
[General]
  Datatype:rf
  Version:1.0
  Operator:
  Date:
  Lot:
  Wafer:
  Die:-1,0
  Tile:label 0
  Device-type:mosfet
  Type:nmos
  Nodes:
  Instances:w=32,l=0.25,nf=8,ad=2e-11,as=5e-11,pd=5e-5,
ps=5e-5
  Temperature:27
  Process:
  Ports:g,d
  Z0:50
```

```
[Measurement Condition]
  DataName:S\-parameter
  Measurement-condition:S-parameter
  Biasing-type:S-parameter
  Biases:vgs,Vbs,Vds
  rfoutputs:S
  dcoutputs:
```

```
[DC Data]
```

```
[RF Data]
  {Vds=0,vgs=0,Vbs=0,x=freq}
  1.0E8,      0.999983,     -0.005338426,     1.184265E-5,
0.002201805,     1.184264E-5,      0.002201805,      0.999982
,     -0.005427228
  4.98E8,      0.999585,     -0.0265799,      2.935445E-4,
0.0109595,      2.935442E-4,      0.0109595,      0.99956,
-0.0270204
  8.96E8,      0.998659,     -0.0477995,      9.494757E-4,
0.0196952,      9.494748E-4,      0.0196952,      0.998578,
-0.0485849
```

There are four modules in above data file.

- [General] Define the general information such as device type, instance, temperature, etc.

- [Measurement Condition] From the above example we know that it's an S-parameter data file.

- [DC Data] It's optional in the rf data file. In the example above there is no DC data available. RF_Data
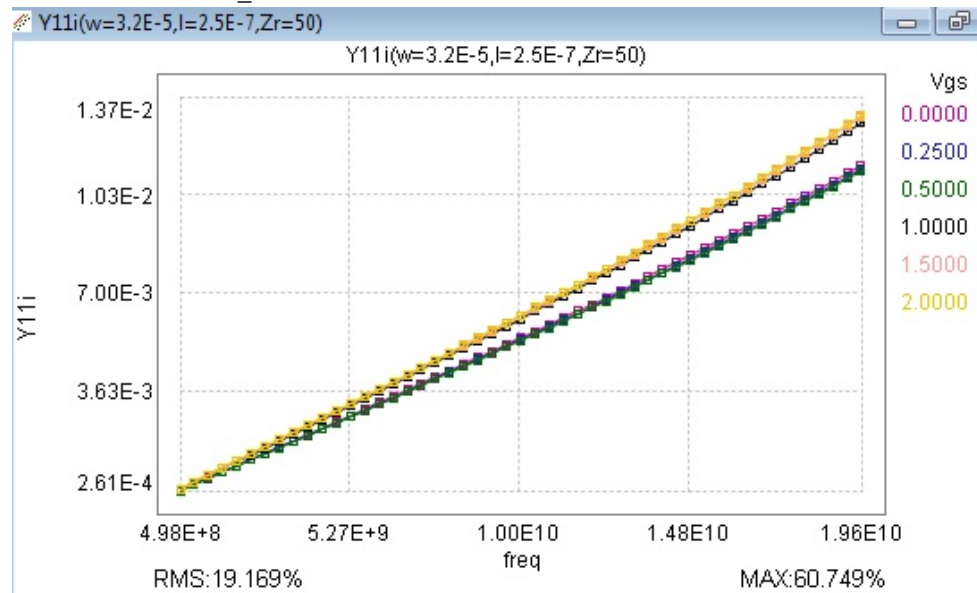
Statistical Data File

Similar with mismatch, there are also two kinds of data formats for statistical. The first one is base on the actual measurement data. And, the second one allows you to input the mean and sigma value of the target. It is introduced in the following section. Below is an example of the first one:

```
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}
     curve0.0}
     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 the information of working mode and device type. From the example described earlier, we know that it is an nmos device in forward working mode.

The second line defines all page related information. The information within round bracket ( ) contains Page Name, target, and P variable. The information within brace { } declares Page Constant including geometry and temperature.

The latter is the data block information. For every curve block, it always begins with key word curve and the corresponding P value(Vbs in this example). All data information is then listed behind it. There are two values in every line: the site number and the target value(Ids in this example). The example data plot is shown in Figure statistical_data.

Statistical_data
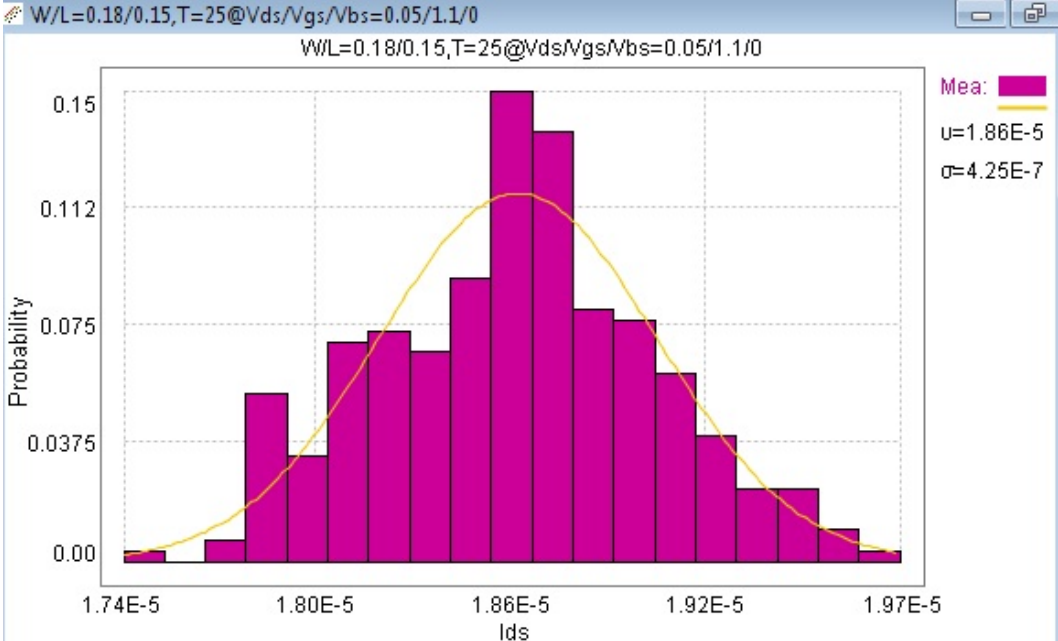
You can also choose the other format for the statistical data, the example is shown below:

```
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
(code}
```

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 information such as corner type, time,
data type, device type,  instance, bias condition, and
target.
The second part of the data file includes the data
block information. Every block  begins with the keyword
*<em>Data</em>* following the variables. All data are
then listed from the second line. It's noted the last
two values in every line are corresponding to the mean
and the sigma of the target. In this example,
```

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*
```

.

Verilog-A Data File

An example of Verilog-A data file is shown below.

```
condition\{corner = tt,date = oct_20_02,instrument=\
(hp4145, probe_station\),mode=forward,type=nmos}
Page (name=ids_vgs_vbs, x=vgs,p=vbs, y=ids) { vds= .05 ,
w=1.000,L=10.000,T=25.000  }
curve { 0 }
0            1.08E-12
.05          4.64E-12
.1           2.07E-11
```

```
.15              9.195E-11
.2               4.0652E-10
.25              1.7173E-09
.3               6.374E-09
.35              1.9707E-08
.4               4.7926E-08
.45              9.3226E-08
.5               1.5256E-07
```

The first line of the data file contains the information of corner type, time, measurement instrument, working mode and device type.

The second line defines all page related information. The information within round bracket ( ) contains Page Name, X variable, P variable and Y variable. The information within brace { } declares Page Constant including bias condition, geometry and temperature.

The latter is the data block information. For every curve block, it always begins with key word curve and the corresponding P value (Vbs in this example). All data points are then listed behind it. There are two values in every line: the first is the value of X variable (Vgs in the example) and the second is the value of Y variable (Ids in this example). The example data plot is shown in following figure.

Verilog-A data graph
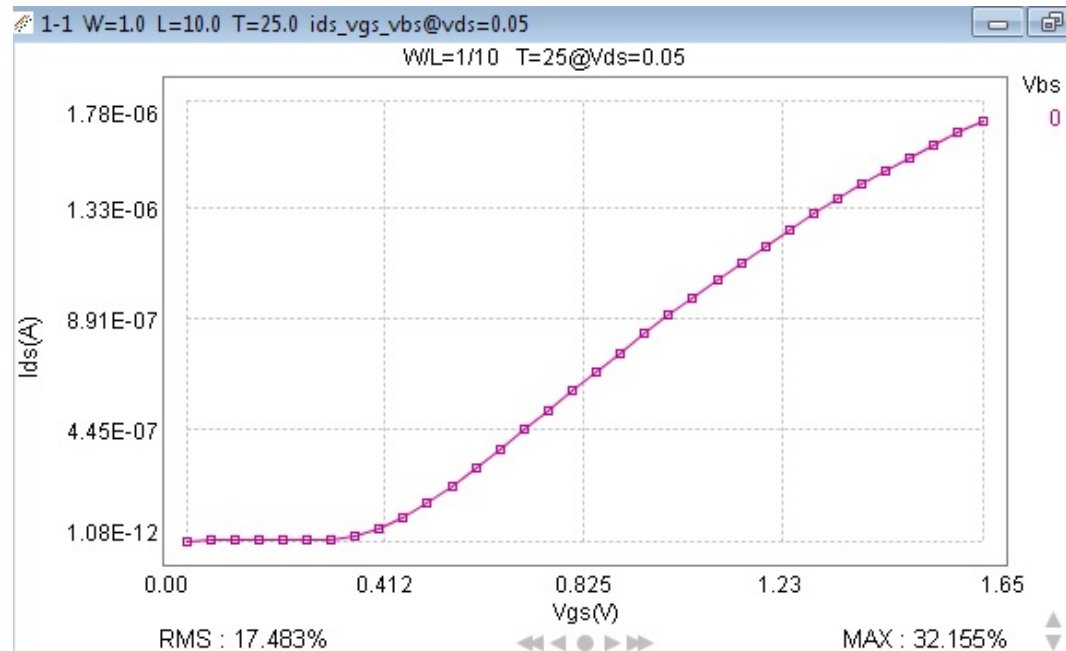
DP Data File

An example of DP data file is shown below.

```
condition{corner = tt,date = oct_20_02,instrument=
(hp4145, probe_station),mode=forward\}
    Datatype{target}
    Version{2.0}
    type{nmos}
```

```
     Delimiter{,}
     Instance{L, W, T}
   Input{Vgg=1, Vdd=1, Vbb=-1,Vdlin=0.05}
```

```
 Data{ w, l, t, vth3, vth5 , Ion5, Ion3, Idh, Idl,
 Ieff,   Imid,  Ioff5, IoffS,  Ib5, Ig5, Ig3, gm, gds}
     10 ,   0.08  ,  25 , 0.651553   ,    0.588125 , 287.850
499,     94.820691      ,        259.189359 ,   2.38974E-1
 ,   2.05312E-1  ,    0.171649    ,      1.606672E-7
 ,     5.966716E-8,         -1.010000E-7,
0  ,                0,          4.946742E-5
 ,        1.308196E-6
    10 ,   0.15 ,  25  , 0.502278   ,   0.447852,  323.444
683,        84.189907    ,        296.491487,     5.705158
 ,    5.097868  ,     4.490579 ,        3.325582E-6,
3.224582E-6,            -1.010000E-7 ,            0
 ,                0  ,        9.845651E-4
 ,        2.416751E-5
    10 ,   0.24 ,  25  , 0.424177   ,   0.374332,  293.543
081,       65.367928    ,        270.459102,    15.541956
 ,    14.333136 ,     13.124317,        2.324948E-5
 ,      2.314848E-5,           -1.010000E-7,
0 ,                 0  ,          2.065517E-3
 ,        4.888236E-5
    10 ,   1    ,  25  , 0.309192   ,   0.267681,  124.335
006,       20.776761   ,        114.883828,     15.649011
 ,    14.901380 ,     14.153749,        1.978629E-4,
1.977619E-4,           -1.010000E-7,            0
 ,                0  ,        1.244241E-3
 ,        3.083130E-5
    10 ,   10   ,  25  , 2.71819E-1,    0.233881,   15.0942
13 ,        2.258570      ,        13.892741   ,    2.181672
 ,    2.102561   ,    2.023449 ,        6.146755E-5,
6.136655E-5,           -1.010000E-7,            0
 ,                0 ,         1.532806E-4
 ,        3.245076E-6
```

The first part of the data file contains the information such as corner type, time, measurement instrument, working mode, data type, device type, instance and bias condition.

The second part of the data file includes the data block information. The keyword Data defines the instance names(w,l,t) and the page names (v

```
  th3, vth5 , Ion5.....\)
```

. All data are then listed from the second line. There are total 18 values in every line in the sequence defined in the first line:

```
w, l, t, vth3, vth5 , Ion5, Ion3, Idh, Idl, Ieff, Imid,
Ioff5, IoffS, Ib5, Ig5, Ig3, gm, gds
```

.

The example data plot is shown in Figure dp_data

dp_data
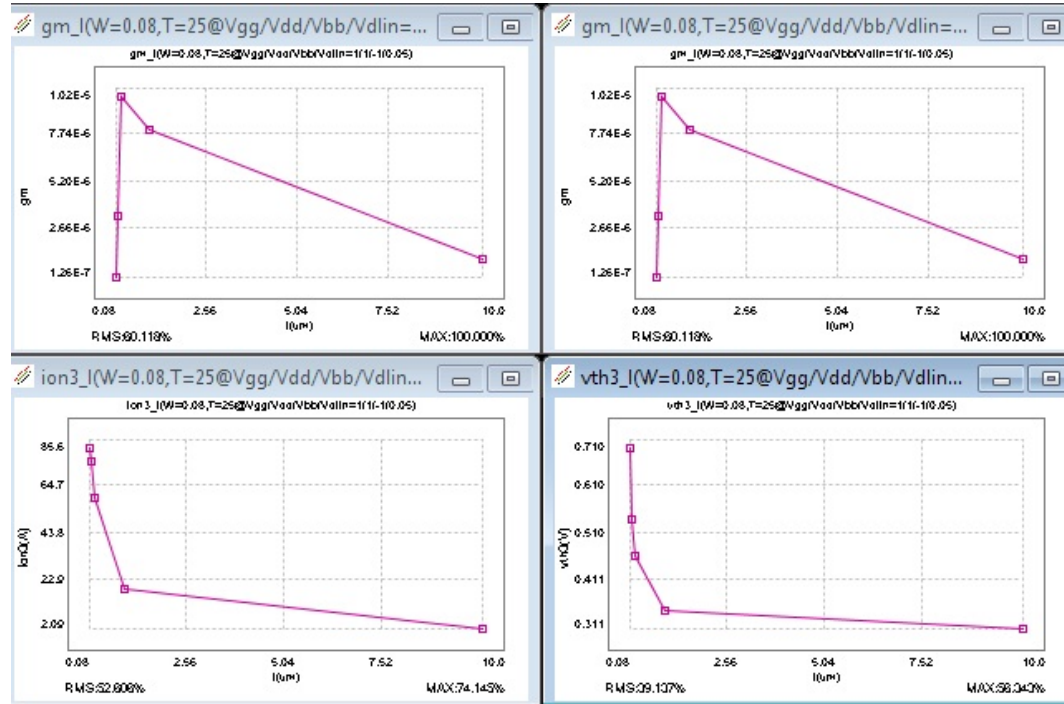
The data files generated by MBP may comprise multiple devices, depending on whether single or a series of devices are taken in the measurement.

