



HypScript Automation

HyperLynx VX.2.5

October, 2018

© 2013 - 2018 Mentor Graphics Corporation
All rights reserved.

This document contains information that is proprietary to Mentor Graphics Corporation. The original recipient of this document may duplicate this document in whole or in part for internal business purposes only, provided that this entire notice appears in all copies. In duplicating any part of this document, the recipient agrees to make every reasonable effort to prevent the unauthorized use and distribution of the proprietary information.

This document is for information and instruction purposes. Mentor Graphics reserves the right to make changes in specifications and other information contained in this publication without prior notice, and the reader should, in all cases, consult Mentor Graphics to determine whether any changes have been made.

The terms and conditions governing the sale and licensing of Mentor Graphics products are set forth in written agreements between Mentor Graphics and its customers. No representation or other affirmation of fact contained in this publication shall be deemed to be a warranty or give rise to any liability of Mentor Graphics whatsoever.

MENTOR GRAPHICS MAKES NO WARRANTY OF ANY KIND WITH REGARD TO THIS MATERIAL INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.

MENTOR GRAPHICS SHALL NOT BE LIABLE FOR ANY INCIDENTAL, INDIRECT, SPECIAL, OR CONSEQUENTIAL DAMAGES WHATSOEVER (INCLUDING BUT NOT LIMITED TO LOST PROFITS) ARISING OUT OF OR RELATED TO THIS PUBLICATION OR THE INFORMATION CONTAINED IN IT, EVEN IF MENTOR GRAPHICS HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

U.S. GOVERNMENT LICENSE RIGHTS: The software and documentation were developed entirely at private expense and are commercial computer software and commercial computer software documentation within the meaning of the applicable acquisition regulations. Accordingly, pursuant to FAR 48 CFR 12.212 and DFARS 48 CFR 227.7202, use, duplication and disclosure by or for the U.S. Government or a U.S. Government subcontractor is subject solely to the terms and conditions set forth in the license agreement provided with the software, except for provisions which are contrary to applicable mandatory federal laws.

TRADEMARKS: The trademarks, logos and service marks ("Marks") used herein are the property of Mentor Graphics Corporation or other parties. No one is permitted to use these Marks without the prior written consent of Mentor Graphics or the owner of the Mark, as applicable. The use herein of a third-party Mark is not an attempt to indicate Mentor Graphics as a source of a product, but is intended to indicate a product from, or associated with, a particular third party. A current list of Mentor Graphics' trademarks may be viewed at: www.mentor.com/trademarks.

End-User License Agreement: You can print a copy of the End-User License Agreement from: www.mentor.com/eula.

Mentor Graphics Corporation
8005 S.W. Boeckman Road, Wilsonville, Oregon 97070-7777.
Telephone: 503.685.7000
Toll-Free Telephone: 800.592.2210
Website: www.mentor.com
SupportNet: supportnet.mentor.com/

Send Feedback on Documentation: supportnet.mentor.com/doc_feedback_form

Table of Contents

Introduction	5
Running HypScripts	5
Setting Directories	6
SET_DIR, INPUT, [<Relative_Path>]	6
SET_DIR, OUTPUT, [<Relative_Path>]	6
REPORT_SET_FILE, Script.log	6
Loading a Design	8
LOAD_FFS_FILE, <filename>	8
LOAD_HYP_FILE, <filename>	8
LOAD_PJH_FILE, <filename>	8
LOAD_ODBPP_FILE, <ODB folder>	8
SI Commands	9
General Settings	9
NET_SELECT_BY_NAME, <net_name>	9
NET_SELECT_BY_PIN, <Ref Des>(<Pin>)	9
SIM_SET_FLAVOR, <simulator>	9
SIM_SET_LOSSY, <state>	9
HYBRID_MODELING, <action>	10
VIA_MODEL, <model>	10
SIM_SET_RESOLUTION, <type>[, <value>]	10
SIM_ENABLE_ROUGHNESS, <State>	10
SIM_REMOVE_NFP, <State>	11
CROSSTALK, <setting>	11
ADVANCED_COUPLING, <setting>	12
PWR_AWARE_IBIS, ON/OFF	14
PWR_BUS_COUPLING, ON/OFF	14
SIM_ENABLE_TRAPEZOIDAL_TRACE , ON/OFF	14
SIM_SERIES_CAPACITOR_SETTLING , ON/OFF	14
Model Settings	15
ASSIGN_IC_MODEL, <pin_name>, <model_file>, <device> [, < model_pin>]	15
SET_BUFFER_DIR, <pin_name>, <setting>	15
SET_MODEL_SELECTOR, < pin_name >, <model>	16
Using the Oscilloscope	17
SCOPE_OPEN	17
NET_SET_STIM, EDGE	17
NET_SET_STIM, OSC	17