Before loading a measurement data file, specify the model type first. If the data file doesn't match current model type, loading is disabled. To choose proper model type, click **Model > Select Model**. It pops up as shown in following figure.

SelectModel_new
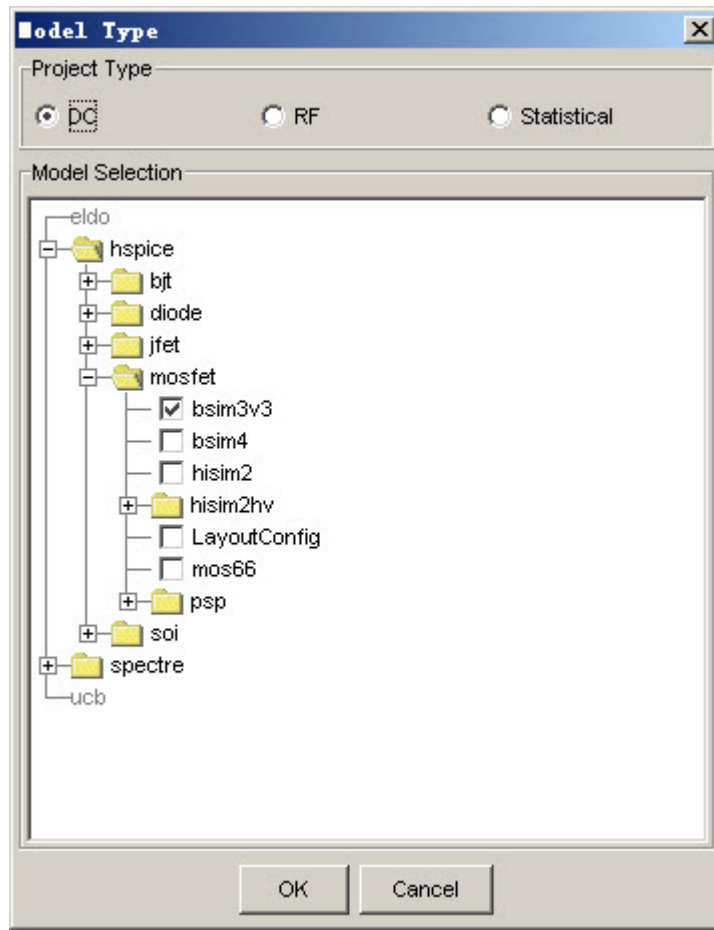
Virtual Measurement Data Files

Virtual data is also one of the measurement data file. They have the same format, except all the values in the virtual data file are set to zero or identical to the simulation values from certain model. Since, simulation in MBP is based on measurement data, this file enables you to run simulation at desired bias and geometry conditions. To know how to generate virtual data, please refer to Utility.

Model Files

MBP fully supports HSPICE and Spectre models. You can load existing HSPICE (or Spectre) compatible model into MBP or export the model parameters in MBP to a model file. Again, specify proper model type before loading an existing model. To load a model, click File > Model > Load Model, and select the desired model card from the file browser window.

To export the extracted model parameters into a SPICE model, click File > Model > Save, and specify the model name and the model directory, then click Save

MBP also enables you to convert HSPICE compatible model/lib to Spectre compatible model/lib directly, by clicking Export > Export to Spectre, shown in figure Convert below.

Convert
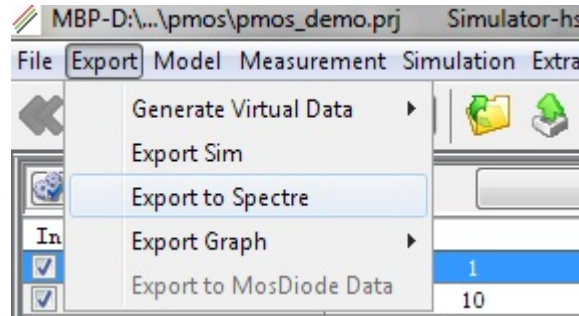
Project Files

The project files store system and program operation information. You can retrieve measurement and model by loading a project.

A new project can be created by clicking File > Project > New from the main menu in GUI. You can load the previously saved projects by clicking File > Project > Open. The pop-up window is shown in Figure: Open Project Window. MBP also provides a convenient feature to find the project files previously loaded easily. Click File > Project > Reopen, the recent opened project list should appear there.

To save the current project, click File > Project > Save or File > Project > Save as... to save as a different project name. The project files are saved in ASCII format, which allows you to read the content of the projects.

Open Project Window
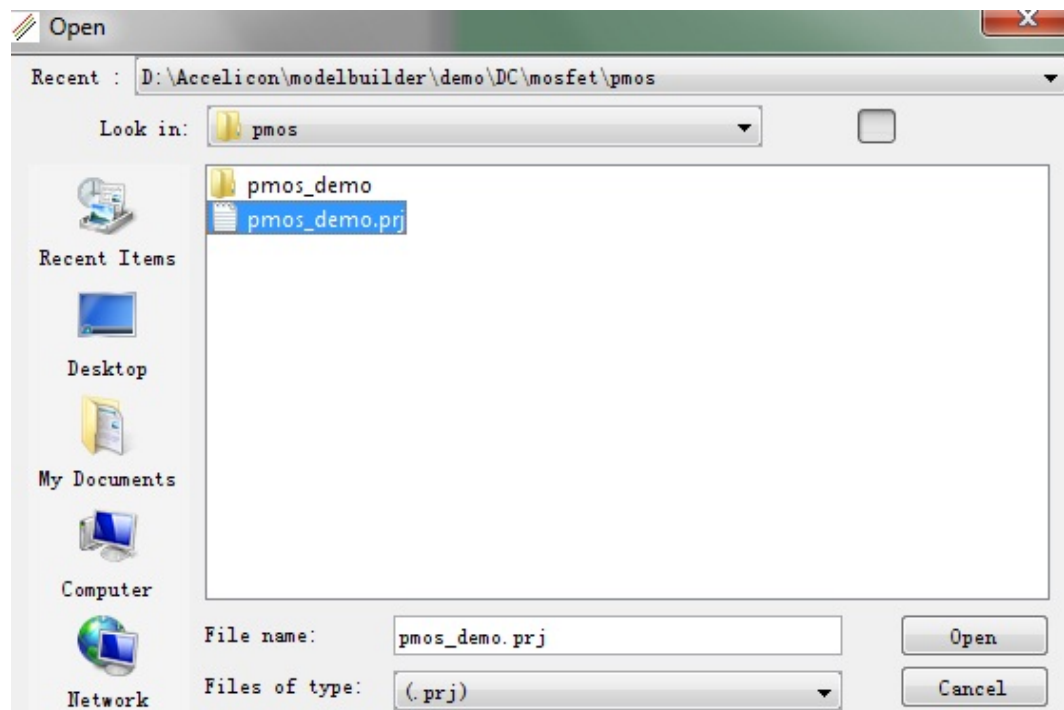
NOTE Project file only keeps the links to measurement data. If you have moved data away from original site, MBP fails to read in the measurement data with project loading. However, for the model, MBP first checks the link. Even it is empty, MBP outputs last saved model to the project. Remember to save the model file after tuning in such case, since, the model is recorded in the project file instead of corresponding to an existing model file.

Parameters

Defining Parameters

Using Optimization Window

MBP integrates parameter window and equalizer together. By default, optimization window is located in the bottom pane. You can also float the window by double-click the Parameters tab. Sample optimization windows in both fixed and floating status are shown in following figure.

Fixed and Floating Optimization Window
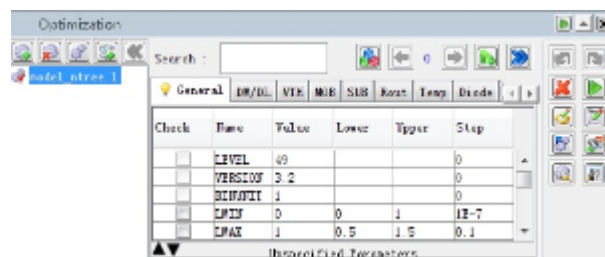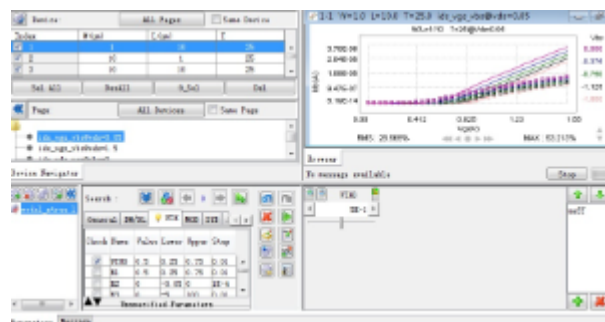
In a model file, some parameters may not be specified. Place the cursor on the Unspecified Parameters so that the double arrow cursor appears. Click and drag the cursor upward, all unspecified parameters (except partial bin parameters for BSIM3v3 and BSIM4) are visible, as show in following figure. Click and drag any unspecified parameter to parameter window, then you can assign the value. You can also right-click on the parameter window and add parameters through Append /Insert option.

Show unspecified parameters
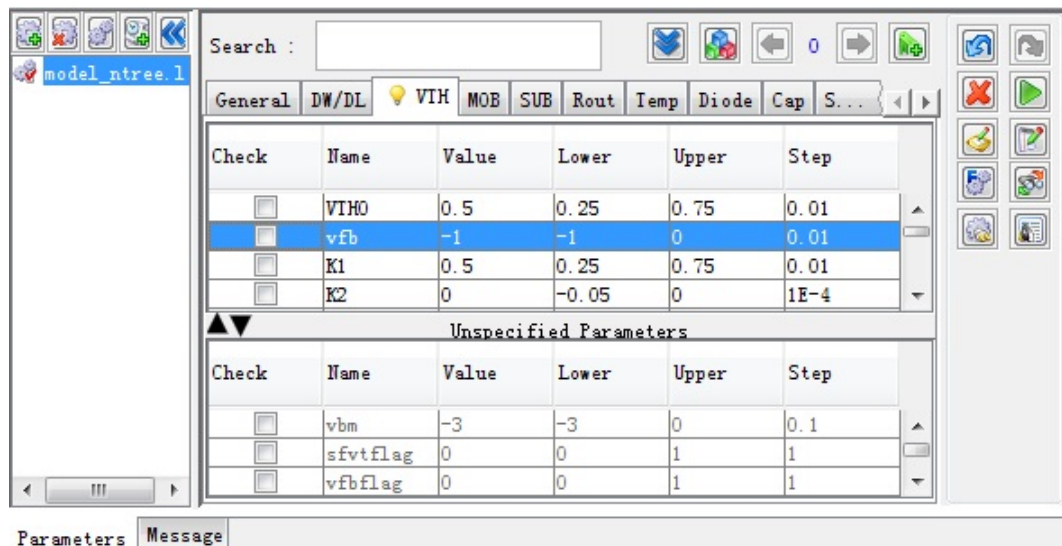
Parameter Selection and Group Selection in Optimization Window

The parameter window organizes the parameters in submenus. The parameter classification routine could be found in the file paramclassify.conf (refer to MBP Files for more details). MBP provides search and autofill functions to find the parameter desired. Type parameter name in the Search bar, and the corresponding parameter gets highlighted in the parameter window. To select the highlighted parameter to perform optimization, press the Enter key or click on the parameter. A sample parameter window with k1, k2, k3, and k3b selected is shown in following figure.

Parameter Window
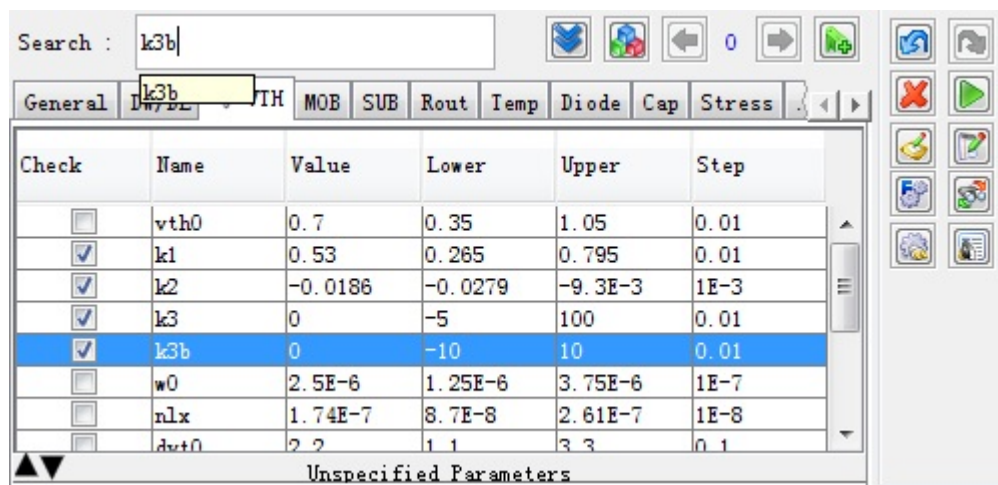
In addition, auto fill function lists all matched parameters in the Search drop down box. You can click one directly or use the Down arrow key to highlight one, then press the Enter key, shown in following figure. MBP also supports wildcard *** and ? , for example, you can type such as *l, *vt?0 to search the parameters.

Auto fill function

Check		Value	Lower	Upper	Step
✓		0.5	0.25	0.75	0.01
✓		0	-0.05	0	1E-4
✓		0	-5	100	0.01
✓	K3B	0	-10	10	0.01
☐	WO	0	0	1E-5	1E-8
☐	NLX	0	0	1E-6	1E-9
☐	DVTO	0	0	10	0.01
☐	DVT1	0	0	1	1E-3
☐	DVT2	0	-0.05	0	1E-3

Search : k

Tabs: General | ... | H | MOB | SUB | Rout | Temp | Diode | Cap | Stress | WPE | G...

Dropdown list: K1, K2, K3, K3B, KETA, KT1, KT1L, KT2

Unspecified Parameters
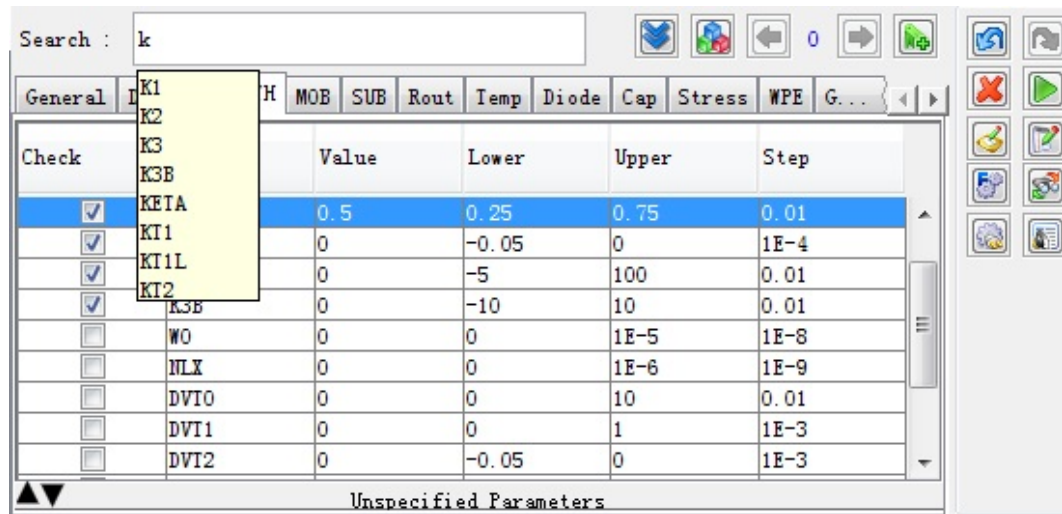
Correspondingly, there are three methods to deselect parameters:

- Uncheck parameter box in the table.
- Click parameter name, hold the left button and drag it back to the list.
- Click Delete All icon to deselect all the parameters.