GLOBAL_SET_IBIS_SIM, <corner>	17
SCOPE_SET_HORZ, <value>	18
SCOPE_SET_VERT, <value>	18
SCOPE_SET_SIM, COSIM.....	18
SCOPE, <command>	18
SCOPE_START	18
SCOPE_CLOSE	19
Using EZwave.....	20
PROBE_NET, <action>.....	20
SIM_REC_WAVEFORMS, <set_of_waveforms>.....	22
Sweeps.....	23
SWEEP_MANAGER_OPEN.....	23
SWEEP_MANAGER_CLOSE.....	23
DDRx Wizard.....	23
DDR_WIZARD, <action>.....	23
Terminator Wizard.....	24
TWIZ_START	24
TWIZ_SAVE_FILE, <filename>.....	24
General Batch Simulation.....	24
BOARD_WIZARD, <gbs_file>.....	24
BOARD_WIZARD_AUDIT.....	24
SerDes Simulation.....	25
SERDES_WIZARD, <sws_file>.....	25
SERDES_COMPLIANCE, <sws_file>.....	25
SERDES_MEASURED, <sws_file>	25
FastEye/IBIS-AMI	26
FASTEYE_ANALYSIS, <few_file>	26
AMI_ANALYSIS, <few_file>	26
3D Via Model - LineSim.....	26
VIA_PROP, <action>	26
PI Commands.....	27
DC Drop.....	27
DC_DROP, <action>.....	27
BATCHDC_DROP, <action>	28
Decoupling Wizard.....	29
DECOUPLING_WIZARD, <action>	29
Advanced Decoupling Wizard	30
ADV_DECOUPLING_WIZARD, <action>.....	30

Bypass Wizard.....	30
BYPASS_WIZARD, <action>.....	30
Plane Noise.....	32
PLANE_NOISE, <action>.....	32
Thermal Commands.....	33
THERMAL, <action>.....	33
BATCHTHERMAL, <action>.....	33
Export Commands.....	34
Export to FFS.....	34
EXPORT_TO_FFS, <action>.....	34
Export to SPICE Netlist.....	35
SPICE, SAVE, <filename>.....	35
Export Net to S-parameter.....	35
EXTRACT_S, <action>.....	35
Export board to EBD.....	36
HYP_TO_EBD, <ref_des> [, <filename>].....	36
Export board to ICX.....	36
EXPORT_TO_ICX, <action>.....	36
Export Signal-Via Model.....	37
VIAMODEL_EXTRACTOR, <action>.....	37
DIFFVIA_EXTRACTOR, <action>.....	38
Export Stackup.....	38
EXPORT_STACKUP, <file_name.stk>.....	38
Export PDN Model.....	39
PDN_WIZARD, <action>.....	39
Export Design Archive.....	39
EXPORT_TO_ARCHIVE, <action>.....	39
Miscellaneous Commands.....	40
* <comment>.....	40
SET_SEL_FILTER, <filter>.....	40
CENTER_VIEW, <cx>, <cy>.....	40
ZOOM_VIEW, <dx>, <dy>.....	40
ZOOM_VIEW, board.....	40
SET_VIEWPORT, <cx>, <cy>, <dx>, <dy>.....	40
MAXIMIZE_TPE.....	40
MOUSE_MOVE, x, y.....	41
MOUSE_DBLCLICK.....	41
BREAK.....	41

FINI	41
PAUSE	41
SLEEP [, N]	41
TIME_STAMP	41
TIME_STAMP_SET_FILE, <filename>	41
TIME_STAMP_TO_FILE [, <action>]	41

Introduction

This document provides information about the HypScript based automation capabilities of HyperLynx, which had its first controlled release in version 9.0. All available HypScript commands and some examples are described in the following sections.