A useful function, Group Selection is worth mentioning here. During optimization, to fit certain target data, one often chooses certain group of parameters. MBP provides easy way to select model parameters in groups. For example, in the floating optimization window, select parameters Vth0, K1 and K2, and press the arrow button to unfold Group Selection Window as illustrated in Figure: Select parameters in groups. Click the Add icon at the bottom and input the name as vth (user-defined), then vth is added to the Group Selection Window. And whenever you double-clicks on vth, parameters of Vth0, K1 and K2 gets selected from the table. Click the Remove icon beneath the window to remove the highlighted Group Selection from the list.

NOTE — User-defined Group Selection is available to all projects with the same model type.

Select parameters in groups
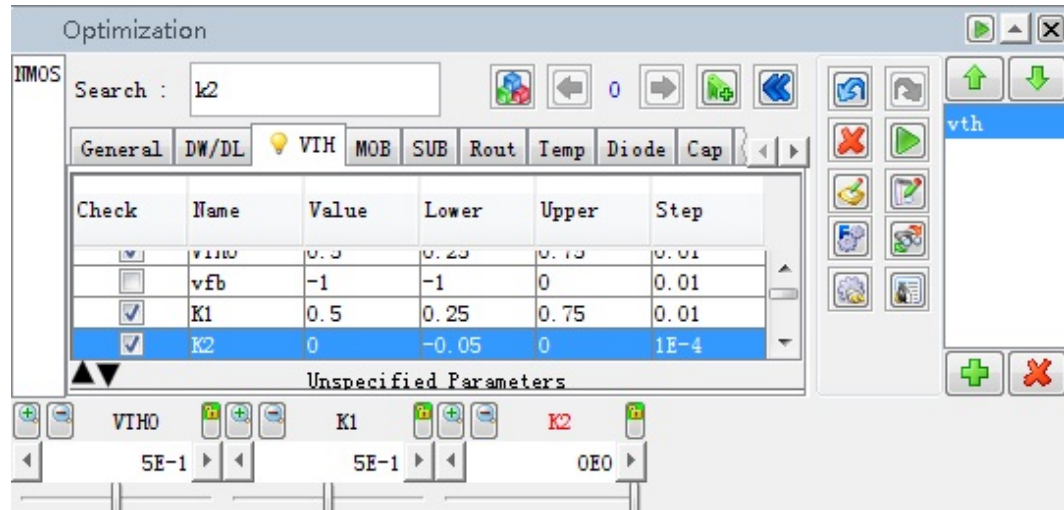
Editing Parameters

Following operations can be performed on parameters: Add, Remove, L, W, P, Set Default, Enable Hard Boundary, Parametrize, De-Parametrize and Comment.
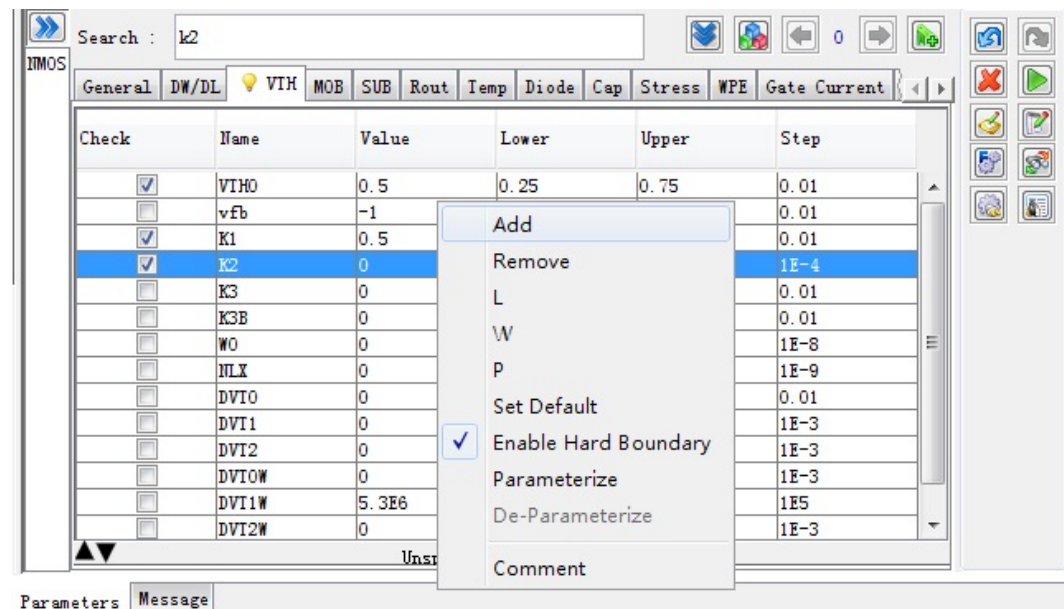
Operation	Description
Add	To add new parameters to the table, fill all the items in the pop-up window before pressing OK but following figure. Add a new parameter as follows:

Operation	Description
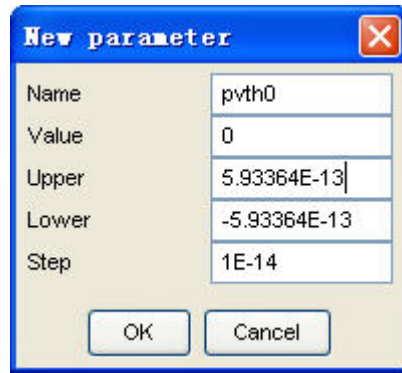
Remove	To remove the highlighted parameters from both the table and model file.
L, W, P	Only if highlighted parameter is binnable, this option becomes available.Left click on L/W/P inserts bin parameter to the model. For example, select **Vth0** and click **L**, **lvth0** gets inserted.
Set Default	To reset the parameter to its default value.
Enable Hard Boundary	Click to enable hard boundary
Parametrize	To parametrize the parameters. It only can be used for the Non-binnable and Bin-core parameters.
De-parametrize	To remove parametrize.
Comments	To activate new window named **Parameter Guide** as follows:

Operation	Description
	![Parameter Guide window](Parameter Guide) Search : vth0 mobmod capmod tempmod nqsmod acnqsmod noimod stimod wpemod igcmod igbmod lmin lmax wmin wmax tox toxm xj gamma1 gamma2 nch nsub vbx xt DW/DL VTH vtho vth0 Parameter -- vth0 binable parameter. Threshold voltage Info Default Value : 0.7 Hard Boundary : [-100.0,100.0] Alias : vtho Bin Type vth0 = vth0 + lvth0/L + wvth0/W + pvth0/(L*W Extraction Tip Equations $V_{th0ox} = V_{th0} - K_1\sqrt{\Phi_s}$

> **NOTE** The added parameter must be **legal** to the simulator. You can find all supported parameters (except some bin parameters of MOS model) in model settings files described in MBP Files*.

If you want to adjust the parameter value, upper and lower boundary etc, double-click the parameter name to pop up the parameter adjustment window as shown in [Figure: Parameter adjustment window|#paramadjwin}. A more convenient way of adjusting parameter boundary is clicking Expansion icon ![icon] and Back the expansion icon ![icon] . You can specify the scale for expansion by percent. And the default value is 50%.

Parameter adjustment window
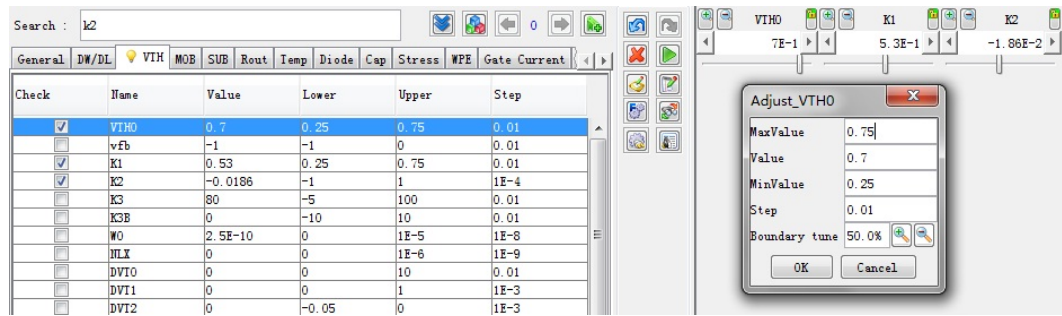
NOTE

- The values for **Upper**, **Value** and **Lower** should be always in descending order.

- Model parameter may have alias. MBP gives warning message if parameters that are already in model file are added.

You can also click the right arrow ▶ and the left arrow ◀ under each parameter name to adjust the parameter value. Another easy-to-use way of tuning parameter is by simply moving the bar ⊥ corresponding to the parameter.

There are three modes available:

1. Fast commit mode: Both the parameter value and the simulation curve response immediately when dragging the bar.

2. Slow commit mode: Only the parameter value responses immediately when dragging the bar. The simulation curve only changes only after releasing the mouse.

3. Manual commit mode: Only the parameter value responses immediately when dragging the bar. The simulation curve does not show any change. These three modes can be switched in order by clicking the icon highlighted in following figure. In the example, F means it's in the Fast commit mode now.

Switching between Fast/Slow/Manual modes
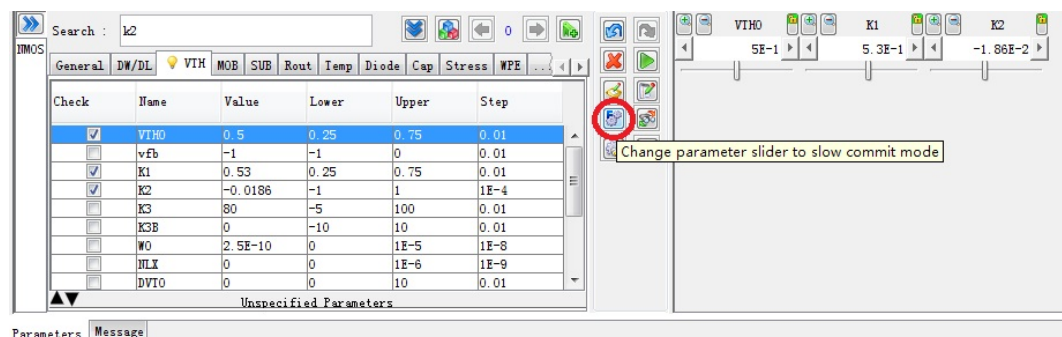
Also, you can edit parameter from the parameter table. As illustrated in following figure, double-click on the value of K2, and type the new value. And if this value exceeds upper or lower boundary, MBP adjusts its boundary automatically.

Input parameter value directly
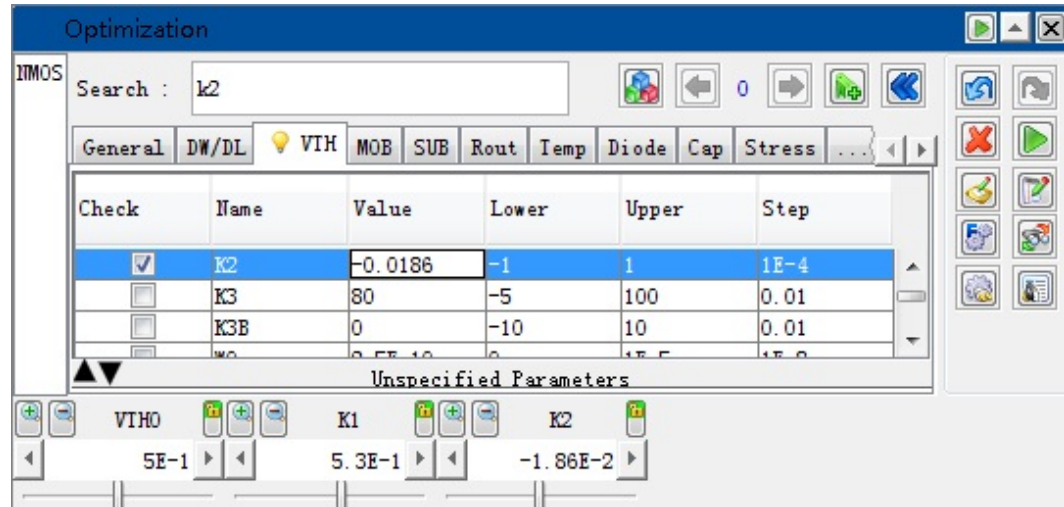
| NOTE | In MBP, some flag parameters like **capmod** in BSIM3v3 model could not be CHECKED or SELECTED from the table. You can only specify their values by directly inputting the value, as described above. |

Locking Parameters

This function is used to keep parameter value unchanged during the optimization. To achieve the same, click the lock icon as shown in following figure. The background color of the icon turns to red after this action.

Lock parameter
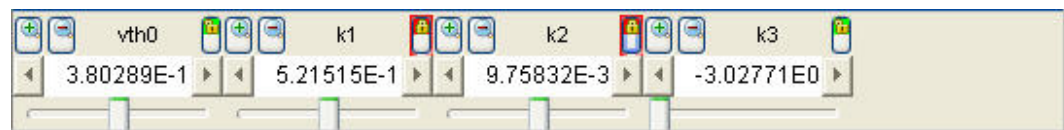
Parameter Equation Editor

Right click on parameter to tune – Equation Parameters – Enable the tweak items of equation:
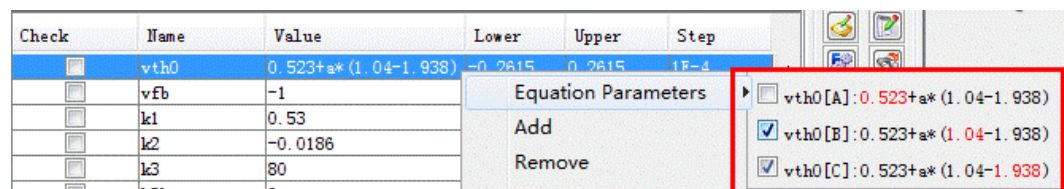
When select the parameter, all the enabled items will be shown and ready for tune.
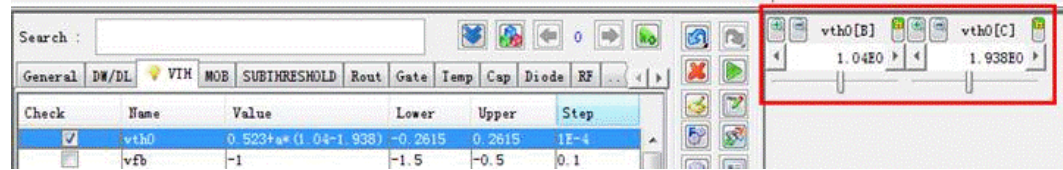
If the equation parameters are not enabled, the selection of parameter will still show delta value as before
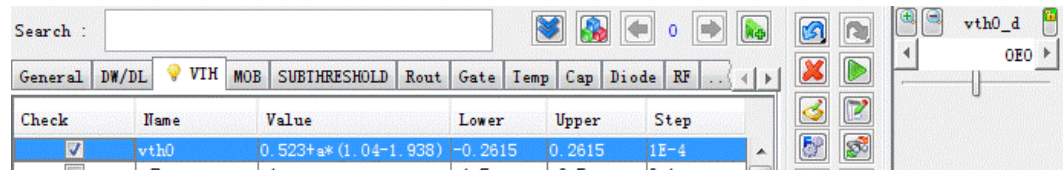
Synchronizing Parameters

This function is used to keep two parameters synchronizing during the optimization. Choose the parameters expected to be synchronized first. After clicking Synchronize parameters icon in the optimization window (shown in Figure: Synchronize parameter), a new window (shown in Figure: Synchronize parameter window) pops up to show all the synchronizing parameters.

Synchronize parameter
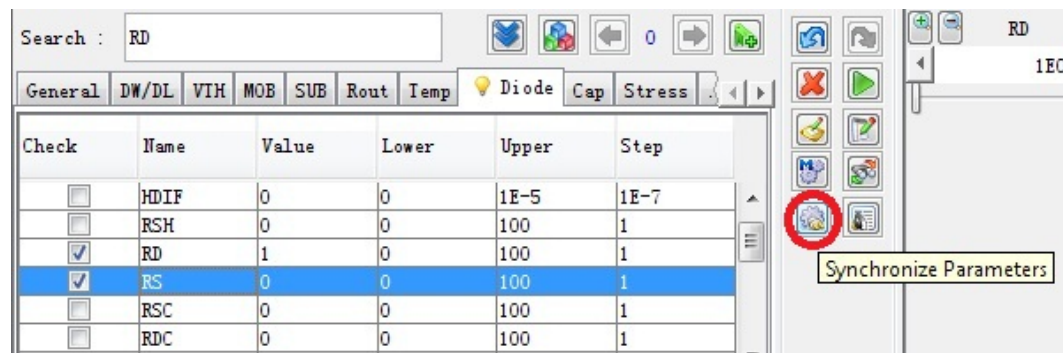
Synchronize parameter window
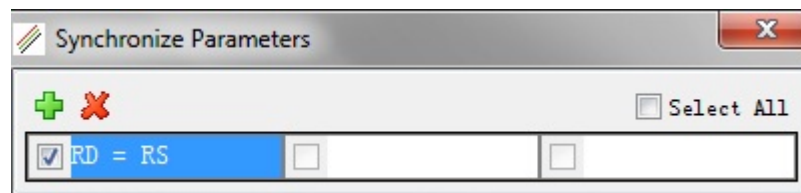
Add the new synchronization, and then check on the box in front of the equation to enable it. If the two parameters are not equal before the synchronization, a warning window pops up. You can choose which parameter value to used as current value for the two parameters, as shown in following figure.

Synchronize parameter warning window
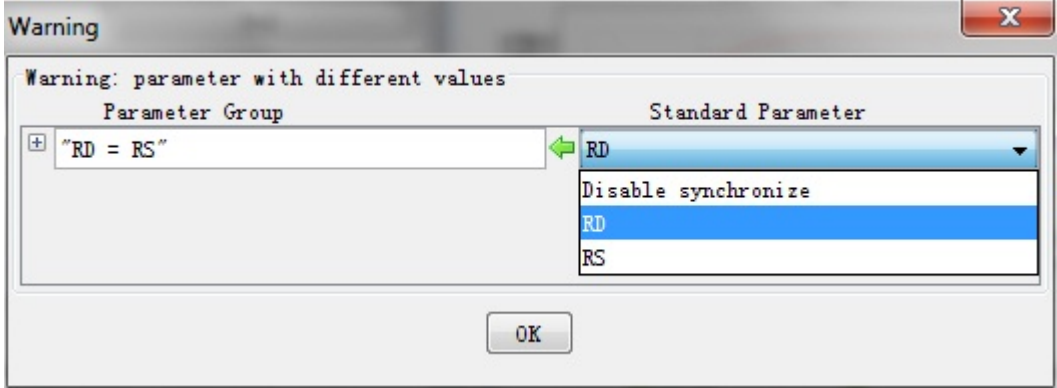
Simulation

Invoking External Simulators

With its internal engine, you can simulate most of macro and compact models with good efficiency and speed. MBP also provides an option to call other simulators for verification. You can invoke external simulators by clicking Simulation > Simulator from the main menu. There are total four external simulators supported by MBP so far: HSPICE from Synopsys, Spectre from Cadence, SPICE3 from UCB, and ADS from Keysight.

You can configure the external simulators by choosing Simulation > External SPICE Options from the main menu. The pop up window is shown in following figure.

External SPICE options
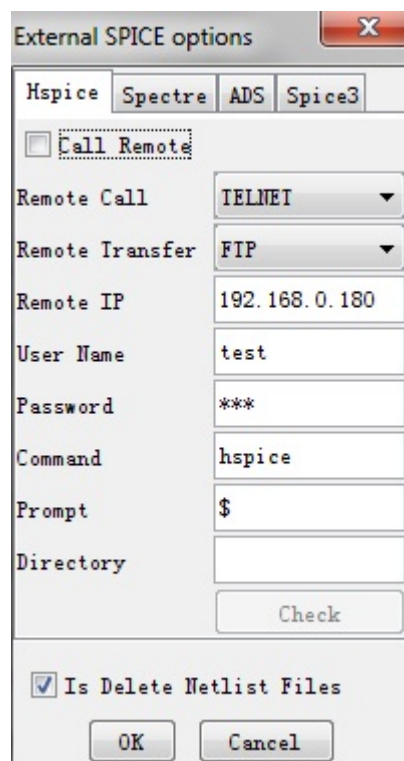
Optimization

Optimization Process

Optimization is one of the most important steps in model extraction. Besides target parameters, it involves the selection on the device, region, and curve for the optimization.

Optimization Region Selection

To select region for optimization, select device first. You can use device navigator to pick up target devices as described in Device Navigation.

Select Single Region for Optimization

MBP allows you to select data points from either single device or multiple devices for optimization. You can press and hold Ctrl key and use cursor to draw a designated rectangle as the optimization region. The mark of the data points within the region becomes solid square to be differentiated from unselected data points. In addition, once a region is selected, the RMS and MAX values under plot are updated immediately to reflect the fitting error between simulation and measurement in the selected region. As shown in following figure, the original RMS and MAX are 1.645% and 3.332%. However, in the selected region, they become to 2.119% and 3.218%, respectively.

RMS and MAX values
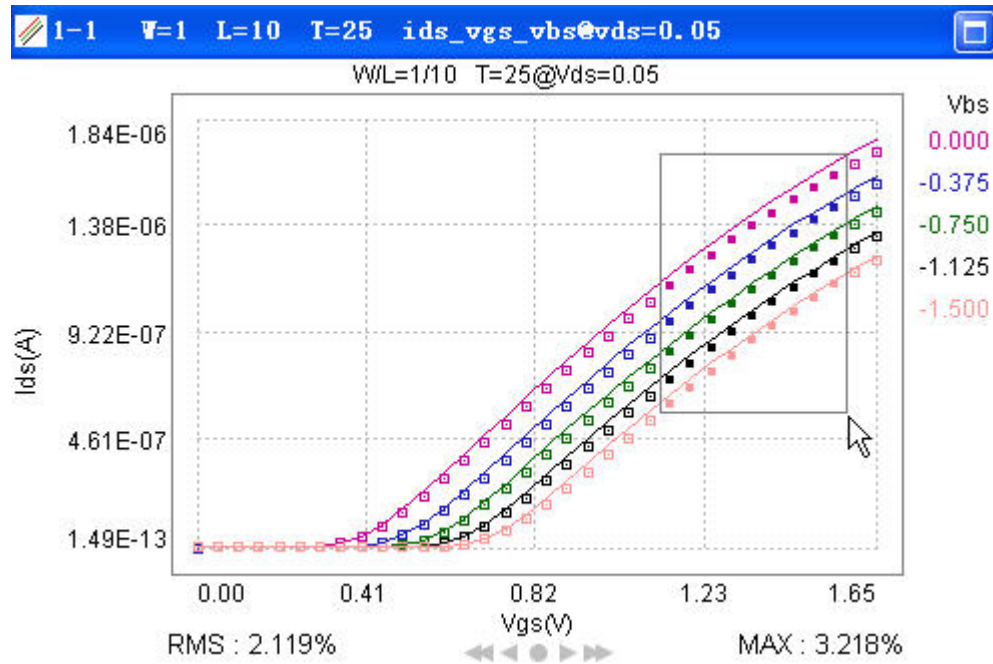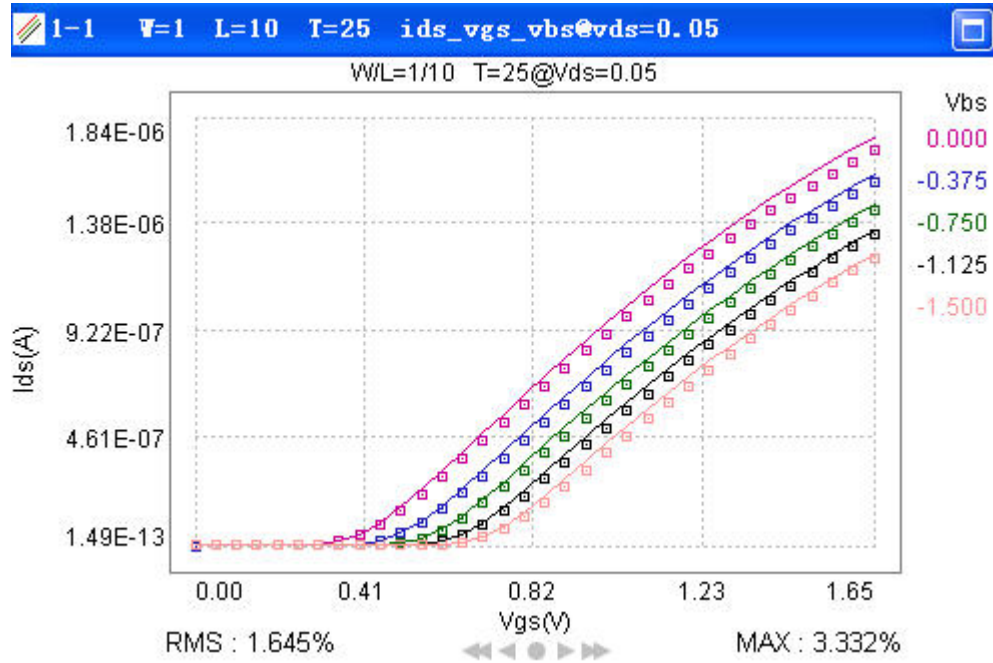
Map Selected Region

If you want to perform optimization to one group of devices with the same optimization region, you can either do it one by one manually or right click on the plot and select Map Selected Region > All, as shown in following figure. This operation marks the selected region of current device duplicate to all the devices selected in the Device Navigator.

Map Region Selection
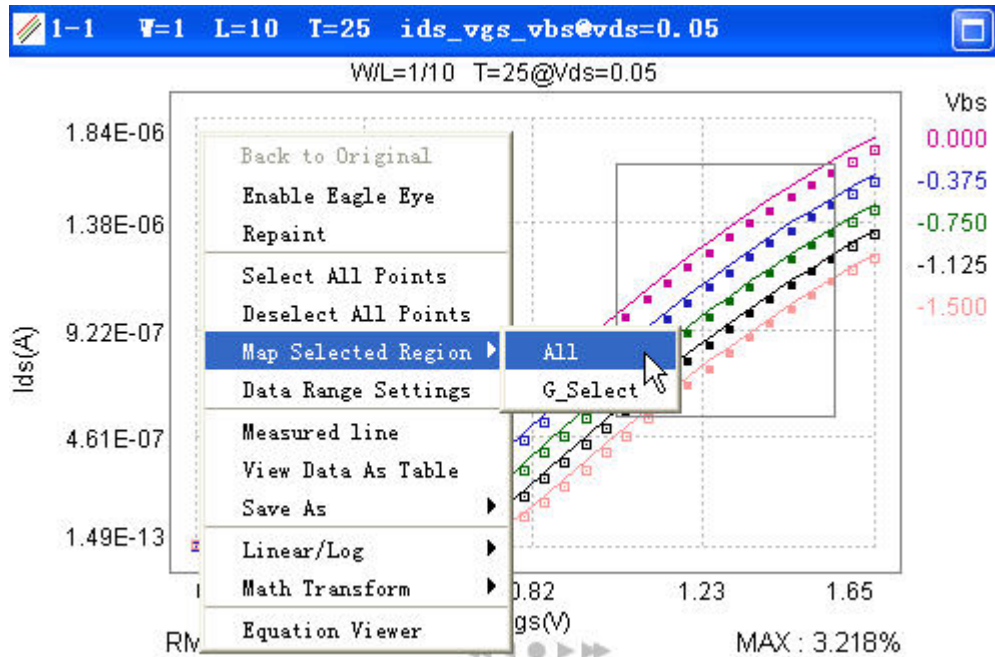
The Map Selected Region is to map the relative location of selected region to all other plots, but not to map the absolute XY coordinates, as illustrated in following figure. There are two options for mapping: All maps the region to all selected devices in the Device Navigator; and G_Select option allows you to specify the target devices.

Obtained regions after map region selection
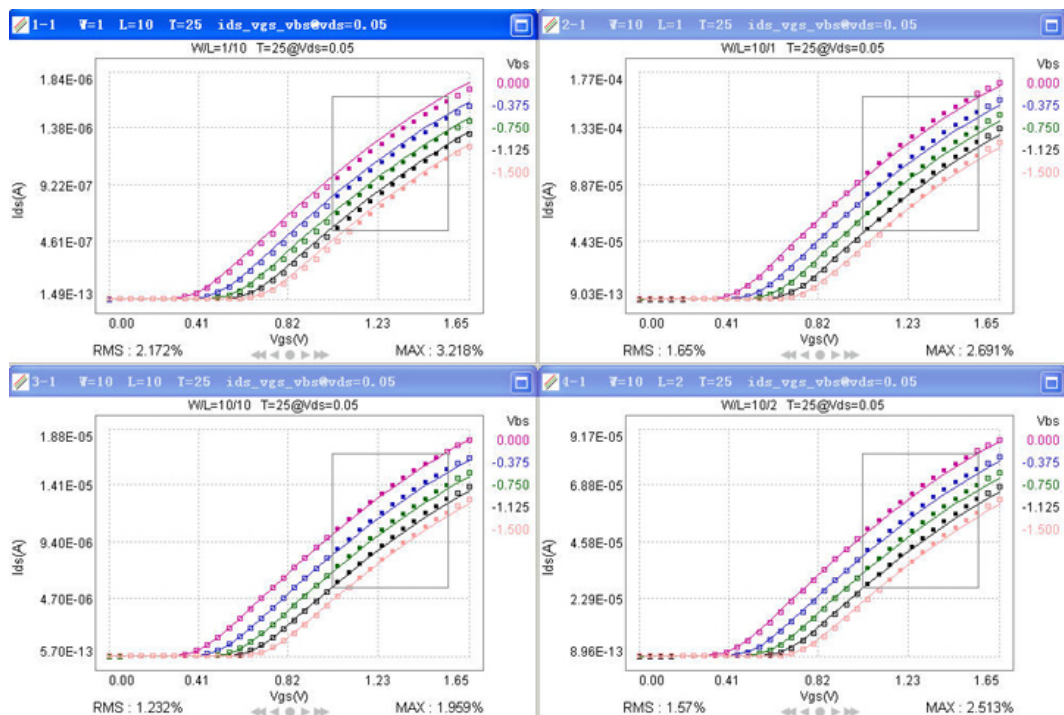
Select Multiple Region

MBP also allows selection of multiple regions on the same plot. Turn on this option by clicking Optimization > Data Selection > Multiple from the main menu or

Multiple Region icon [icon] in the tool bar. Then, you can select multiple regions on the same plot as shown in Figure: Multiple Region Selection.