Caution:

It is important to note that manually interacting with HyperLynx while the HypScripts are running is not supported and may lead to crashes. Therefore it is best to run these HypScripts on a dedicated machine with no user interaction.

Running HypScripts

HypScripts are regular text files that contain specific commands, which will be interpreted and executed by HyperLynx. The commands are executed line by line, meaning that HyperLynx will move on to the next line only when it finished executing the current line.

Caution:

White space on line endings for HypScript commands can result in syntax error. In case of syntax error a message is written to the log file and the HypScript is aborted.

To run a HypScript specify it using the **-SCRIP** command line argument to the HyperLynx executable:

bsw.exe -SCRIP:<Path>/Script.hypscript

Another very useful argument is **-INIF**, which allows using a specific bsw.ini file. The bsw.ini file contains general setup and preferences, such as the IC model libraries. Optionally set the bsw.ini file to read-only to make sure it is not updated or overwritten.

Example:

bsw.exe -SCRIP:<Path>/Script.hypscript -INIF:<Path>/bsw.ini

There are a few other command line arguments that set different working directories and these are presented in the following section.

Minimized Mode:

HyperLynx will run in minimized mode when using the **-SCRIP_MINIMIZED** command line option. This limits HyperLynx from taking over focus and mouse control while the HypScripts are running.

Setting Directories

The following directories can be set when running a HypScript:

- **Input** directory is where all the input files provided to HyperLynx are located. This includes for example hyp, ffs, cce, and any other setting files.

We can note here that the Input directory in most cases can't be read only. The files inside the directory can be read-only, but not the directory itself because in most cases HyperLynx needs to be able to write out temporary files to this directory.

- **Output** directory is where all the output files and results are saved
- **Log** directory is where the log file created from running the hypscript is saved

These directories can be set in two steps: from the command line and from the HypScript. There is a command line argument for each of these paths, which can be used to set an absolute path to a high level directory. Going down to the test specific subdirectories can be done from the hypscript using relative paths. This helps to make the tests portable.

Setting directories **from the command line** can be done using the following arguments:

- Input directory: **-SCRPIN:<Absolut_Path>**
- Output directory: **-SCRPOUT:<Absolut_Path>**
- Log directory: **-SCRPLOG:<Absolut_Path>**

Setting paths **from inside the hypscript** file can be done using the following commands:

- Input directory:
SET_DIR, INPUT, [<Relative_Path>]
- Output directory:
SET_DIR, OUTPUT, [<Relative_Path>]
- Log file: - specify the name of the log file within the hypscript. This will be saved in the location set with **-SCRPLOG** from command line:

REPORT_SET_FILE, Script.log

REPORT_SET_FILE must be the **first command** in the HypScript file. This log file can give valuable information when debugging a hypscript, because it will show for example if any of the directories were not found or if there is any syntax error.

The final paths used by HyperLynx will be **<Absolut_Path>/<Relative_Path>**

A few notes regarding the paths:

- If <Relative_Path> is blank in the SET_DIR command, HyperLynx will use the <Absolute_Path> specified from command line
- Alternatively, you can set the full path from the hypscript using the SET_DIR command – just put in a full path for <Relative_Path>
- No need to include a trailing slash when specifying both <Absolute_Path> and <Relative_Path>

Example – Setting Directories

If we have the following command line:

```
<Path>\bsw.exe -SCRIP:C:\Test\Script.hypscript -INIF:C:\Test\bsw.ini  
-SCRPIN:C:\Test\Inputs -SCRPOUT:C:\Test\Outputs -SCRPLOG:C:\Test\Logs
```

And we'll have the following in C:\Test\Script.hypscript:

```
* HypScript test file  
* REPORT_SET_FILE must be the first command line in the file  
REPORT_SET_FILE, Script.log  
  
SET_DIR, INPUT, Ffs/BlackBox/  
SET_DIR, OUTPUT, Ffs/BlackBox/
```

The following paths will be used:

- Input directory: C:\Test\Inputs\ Ffs\BlackBox
- Output directory: C:\Test\Outputs\ Ffs\BlackBox
- Log File: C:\Test\Logs\Script.log

Notice how the final paths are constructed from the absolute path given in command line and the relative path from the hypscript (it is color coded for Inputs directory).

Also, notice that by using forward slashes in the hypscript (“/”), we would be able to run the hypscript on both Windows and Linux – we would only need to change the absolute paths from command line.

We could also note here that the lines that start with * symbol are comment lines.

Loading a Design

Loading a design from a hypscript can be done using the following commands:

LOAD_FFS_FILE, <filename>

Loads the specified schematic into LineSim. If the specified filename is a full path (i.e. includes a drive specifier or UNC prefix), then the specified filename will be used as-is. If the specified filename is NOT a full path, then the specified filename is assumed to be relative to the [Input directory](#).

LOAD_HYP_FILE, <filename>

Loads the specified board file into BoardSim. If the specified filename is a full path (i.e. includes a drive specifier or UNC prefix), then the specified filename will be used as-is. If the specified filename is NOT a full path, then the specified filename is assumed to be relative to the [Input directory](#).

LOAD_PJH_FILE, <filename>

Loads the specified multi-board project into BoardSim (.pjh file). If the specified filename is a full path (i.e. includes a drive specifier or UNC prefix), then the specified filename will be used as-is. If the specified filename is NOT a full path, then the specified filename is assumed to be relative to the [Input directory](#).

LOAD_ODBPP_FILE, <ODB folder>

Loads the specified ODB design folder into BoardSim. The specified filename is assumed to be relative to the [Input directory](#).

SI Commands

General Settings

NET_SELECT_BY_NAME, <net_name>

Corresponds to “Select/Net by Name for SI Analysis.” The value of <net_name> is the name of the net. It selects a net for SI analysis in BoardSim – the command is not available in LineSim.

For multi-board add _B0x (board number) after the net name (ex. DATA1_B00).

Starting with HL_9.2 this command supports selecting multiple nets. The net names should be separated by space.

Example:

```
NET_SELECT_BY_NAME, DATA1 DATA2 DATA3
```

NET_SELECT_BY_PIN, <Ref Des>(<Pin>)

Corresponds to “Select/Net by Reference Designator for SI Analysis.” Specify Reference Designator and Pin Name in parenthesis.

Example:

```
NET_SELECT_BY_PIN, U(B2)
```

SIM_SET_FLAVOR, <simulator>

Sets which simulator to use. The value of <simulator> can be:

- **HSPICE** = HSPICE
- **ADMS** = ADMS
- **ICX** = HyperSim
- **Auto** = simulator will be chosen automatically

SIM_SET_LOSSY, <state>

Controls the use of Lossy transmission lines. Value of <state> can be:

- **ON** = transmission lines will include loss behavior
- **OFF** = transmission lines will be lossless

HYBRID_MODELING, <action>

- **ON/OFF** = Corresponds to “Setup / Enable PI Effects in SI Simulations”
- **IMPORT_MODEL, modelName, modelPath [, boardName]**
= Imports Power Aware SI Model. The assignment to a net is done automatically based on the port mapping data.

Example:

```
HYBRID_MODELING, ON  
HYBRID_MODELING, IMPORT_MODEL, plugin, plugin.s6p, B01
```

VIA_MODEL, <model>

Sets the via modeling to be used during simulation. Value of <model> can be:

- **NONE** = Vias are not to be simulated
- **CAP_ONLY** = Model vias as single lumped capacitors
- **AUTO** = Auto-calculate the circuit topology and values to be used
- **GLOBAL** = Uses global L and C for vias. Default L and C values are used unless overridden by .BUD
- **PADSTACK** = Uses padstack-specific L and C. If .BUD does not specify L and C for a padstack, then that padstack uses "AUTO". See "AUTO" above.)

SIM_SET_RESOLUTION, <type>[, <value>]

Sets the simulation time resolution for both Oscilloscope and EZwave simulations.

<type> is resolution type:

- **Auto** = For automatic resolution calculation
- **Fixed** = For user-defined resolution

<value> is resolution value in picoseconds for Fixed resolution type

Example:

```
SIM_SET_RESOLUTION, Auto  
SIM_SET_RESOLUTION, Fixed, 1
```

SIM_ENABLE_ROUGHNESS, <State>

Allows to enable / disable the calculation of surface roughness effects during simulation. It corresponds to “Setup / Enable Surface Roughness” menu selection.

<State> can be the following:

- True = Enabled
- False = Disabled

Example:

SIM_ENABLE_ROUGHNESS, True

SIM_REMOVE_NFP, <State>

Allow to remove / restore non-functional via pads. It corresponds to "Setup / Remove Non-Functional Pads" menu selection.