> **NOTE** If you intend to choose a new region on another plot, the previous region is kept and no need to check **Multiple Region**.

Multiple Region Selection
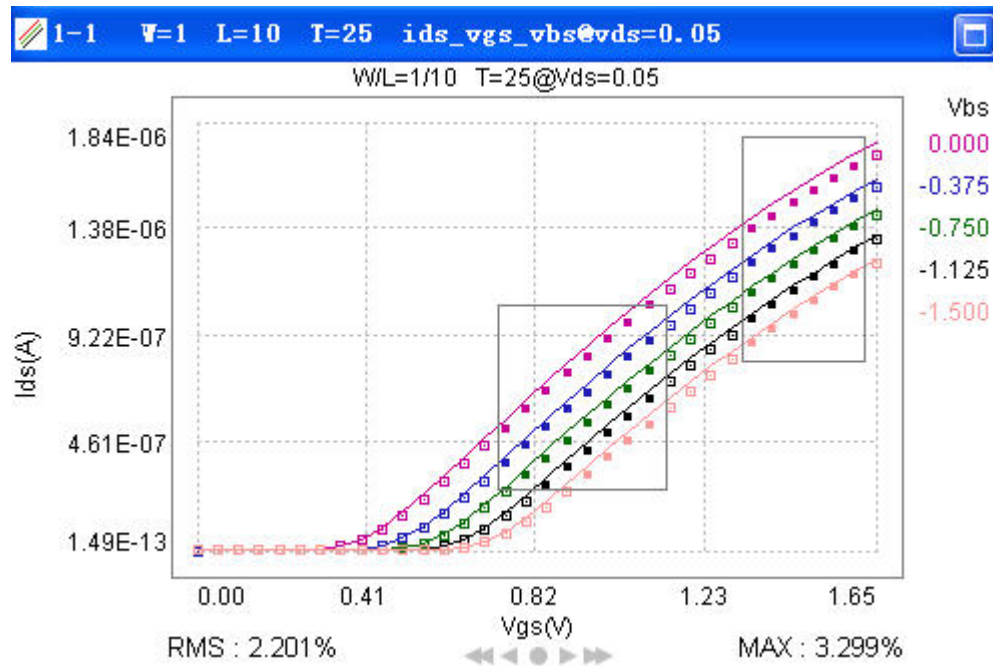
Deselect Optimization Region

To deselect the optimization region on the current plot, right click the plot and select Deselect All Points. To deselect the optimization region on all the plots, choose Optimization > Data Selection > Clear All Pages from the main menu or

click Clear All Pages icon [icon] in the tool bar.

> **NOTE** Make sure to clear previously selected regions before selecting new region. If any unexpected regions are included, the result of automatic optimization may be far beyond expectation.

Optimization Curve Selection

By default, all the curves within the selected region is included in the auto optimization. However, in some cases, only some curves are fitting targets. For example, to optimize MOS parameter VTH0 at initial steps, modeling engineer may

be only interested in the fitting results of linear gummel curve at Vbs=0. For that purpose, double-click those non-zero Vbs values(-0.375, -0.75, -1.125 and -1.5). Then the corresponding curves turn grey and the points is excluded in the auto optimization, as shown in following figure. Double-click again to restore these curves to the initial status.

Select single curve for VTH0 optimization
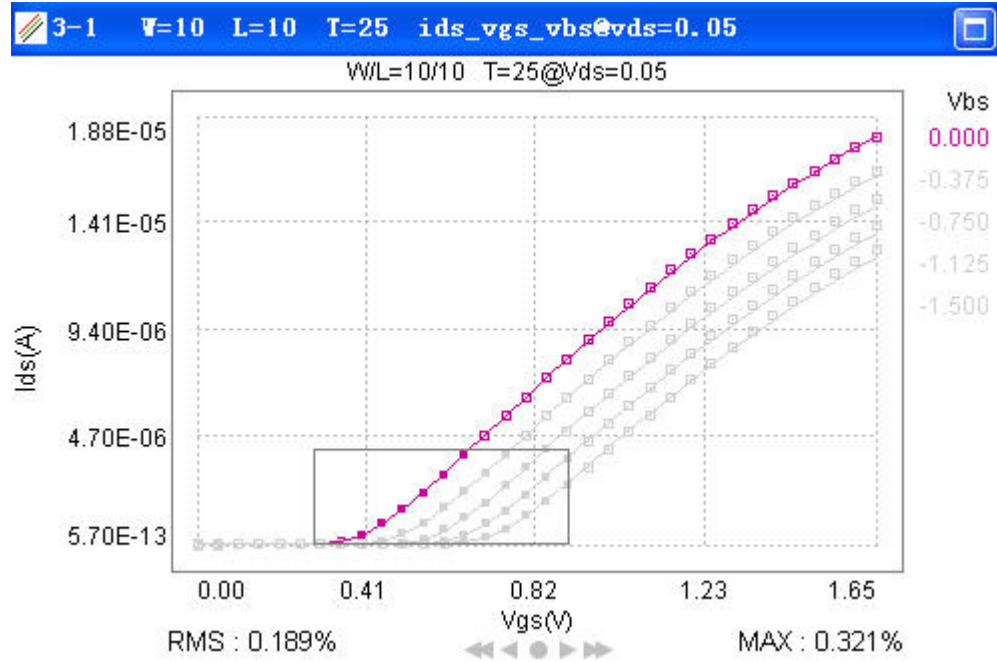
Running Optimization

After selecting the region and the curves to optimize, you can click Optimize icon ▶ in the parameter window to start the auto optimization. After a while, you can see the results of optimization.

> **NOTE** Selection of proper parameters for the optimization region and setting proper parameter values and boundaries is necessary, otherwise the results might be bad. Besides auto optimization, you can also perform optimization manually as mentioned in the previous section.

The optimization procedure is to minimize the RMS of the selected data points. Thus, RMS settings affect the final optimization results. As shown in Figure: Influence of RMS settings on optimization, all the capacitance values are less than 3E-11. If the resolution of RMS is set as 1.0E-9, zero RMS all over the plot is displayed. Under such situation, even if you perform the optimization with its optimizer, nothing happens.

> **NOTE** During optimization, parameter value does not exceed boundaries.

Influence of RMS settings on optimization
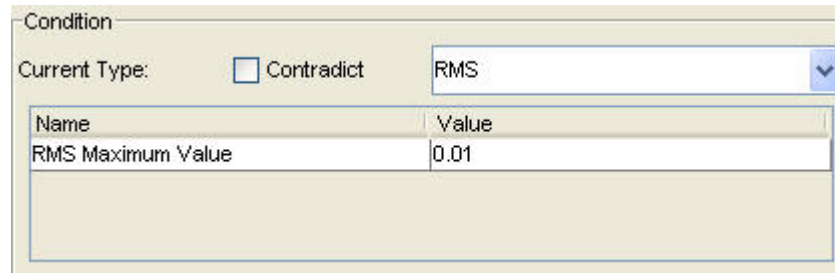
Running Optimization

After selecting the region and the curves to optimize, you can click Optimize icon
in the parameter window to start the auto optimization. After a while, you can
see the results of optimization. Note: Selection of proper parameters for the
optimization region and setting proper parameter values and boundaries is
necessary, otherwise the results might be bad. Besides auto optimization, you can
also perform optimization manually as mentioned in the previous section.

Checking points fitting error

You can check the error between the measurement and simulation for different

points. To realize this, click the Pin a Data Reader icon in the tool bar. Then
click the data point interested, the difference value is displayed on the graph in real
time, as shown in following figure. To clear one pin, click the corresponding data
point again. If you want to clear all the pins, right-click on the graph and choose
Clear all pins.

Adding pins to the points in one graph
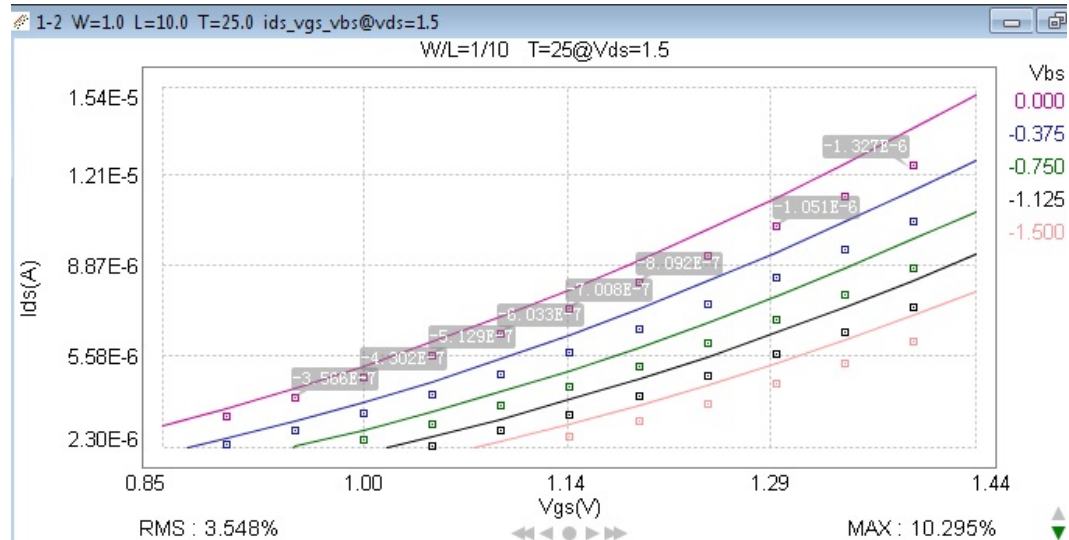
In the example above, this difference is expressed as M-S (Measurement data minus simulation value). MBP provides three expressions (Show RMS, Show M-S, and Show RMS and M-S) to choose from. Choose * Tool > Preferences* from the main menu, then click Pinner Options, the window is shown as following figure.

Pinner options
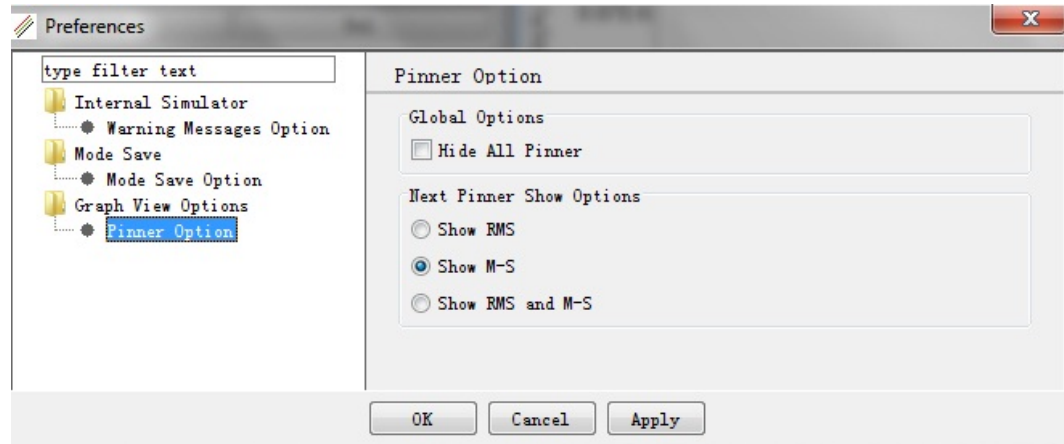
Optimization Summary

After the optimization, you can review the summary by choosing Optimization > Summary from the main menu, as shown in following figure. You can find the change of the error and the selected parameters before and after the optimization.

Optimization summary
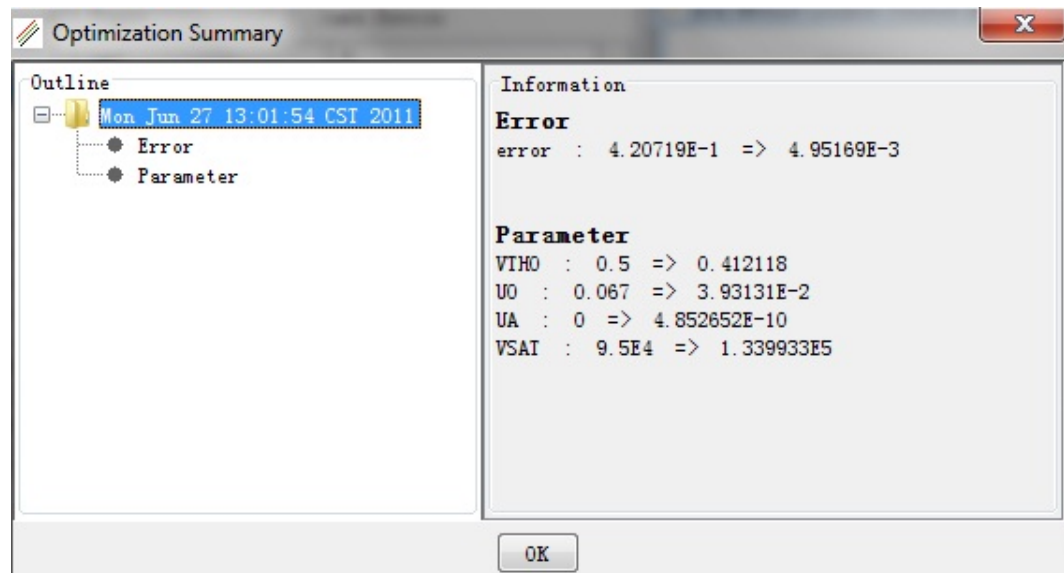
Optimization Utilities and Options

MBP also provides options and some handy features when running optimization.

Choose Optimizer

MBP provides different optimizers to choose from. Click Optimization > Choose Optimizer from the main menu, the window pops up as shown in following figure. The default optimizer is condor. You can choose other optimizer (such as GA, PSO...) from the drop-down list.

Choose optimizer
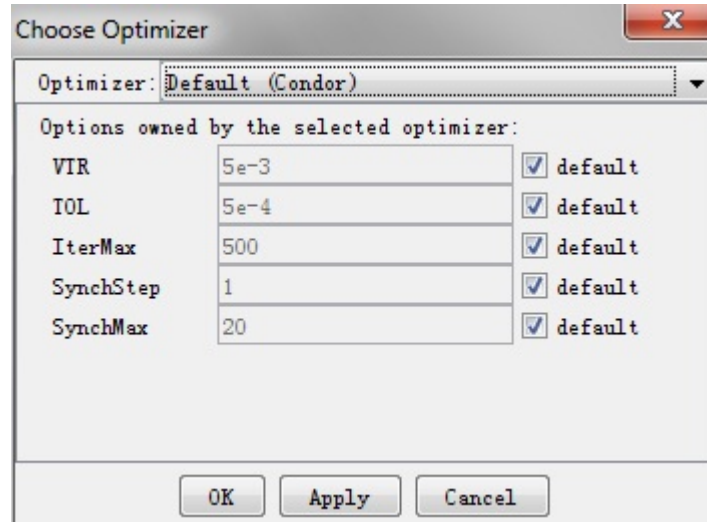
Optimization Options

- Step and Boundary Settings: Click Extraction > Options > Optimization Options from the main menu to pop up the Optimization Options window as shown in Figure: Optimization options. Set the default step to One Order Lower or Two Order Lower. For example, let's suppose vth0 is 0.7V. Then, after loading the model, for one order lower, the step is 1E-2 V; for two order lower, the step is 1E-3 V.