<State> can be the following:

- True = Enabled
- False = Disabled

Example:

SIM_REMOVE_NFP, True

CROSSTALK, <setting>

Corresponds to "Enable Crosstalk Simulation" menu, the "Setup/Coupling Thresholds" dialog, and a check box from Setup/Options/General/Advanced tab. The value for <setting> can be:

- **ENABLE** = Corresponds to "Enable Trace Coupling" menu check box. Enables CrossTalk simulation
- **DISABLE** = Corresponds to "Enable Trace Coupling" menu check box. Disables CrossTalk Simulation
- **CACHE_ON** = Corresponds to enabling "Use field-solver cache" under "Setup/Options /General/Advanced" tab
- **CACHE_OFF** = Corresponds to disabling "Use field-solver cache" under "Setup/Options /General/Advanced" tab
- **ELECTRICAL** [, <threshold>] = Corresponds to "Use electrical thresholds" radio button in "Setup/Coupling Thresholds" Dialog. <threshold> parameter corresponds to "Include nets with coupled voltage greater than" field and it is in Volts. If the parameter is omitted, the current value is reported to the log file.
- **GEOMETRIC** = Corresponds to "Use geometric thresholds(advanced)" radio button in "Setup/Coupling Thresholds" dialog. It is available only in BoardSim.
- **MAXIMUM_DISTANCE_FROM_AGGRESSOR** , <value>

Corresponds to "Maximum distance from aggressor" edit box in "Setup/Coupling Thresholds" dialog. It is specified in meters. Use it only in case of geometric threshold.

- **MINIMUM_COUPLED_TRACE_LENGTH , <value>**

Corresponds to "Minimum total coupled trace length" edit box in "Setup/Coupling Thresholds" dialog. It is specified in meters. Use it only in case of geometric threshold.

- **HORIZONTAL_NEIGHBOR_LIMIT, <value>**

Corresponds to "Horizontal Neighbor Limit" edit box in "Setup/Coupling Thresholds" dialog. <value> must be an integer. Use it only in case of geometric threshold.

- **VERTICAL_NEIGHBOR_LIMIT, <value>**

Corresponds to "Vertical Layer Limit" edit box in "Setup/Coupling Thresholds" dialog. <value> must be an integer. Use it only in case of geometric threshold.

ADVANCED_COUPLING, <setting>

Assigns Coupling settings and corresponds to "Setup / Coupling Settings" dialog. The value for <setting> can be:

- **CONSIDER_SAME_NET_COUPLING [, ON/OFF]**

Corresponds to "Include self-coupling" check box in Coupling Settings dialog. Set it to either ON or OFF. If [, ON/OFF] parameter is omitted, the current value is reported to the log file. It is relevant only if trace coupling has been enabled (CROSSTALK, ENABLE)

- **ALLOW_ARBITRARY_TRACE_ANGLES [, ON/OFF]**

Corresponds to "Include non-parallel coupling" check box in Coupling Settings dialog. If [, ON/OFF] parameter is omitted, the current value is reported to the log file. It is relevant only if trace coupling has been enabled (CROSSTALK, ENABLE)

- **INCLUDE_SPARAM_COUPLING [, ON/OFF]**

Corresponds to "Include S-parameter coupling" check box. If parameter is omitted, current value will be reported.

- **SPARAM_MIN_FREQUENCY [, <value MHz>]**

Corresponds to S-parameter "Frequency range > Min" edit box box, in MHz. If parameter is omitted, current value will be reported.

- **SPARAM_MAX_FREQUENCY [, <value MHz>]**

Corresponds to S-parameter "Frequency range > Max" edit box box, in MHz. If parameter is omitted, current value will be reported.

-
- **SPARAM_ALL_COUPLING, [, ON/OFF, [, <dB threshold>]]**
Corresponds to "Use same thresholds for all" controls. If parameter is omitted, current value will be reported.
 - **SPARAM_PACKAGE_COUPLING, [, ON/OFF, [, <dB threshold>]]**
Corresponds to "Include package coupling" controls. If parameter is omitted, current value will be reported.
 - **SPARAM_CONNECTOR_COUPLING, [, ON/OFF, [, <dB threshold>]]**
Corresponds to "Include connector coupling" controls. If parameter is omitted, current value will be reported.
 - **SPARAM_AREA_COUPLING, [, ON/OFF, [, <dB threshold>]]**
Corresponds to "Include 3D-EM solved area and Power Aware SI coupling" controls. If parameter is omitted, current value will be reported.
 - **INCLUDE_TRACE_TO_AREA_FILL_COUPLING [, ON/OFF]**
Corresponds to "Include trace to area fill coupling" check box in Coupling Settings dialog. Set it to either ON or OFF. If [, ON/OFF] parameter is omitted, the current value is reported to the log file. It must be used with Geometric coupling (CROSSTALK, GEOMETRIC)
 - **AREA_FILL_SEARCH_DISTANCE, <value>**
Corresponds to "Area Fills Search Distance" in Coupling Settings dialog. <value> is expressed in Meters. Use it only if trace to area fill coupling is turned ON.
 - **IGNORE_SEGMENTS_SHORTER_THAN, <value>**
It corresponds to "Area fill grid" in Coupling Settings dialog. <value> is expressed in Meters. Use it only if trace to area fill coupling is turned ON.

Example – Coupling settings:

* Enable trace coupling
CROSSTALK, ENABLE

* Use Geometric thresholds
CROSSTALK, GEOMETRIC

* Set the Geometric thresholds
CROSSTALK, MAXIMUM_DISTANCE_FROM_AGGRESSOR, 0.001016
CROSSTALK, MINIMUM_COUPLED_SEGMENT_LENGTH, 0.000762
CROSSTALK, HORIZONTAL_NEIGHBOR_LIMIT, 2
CROSSTALK, VERTICAL_NEIGHBOR_LIMIT, 2

* Trace to trace coupling settings

ADVANCED_COUPLING, CONSIDER_SAME_NET_COUPLING, ON
ADVANCED_COUPLING, ALLOW_ARBITRARY_TRACE_ANGLES, ON

* Trace to area fill coupling settings

ADVANCED_COUPLING, INCLUDE_TRACE_TO_AREA_FILL_COUPLING, ON
ADVANCED_COUPLING, AREA_FILL_SEARCH_DISTANCE, 0.001016
ADVANCED_COUPLING, IGNORE_SEGMENTS_SHORTER_THAN, 0.000508

PWR_AWARE_IBIS, ON/OFF

Enables / disables PI effects in SI simulations. It corresponds to “Setup / Enable PI Effects in SI Simulations” menu selection.

PWR_BUS_COUPLING, ON/OFF

Enables / disables Power bus coupling in BoardSim when PWR_AWARE_IBIS is ON.
Corresponds to “Setup / PI Effects settings / Include power bus coupling”

SIM_ENABLE_TRAPEZOIDAL_TRACE , ON/OFF

Enables / disables the use of Trapezoidal Trace Shapes. Interactively this setting is under the Manufacturing tab in the Stackup Editor.

SIM_SERIES_CAPACITOR_SETTLING , ON/OFF

Enables / disables settling of series capacitors before every simulation. It corresponds to “Setup / Options / General – Advanced tab, Settle Series Capacitors Before Every Simulation” checkbox

Model Settings

ASSIGN_IC_MODEL, <pin_name>, <model_file>, <device> [, <model_pin>]

Assigns model to the specified IC pin.

<pin_name> is the complete IC pin name, in form of <Component Reference Designator>.<Pin>, for example, U10.A1

For multi-board add _B0x (board number) after the reference designator, for example, U10_B00.A1

<model_file> is model file name without path, for example, v80a.ibs

<device> is the IBIS component name or SPICE circuit name, for example, MT41J1G4RH

<model_pin> is name of model pin corresponding to the IC pin, e.g. B3

Example:

ASSIGN_IC_MODEL, U3.16, u67a.ibs, MT47H128M4CF, C2

SET_BUFFER_DIR, <pin_name>, <setting>

Assigns buffer settings – it corresponds to the buffer settings in the “Assign Models” dialog. To set the buffer in BoardSim, the net containing the buffer must be selected prior to running this command.

This command makes changes directly on the internal database, so the settings won't be synchronized with the displayed symbols in LineSim.