Checking Auto Step Tune disables above two options. Then by default, the step is always 1/20 of its boundary length (boundary length is defined as the difference between upper boundary and lower boundary.).

To use mouse wheel, determine the step in Mouse Wheel Divider. You can set the percentage of the whole boundary difference. On rolling the mouse wheel, the parameter value changes accordingly.For example, if its boundary is [0,100], the value here is set to 20 (percent). So, every time you roll the wheel, the value increases/decreases by 20. And once the cursor reaches upper or lower boundary, you can enable automatic boundary expansion by continuous rolling, if the box* Shift boundary when adjusting boundary with mouse wheel* is checked.

You can either expand the boundary on both directions (by checking the Bi-direction box) or make the optimizer to expand intelligently (by checking the Intellectual box). For example, if the value is in the upper 30% of the boundary range, the upper boundary is expanded. The lower one is expanded if it is in the lower 30%. And if the value meets both requirements, both boundaries are expanded.

Load & reset is for soft boundary control. Once a model is loaded into MBP, MBP sets parameter soft boundary according to the digit input in the Init ratio. For example, if you specify 50.0(%), then soft boundary is +/-50.0% of its absolute value. Besides, this setting is applicable to Reset parameter boundary and step icon in the optimization window.
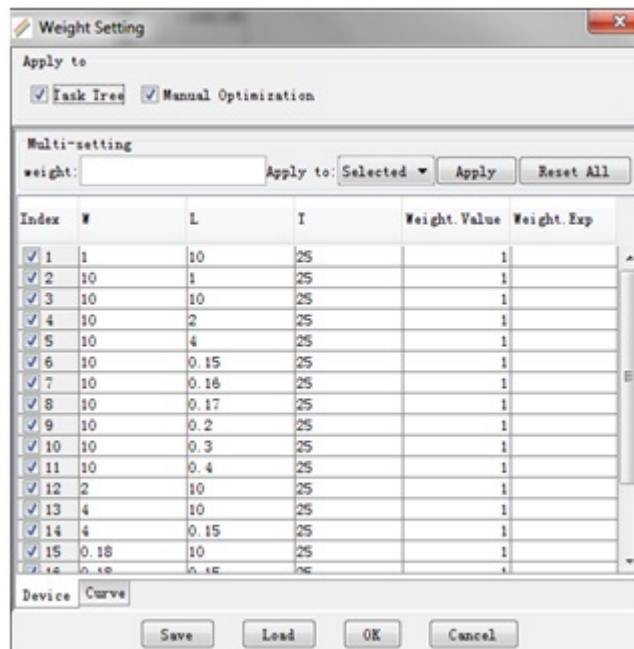
Optimization options
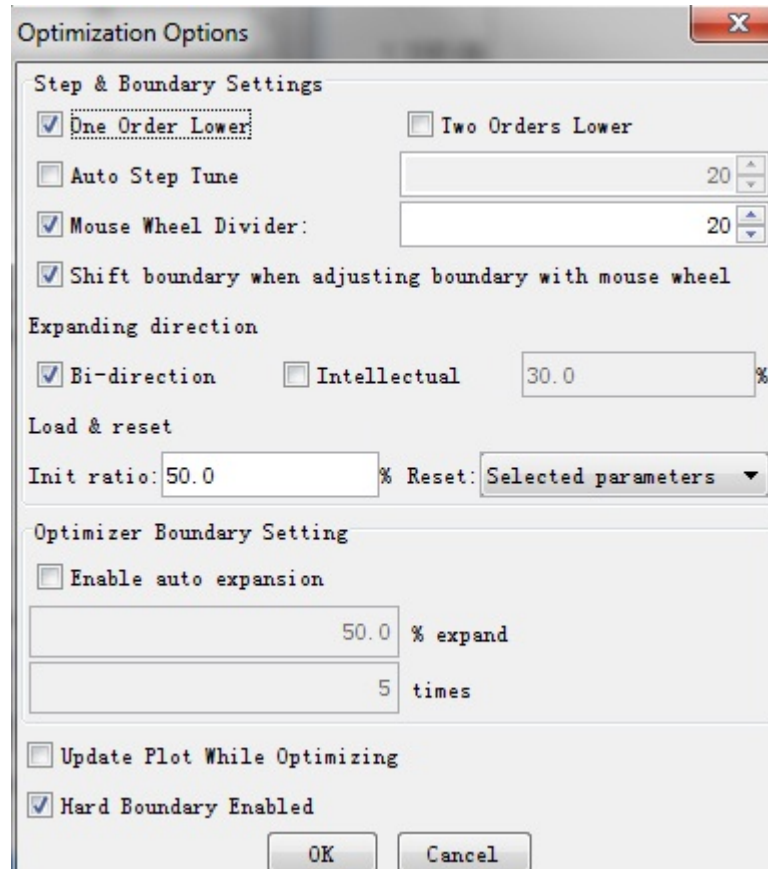
- Optimizer Boundary Setting: When using its internal optimizer for tuning, any parameter is not allowed to exceed its soft boundary. However, MBP allows to auto expansion of the soft boundary during this procedure. To realize it, first check Enable auto expansion box, then input the percent and the times for expansion. To date, soft boundary is always within the range of hard boundary.

- Other Settings
  By checking Update Plot While Optimizing, you can observe the simulation plots changing with the parameters during the whole optimization process. By default, the Hard Boundary Enabled option is checked to enable the hard boundary during the optimization. You can cancel the limitation by uncheck it.

Weight Setting

MBP also allows you to set different weights for different devices when running the optimization. The weight here 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 (remaining unchanged). Since the build-in optimizer implements optimization according to the RMS value, it treats all devices the same opportunity. However, if you set the weight of one device as 2, the RMS value of this device is multiplied by 2. Thus, optimizer treats it much more important than other ordinary devices with weight of 1.

To use this feature, click Extraction > Weight Setting from the main menu. The setting dialog pops up as shown in following figure.

Weight setting

You can refer to the application note Optimization Weight Setting for more details.

Spice Options

MBP supports simulation option of GminDc, Scale and GeoShrink as shown in Figure: Spice options. GminDc is a conductance that is placed in parallel with all pn junctions and all MOSFET nodes for DC analysis. Scale is element value multiplier. And GeoShrink is geometry shrink factor as a global model to apply to all elements. Check TempAsOption option to enable Temp as the option.

You can also append simulator compatible options into MBP, even those options are not supported by its internal engine. For instance, if you want to put .options EXPLI=1e-4 to external HSPICE. Type EXPLI=1e-4 to the line under User Option(s) and press OK button. To keep netlist generated by MBP for external simulator, choose Simulation > External SPICE Options and uncheck Is Delete Netlist Files box.

Check the isUserOriginalFile option if the original model file is used directly when calling external SPICE. You can input the model name (including the path) or some options and global parameters in the blanket below.

Spice options
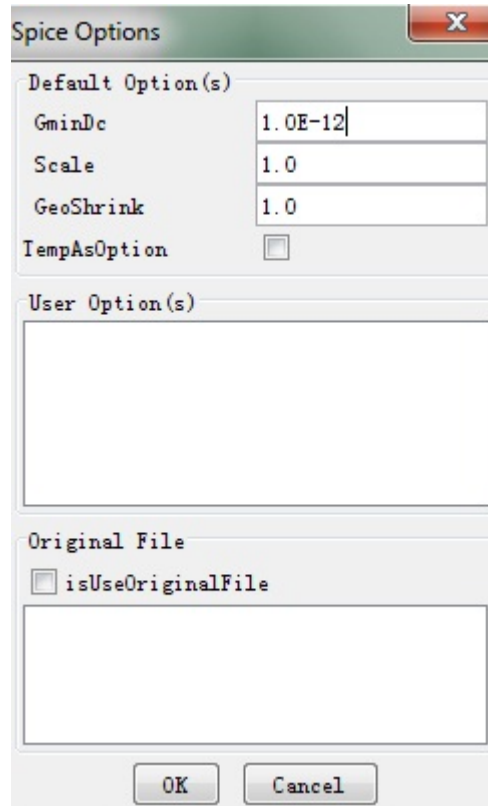
Set Default

Clicking Model > Set default to set all model parameters, except process parameters, to default value. And this function is only valid to MOS models (BSIM3v3, BSIM4, BSIMPD and etc).

Reload Hard Boundary

The hard boundary can be reloaded by clicking Model > Reload Hard Boundary from the main menu. See MBP Files for the details about the model setting files which define the default value and the hard boundary.

Undo and Redo

This feature allows you to undo or redo the optimization operations. The two block arrows on the upper right corner of the parameter window are for this purpose. As shown below in following figure, the left one is for Undo and the right one is for Redo operation.

Undo and Redo
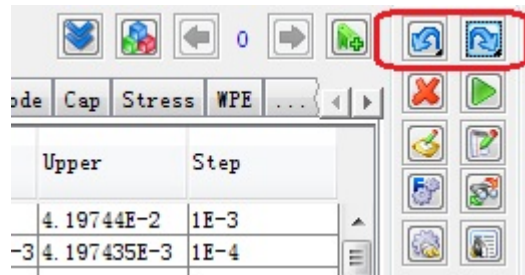
You can also click and hold the button to activate the window as shown in following figure. It records the operation history up to 100 steps. You can choose to undo /redo multiple operations at one time.

Undo multiple operations
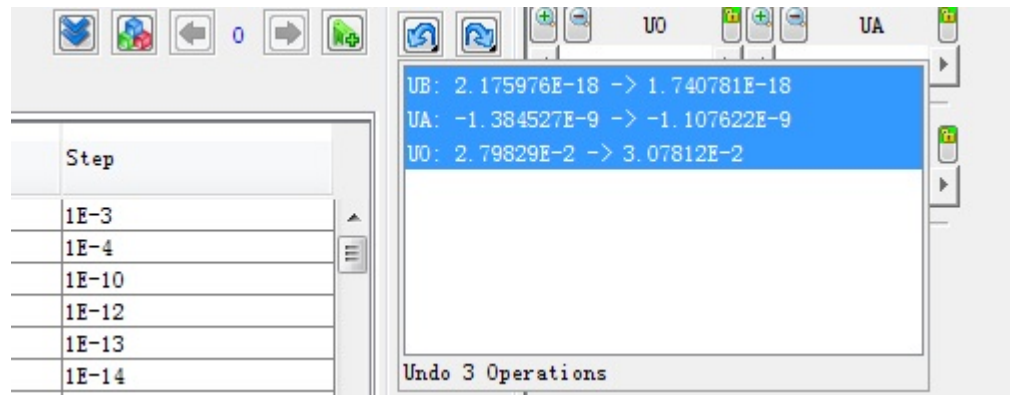
Mark Model Function

MBP provides another method to switch between different models without saving (loading) model files to (from) hard disk.

When tuning model, you can mark current model by clicking the Mark icon . The mark label 1 would be assigned to the old model and the label jumps to 2. During subsequent tuning process, you can also mark other models in a similar method and switch between marked models at any time by clicking the left/right arrow buttons, as shown in following figure.

Mark and switch the model
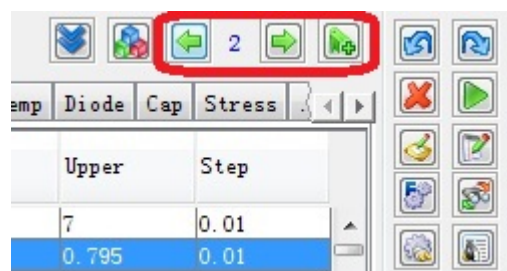
To clear all the marks, right-click on the mark label and choose Clear Marks, shown in following figure.

Clear marks
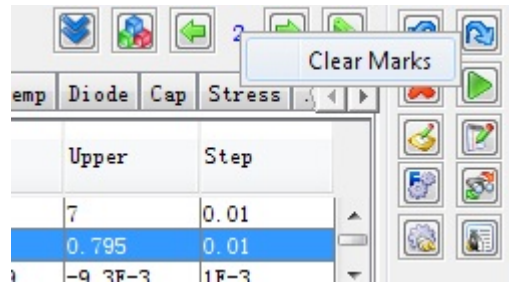
	NOTE	Marked models are saved to RAM instead of hard disk. To create/load a new project or exit MBP loses all its information about the marked models.

Minimize the parameter window during optimization

In floating optimization window mode, double-click on the Optimization title bar to minimize the window. The minimized parameter window is shown in following figure. You can restore the parameter window by the same operation.

Minimized Parameter Window
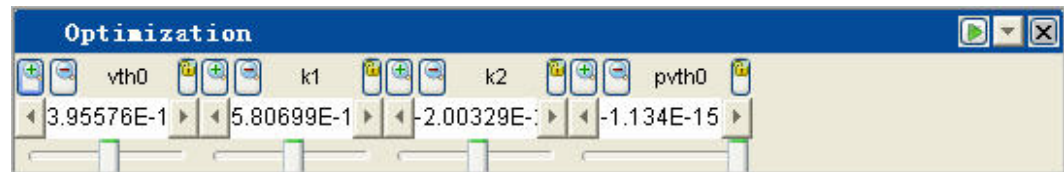
In fixed optimization window mode, you can minimize the window by clicking the Hide icon ⬇ . In the minimized window, click the Show icon ⬆ to restore the normal view. As shown in following figure, you can perform search, optimize, mark, and undo/redo function in the minimized window.

Minimized fixed mode optimization window

Data Display

Data generated after simulation and measurement can be viewed using plots, tables, graphics or text files, and other options like Data Reader.

- Data Reader
- Layout Configuration
- Pin Function
- Plot Details
- Plot Operations and Options
- Window Setting

Data Reader

To determine the XY coordinates of a data point on a plot, click the Data Reader

icon ![icon] on the toolbar. This action opens the Data Display window. A red circle auto-tracks the nearest data point. The displayed value is for measurement Y value, Simulation Y value, P value and X value, respectively. Also, by holding the Alt key and dragging the cursor, you can find the slope between two points for both measurement and simulation. See following figure for an example.

Data reader function
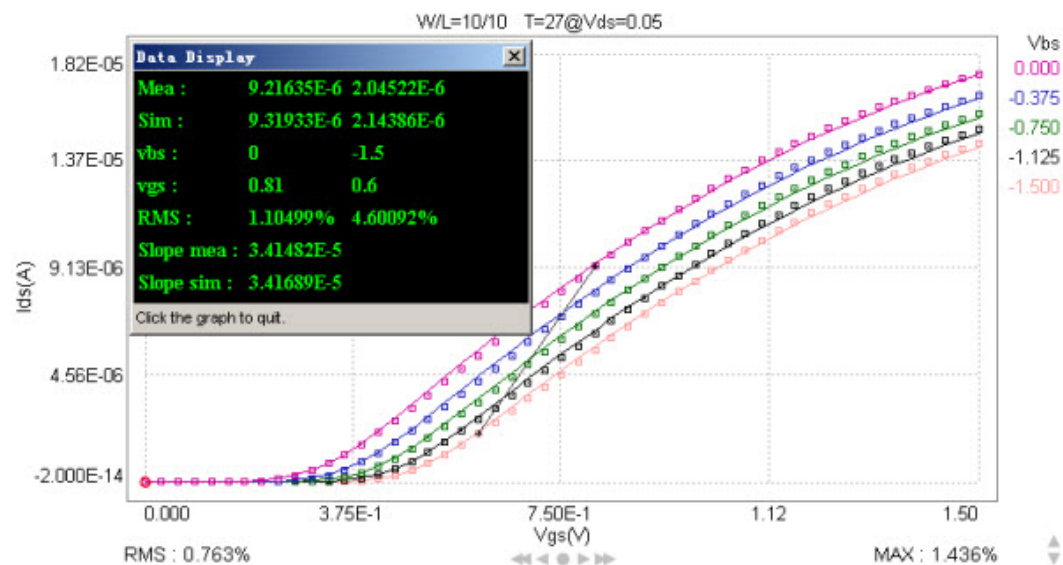
Layout Configuration

Using Layout tools, you can configure the way device pages are displayed.