<pin_name> is the complete IC pin name, in form of <Component Reference Designator>.<Pin>, for example, U10.A1

For multi-board add _B0x (board number) after the reference designator, for example, U10_B00.A1

<settings> is one of predefined values corresponding to the particular combination of Buffer direction, direct/inverted and signal states. It can have the following values:

- **Input** – input, non-inverted, switching (applicable to input and bi-directional buffers)
- **Output** – output, non-inverted, switching (applicable to output, bi-directional, tri-state and SPICE buffers)
- **Output Inverted** – output, inverted, switching (applicable to output, bi-directional and tri-state buffers)
- **Stuck High** – output, non-inverted, high-state (applicable to output, bi-directional and tri-state buffers)
- **Stuck Low** – output, non-inverted, low-state (applicable to output, bi-directional and tri-state buffers)
- **Output Hi-Z** – input, non-inverted, switching (applicable to tri-state buffers)

Example:

SET_BUFFER_DIR, U3.N3, Output

SET_MODEL_SELECTOR, < pin_name >, <model>

Assigns a particular IC model in Model Selector. In BoardSim the net containing the buffer must be selected prior to setting the model.

White space is not allowed after the <model> or the command fails to execute.

<pin_name> is the complete IC pin name, in form of <Component Reference Designator>.<Pin>, for example, U10.A1

For multi-board add _B0x (board number) after the reference designator, for example, U10_B00.A1

<Model> is one of models listed in Model Selector corresponding to the pin

Example:

SET_MODEL_SELECTOR, U4.E3, DQ_40_1066

Using the Oscilloscope

SCOPE_OPEN

Opens the Oscilloscope. Run this command before running any of the following ones related to Oscilloscope – the Oscilloscope needs to be open for the following commands to work.

NET_SET_STIM, EDGE

Enables the radio button Edge under Stimulus for SI simulations.

Furthermore, we can select to use a Rising or Falling edge stimulus using the following command:

NET_SET_EDGE, <edge>

The value for <edge> can be:

- **FALL** = Falling edge
- **RISE** = Rising edge

NET_SET_STIM, OSC

Enables the radio button Oscillator under Stimulus for SI simulations.

Furthermore, we can specify the frequency and Duty of the oscillator stimulus using the following command:

NET_SET_OSC, <frequency>, <duty_cycle>

The value for <frequency> can be specified with units, for example MHz or GHz.

The value for duty cycle must be between 5 and 95. Use this command only if the Stimulus is set to Oscillator.

Example:

```
NET_SET_OSC, 200MHz, 49
```

GLOBAL_SET_IBIS_SIM, <corner>

Sets the IC modeling corner used by the SI simulator. Value of <corner> can be:

- **BEST** = fast-strong
- **TYPICAL** = typical
- **WORST** = slow-weak

SCOPE_SET_HORZ, <value>

Corresponds to the Horizontal Scale setting in the Oscilloscope. Specify <value> with units (ps, ns). The setting corresponds to <value>/div

Example:

SCOPE_SET_HORZ, 2ns

SCOPE_SET_VERT, <value>

Corresponds to the Vertical Scale setting in the Oscilloscope. Specify <value> with units (mV, V). The setting corresponds to <value>/div

Example:

SCOPE_SET_VERT, 2V

SCOPE_SET_SIM, COSIM

Enables “Simulate t-planes” option to run SI/PI Co-simulation. The Oscilloscope must be open to run this command.

SCOPE, <command>

Corresponds to Oscilloscope actions. The Oscilloscope must be open at the time of running these commands (run SCOPE_OPEN command). The value for <command> can be the following:

- **ERASE** = Corresponds to Erase button in Oscilloscope – erases all waveforms
- **LOAD, <filename>** = Loads a .HypWave file to the Oscilloscope. <filename> needs to be specified using full path
- **SAVE, <filename>** = Saves the waveforms from the Oscilloscope as a .HypWave file, which is a binary form
- **SAVECSV, <filename>** = Saves the waveforms from the Oscilloscope as a .CSV file. This is the recommended way of saving the waveforms from the Oscilloscope, because they can be loaded back manually for visualization if needed.

SCOPE_START

Corresponds to the Start Simulation button from the Oscilloscope – it starts the simulation.

SCOPE_CLOSE

Closes the Oscilloscope.