By selecting Tool > Layout as shown in following figure, you can configure how device pages are displayed. For example, if you want to view more plots, select 2x3, 3x3 or more from Graph Panel Layout. Layout Config is often referred together with Device Navigator. For more information, see Device Navigation.

Graph selection window
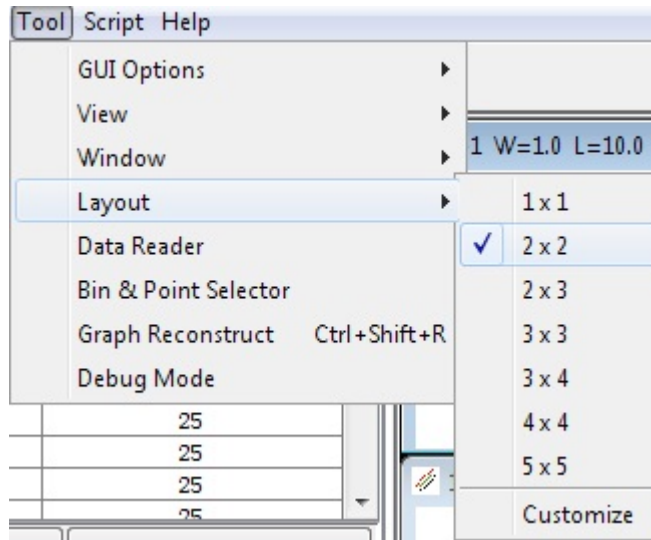
RMS Options

Root Mean Square error calculation is a popular method of assessing the fitting between the simulation and measurement. The RMS function used in MBP is listed below:

$$RMS = \sqrt{\frac{1}{N}\sum_{i=1}^{N}\left(\frac{Im-Is}{max(abs(Im),\,abs(Is))}\right)^2}$$

Im stands for measurement value and Is stands for simulation value. There are two RMS options, resolution and cutoff. Resolution stands for the minimum values that are included for RMS calculation, in other words, all the points below resolution are excluded from RMS calculation. Cutoff is the threshold that for RMS calculation below Cutoff, the RMS function becomes the following:

$$RMS = \sqrt{\frac{1}{N}\sum_{i=1}^{N}\left(\frac{Im-Is}{max(Cutoff,\,abs(Is))}\right)^2}$$

The value set in Cutoff should be surely larger than that in Resolution, otherwise it does not take any actual effect. MBP also provides another RMS method corresponding to "MAX Y" in RMS options, it uses the following equation:

$$RMS = \sqrt{\frac{1}{N}\sum_{i=1}^{N}\left(\frac{Im-Is}{max(abs(Imax),\,abs(Is))}\right)^2}$$

Using MBP

Imax is the maximum measurement value on the plot. To set RMS options, click Tool > GUI Option > RMS from the main menu, the RMS setting window pops up as shown in following figure. Set the values for resolution and cutoff. If you want to make RMS invisible on the plot, just check the option of Hide RMS and MAX.
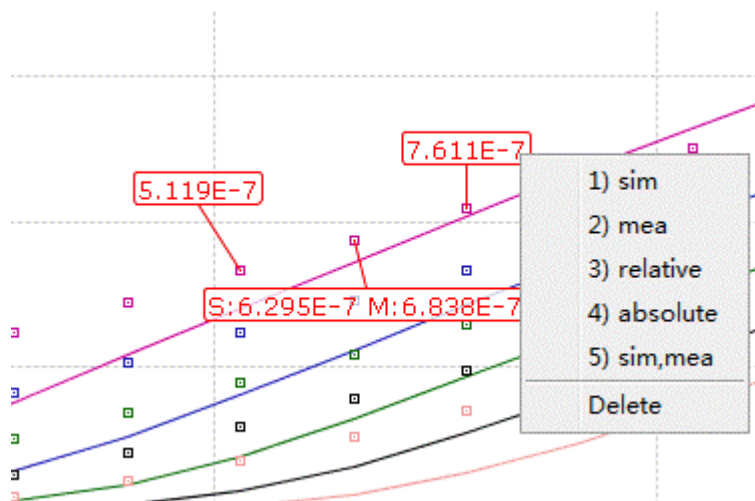
RMS setting window
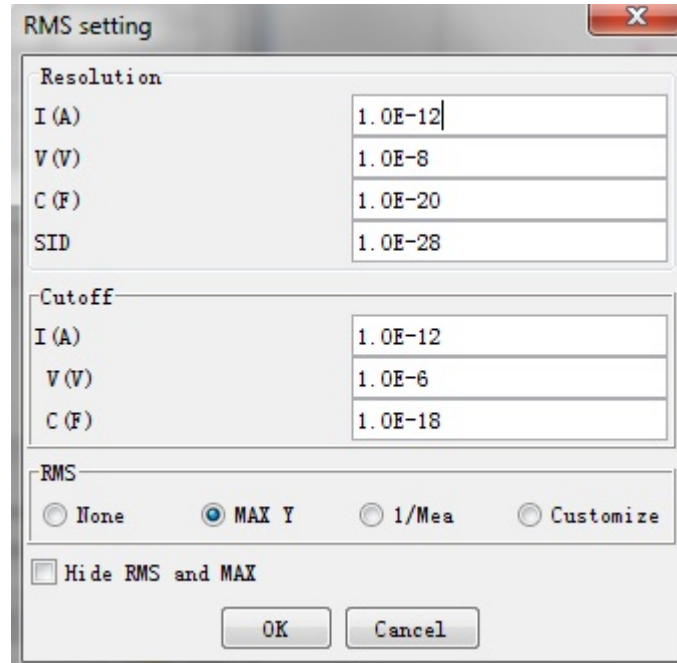
Pin Function

The button in the main manual is used for Pin the value of the points in the plot, When user save the project the Pin information will be saved also.

Pin function

Use can drag the pin on GUI and right-click to switch or delete pin information

User can customized pin contents and simple math expression(*Tool – Preferences – Graph View Options – Pin options*)

- User is enabled to define Pin display order and content
- Support advanced definition with simple math expression
- Pin display style can show/hide border
- Can preview pin display style
- User can export and Import Pin configuration file
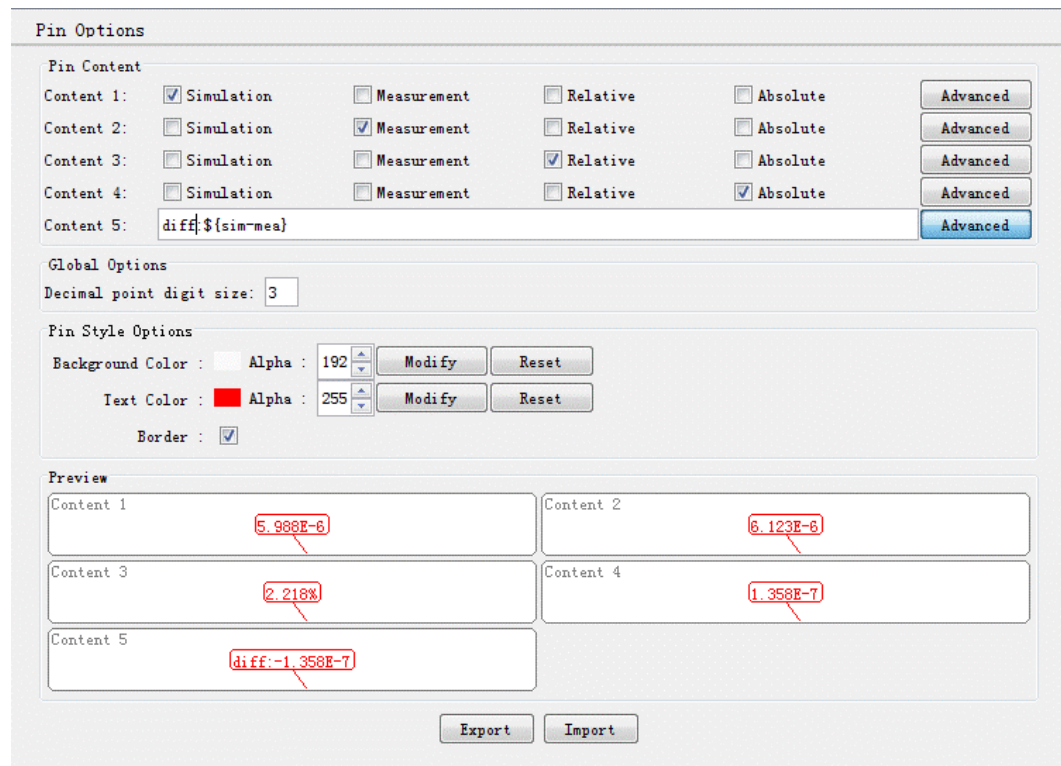
Plot Details

Plot option window

The scale options are defined in following table.

Table: Scale options

|	Mode	Default Settings
Min X	By value	Minimum X
Max X(%)	By scaling	100% of maximum X
Min Y	By value	Minimum Y
Max Y(%)	By scaling	100% of maximum Y

By value: To specify a digit value

By scaling: To set the size a percentage of the maximum measured or simulated data points

You can also define line width(default is 1.0), display precision(default is 3) and Symbol Size(default is 4) in this window.

View

View option
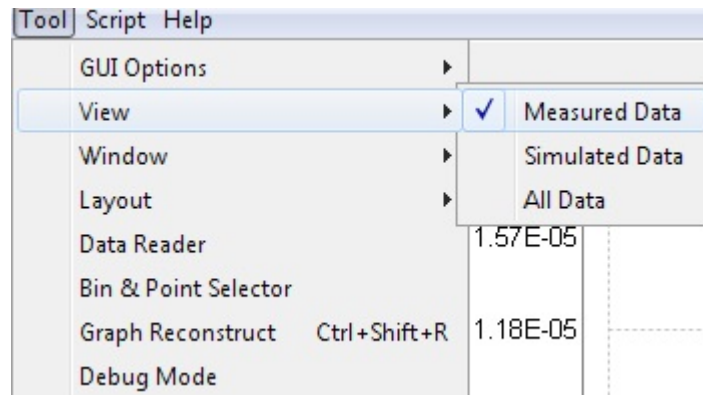
Plot Operations and Options

MBP is quite flexible on plotting. You can zoom in/out on the plot, set data range, change scale, apply math functions on the existing plots and export plots as pictures. The features are described as follows.

Zoom In & Zoom Out

To zoom in, select the region to be zoomed by using cursor (left–click and hold). To return to the original scale, double-click or right-click the plot and select Back to Original.

Eagle eye

If this option is checked (right-click on the plot to show Enable Eagle Eye), once you perform zoom in on a page, one small window automatically pops up at the right bottom corner of current page, as shown in following figure. The pop-up window shows the full image of original page. And the red rectangle selected region has been zoomed in. You can also drag and resize the red rectangle to focus on whatever region for zooming in.

Zoom in with eagle eye option
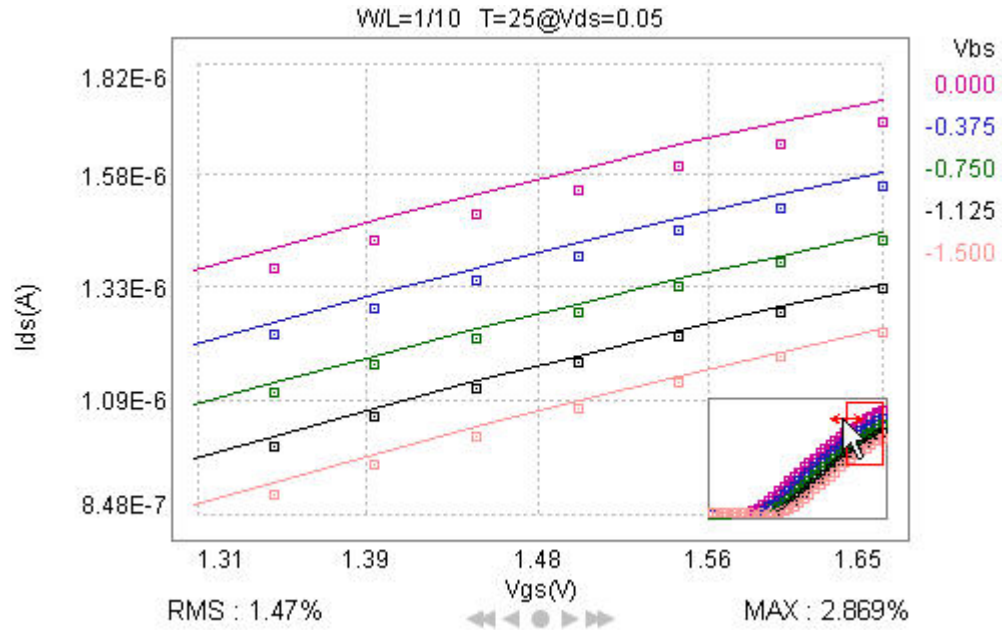
Repaint

To refresh the plot.

Set Data Range

Right-click the plot and select Data Range Settings. Uncheck Default Data Range and the Set Default option for certain scale options. Enter boundary value you want to view with, as shown in following figure. User-defined range could be either within or out of real data range. Each Math Transform (to be introduced later in the section) has its own data range. For instance, if you have defined a range on the plot with y math function, then on applying dx/dy to the plot, this data range set does not take effect, unless returning to y. However, user-defined data range is kept with scale change. In addition, you can apply this data range to all plots that with the same page index in current project by checking the option of Map The Data Range.

Data range settings window
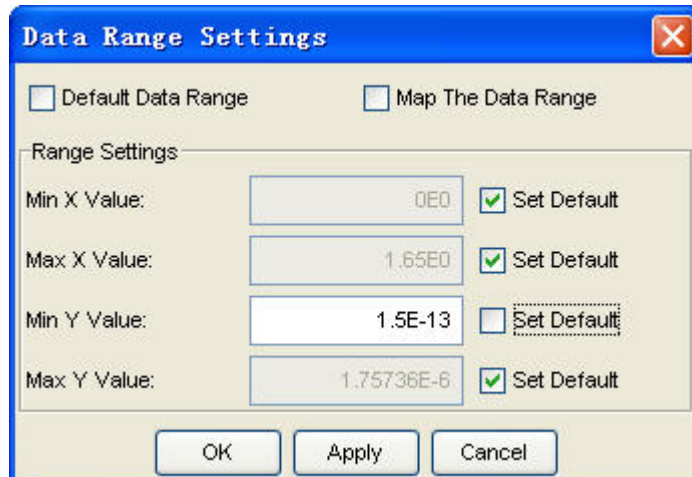
Change Scale

Right-click the plot and select Linear/Log, you have an option of linear and log scale for both X and Y axis. You can also double-click the values of X (or Y) axis to switch the linear and log scale of the axis.

View Data as Table

Right-click the plot and select View Data As Table, the table that contains both measurement and simulation data points pops up. The error between measurement and simulation is also attached behind, as shown in following figure.

Sample data table

Using MBP

Once you apply math function like `dx/dy` on plot, MBP prints out transformed data in the table. However, data shown in an opened table refreshes automatically with the tuning model parameters. Close the current and open a new one to see the change.

Save Plot as Graphic or Text Files

Right-click on the plot and select Save as, you can either save it as a graph (gif) or text file.

Math Transform

MBP allows you to apply math functions to the plot including plotting y (by default), 1/y, y', y'', y''', dx/dy, etc. Right-click on the plot and select Math Transform, then choose the math equations you want to apply. The following figure shows an example of plot after `y'` transform.

y Math transform with x-lin, y-log scale
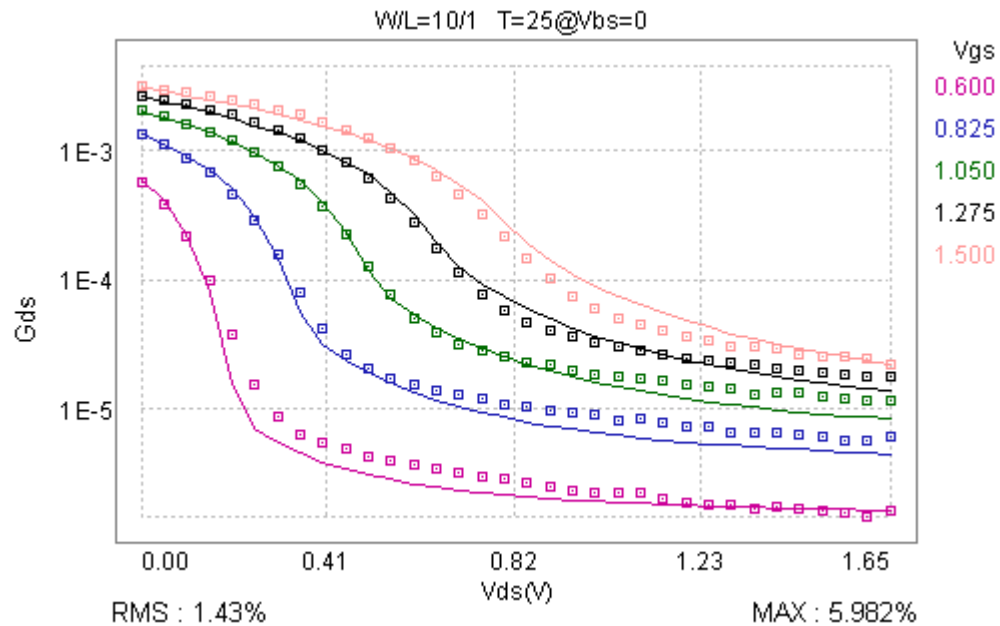
In addition to existing math transform functions, you can also customize math transform functions with some simple programming. Right-click on the plot and select Math Transform > Customize, the Math Transform Manager window pops up as shown in following figure. Right-clicking the plot next time, the appended math function is also in the Math Transform list.
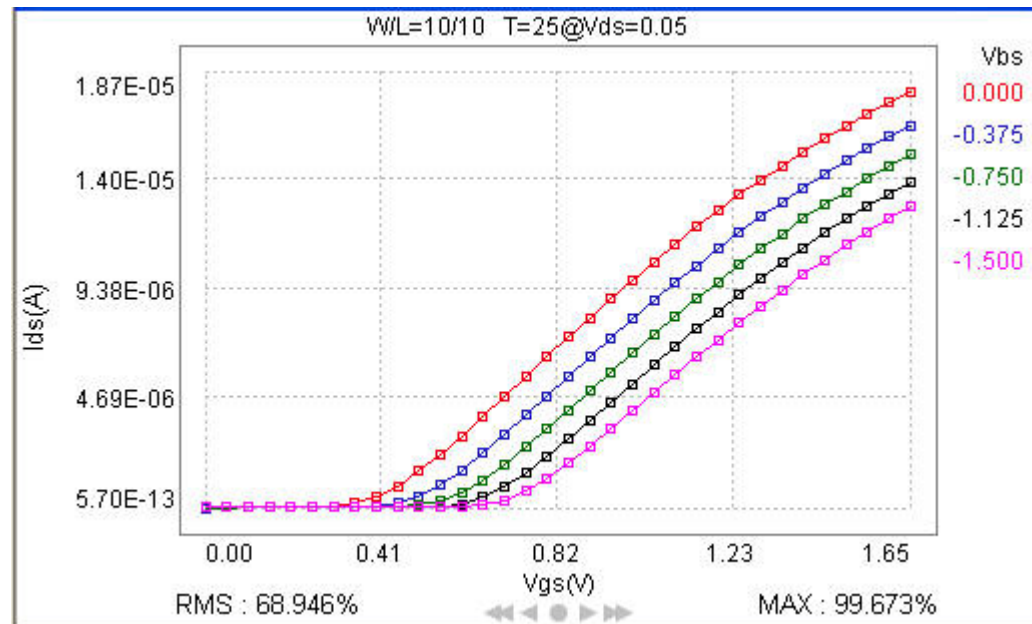
Math transform manager
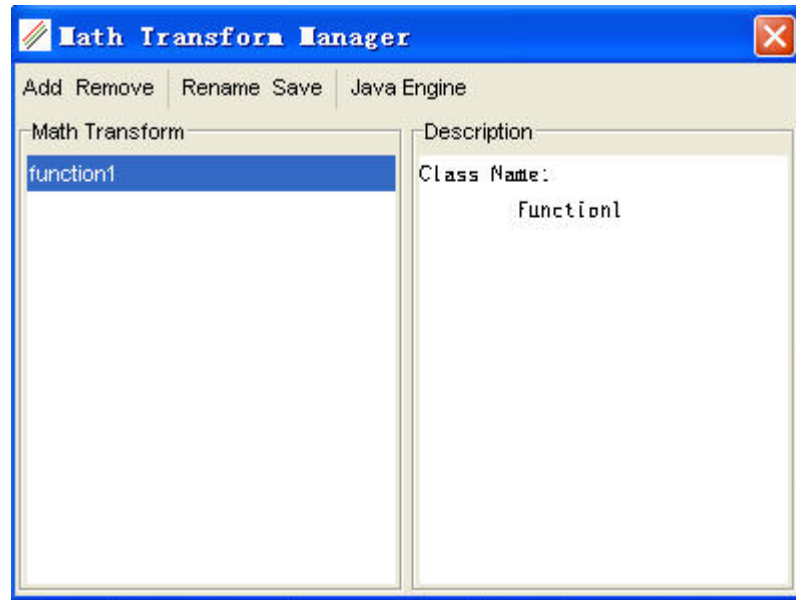
Measured Line

You may like to view measurement data only under certain circumstance such as checking the measurement. By checking Measured line, the plot only shows measurement data, in line plus symbol style, shown in following figure.

Example plot with measured line checked

Extend Simulation

You can also observe the simulation result without the limitation of measurement data. Currently, MBP supports both X axis and P axis direction extension. The setting and result windows are shown in Figure: Extend simulation window and Figure: Extend simulation result, respectively.
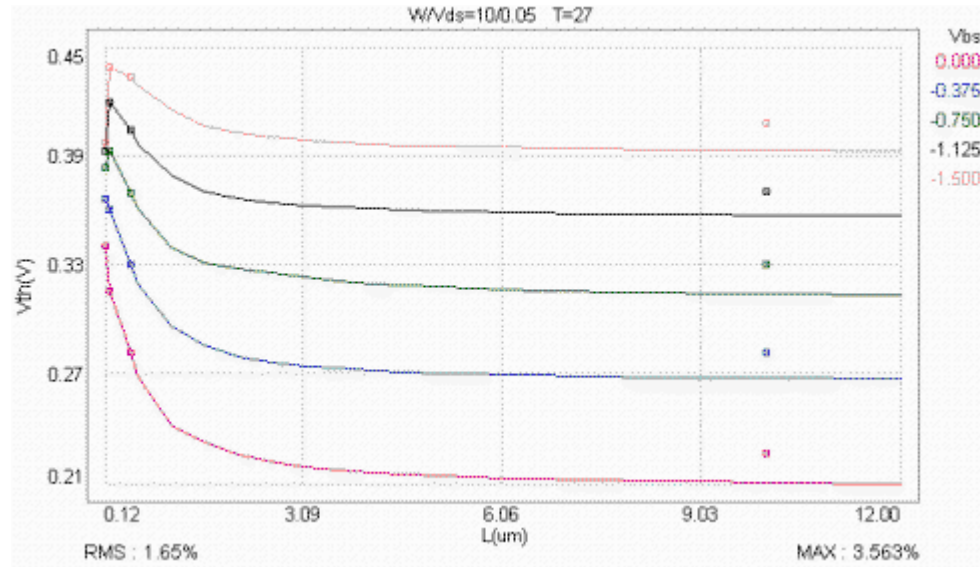
Extend simulation window

Extend simulation result
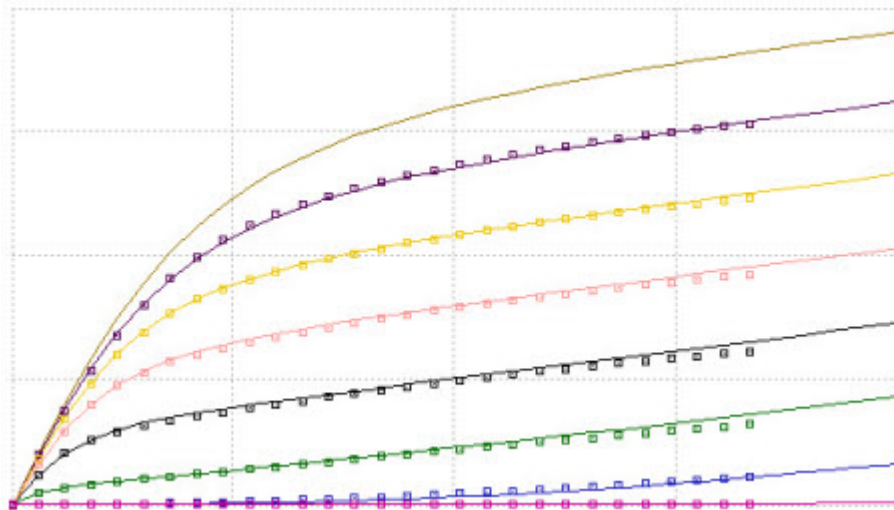
Besides IV curves, Extend Simulation also allows you to deal with IMV plots as shown in following figure. You can extend simulation points out of measured points or between measurement points.
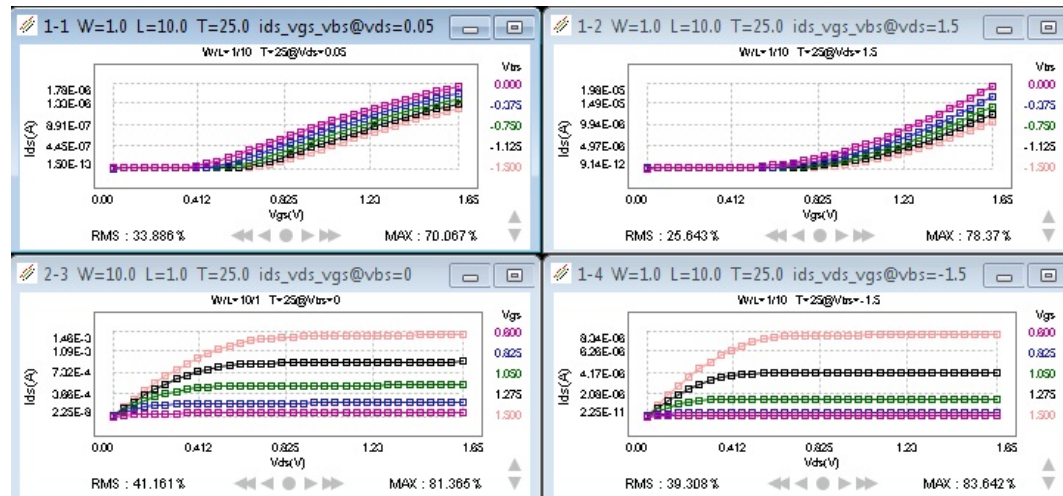
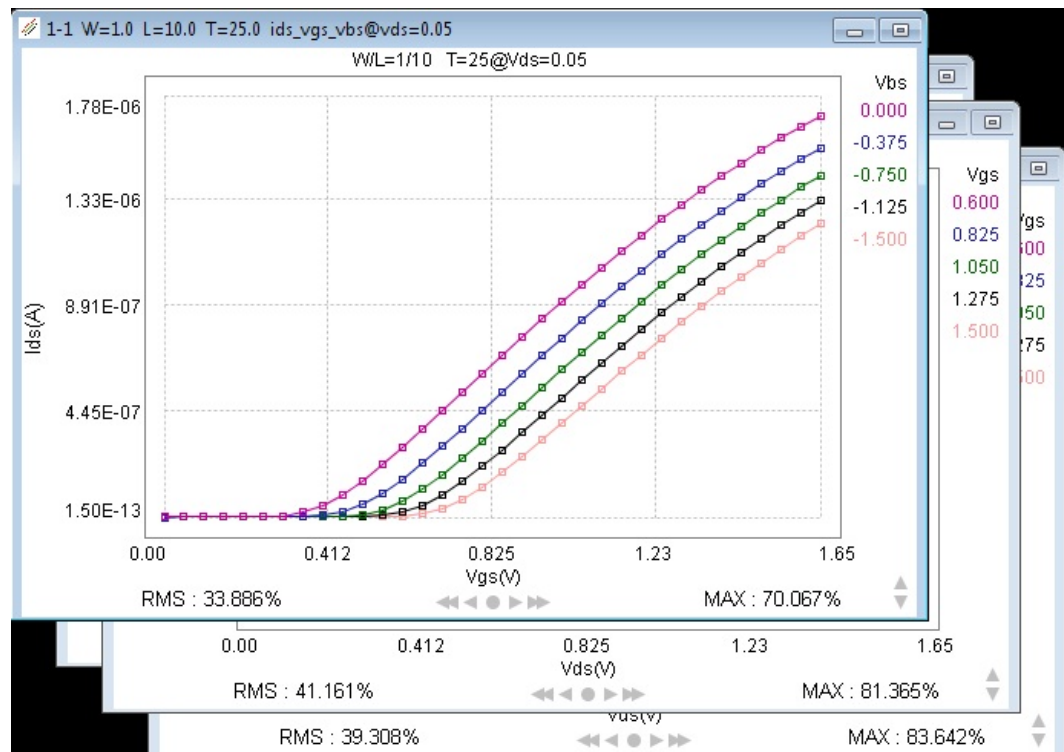Extend simulation result on IMV pages

Window Setting

This option allows you to arrange the plot panel either in Tile or Cascade mode as below shown in Figure: Plot panel in tile mode and Figure: Plot panel in cascade mode.

Plot panel in tile mode

Plot panel in cascade mode

1-1 W=1.0 L=10.0 T=25.0 ids_vgs_vbs@vds=0.05

WL=1/10 T=25@Vds=0.05

Vbs	
0.000	
-0.375	
-0.750	
-1.125	
-1.500	

Vgs	
0.600	
0.825	
1.050	
1.275	
1.500	

Ids(A)

1.78E-06
1.33E-06
8.91E-07
4.45E-07
1.50E-13

0.00   0.412   0.825   1.23   1.65
Vgs(V)

RMS : 33.886%   MAX : 70.067%

0.00   0.412   0.825   1.23   1.65
Vds(V)

RMS : 41.161%   MAX : 81.365%

Vds(V)

RMS : 39.308%   MAX : 83.642%

KEYSIGHT
TECHNOLOGIES