Example – Oscilloscope

Running a simulation using the Oscilloscope:

```
LOAD_FFS_FILE,    Test1.ffe
SCOPE_OPEN
* Set IC modeling type
GLOBAL_SET_IBIS_SIM, TYPICAL

* Set Horizontal / Vertical Scale
SCOPE_SET_HORZ,    2ns
SCOPE_SET_VERT,    1V

* Run simulation using a rising edge stimulus and save the result
NET_SET_STIM,      EDGE
NET_SET_EDGE,      RISE
SCOPE_START
SCOPE, SAVECSV,    test1_rise
SCOPE, ERASE

* Run simulation using a falling edge stimulus and save the result
NET_SET_EDGE,      FALL
SCOPE_START
SCOPE, SAVECSV,    test1_fall
SCOPE, ERASE

* Run simulation using an Oscillator stimulus and save the result
NET_SET_STIM,      OSC
NET_SET_OSC,        200MHz, 49
SCOPE_START
SCOPE, SAVECSV,    test1_osc
```

Using EZwave

PROBE_NET, <action>

These commands can be used to run SI simulations using EZwave. <action> can have any of the following values:

- **OPEN**

Same as “Simulate SI/Run Interactive Simulation in EZwave/And Show Waveforms” menu selection. It opens the Interactive Simulation dialog – this should be the first step before running the following commands.

- **SIM_TIME, <value> / AUTO**

Sets Simulation duration to a specific value or AUTO. <value> should be expressed with units. Run only when Interactive Simulation dialog is open (PROBE_NET, OPEN).

Example:

PROBE_NET, SIM_TIME, 10ns

- **STIMULUS, PER_NET_PIN**

Sets the Stimulus to Per-net/pin. Run only when Interactive Simulation dialog is open.

- **STIMULUS, EDGE [,FALL/RISE]**

Sets the Stimulus to Falling or Rising Edge.

Example:

PROBE_NET, STIMULUS, EDGE, FALL

- **STIMULUS, OSC [, <frequency>, <duty_cycle>]**

Sets Stimulus to Oscillator and optionally allows setting frequency and duty_cycle too. <frequency> must have units (MHz, GHz) and duty cycle must be between 5 and 95.

Example:

PROBE_NET, STIMULUS, OSC ,1.5GHz, 49

- **CORNER, WORST / TYPICAL / BEST**

Sets the IC modeling corner.

- **BEST** = fast-strong
- **TYPICAL** = typical
- **WORST** = slow-weak

Example:

PROBE_NET, CORNER, TYPICAL

- **CLEAR_RESULTS, ON/OFF**

It corresponds to the “Clear previous results” check-box in the Interactive Simulation dialog. It removes previous EZWave waveforms.

- **PLOT_WAVEFORMS, ON/OFF**

It corresponds to the “Plot waveforms” check-box in the Interactive Simulation dialog. It automatically plots the waveforms in EZWave when turned ON.

- **SIMULATE_T-PLANES, ON/OFF**

It enables/disables the “Simulate t-planes” option for running SI/PI Co-simulation.

- **RUN**

It corresponds to the Start Simulation button.

- **SAVE [, <wdb_filename>]**

It saves waveform from the preceding RUN as .wdb file. The specified <wdb_filename> is relative to the [Output directory](#). If <wdb_filename> is not specified, the waveform will be saved with the default name, which is the name of the currently selected net.

Example – Ezwave

Running a simulation using EZwave:

```
LOAD_HYP_FILE,    Test.hyp
NET_SELECT_BY_NAME, N2

PROBE_NET, OPEN
PROBE_NET, SIM_TIME, 10ns
PROBE_NET, STIMULUS, EDGE, RISE
PROBE_NET, CORNER, TYPICAL
PROBE_NET, RUN
PROBE_NET, SAVE
```

SIM_REC_WAVEFORMS, <set_of_waveforms>

Allows specifying a set of additional waveforms to generate/record during EZWave simulation. It corresponds to the “Additional waveforms” settings under Simulation Controls.

<set_of_waveforms> is a set of desired waveforms names delimited by spaces, including the following waveforms:

- **Nets** – waveforms for coupled electrical nets
- **Currents** – waveforms for buffer currents
- **Vias** – waveforms at vias
- **Test** – test waveforms
- **PinDie** – pin and die waveforms
- **Digital** – digital waveforms

Example:

SIM_REC_WAVEFORMS, Nets Currents PinDie Digital

Sweeps

To runs sweeps just open the Sweep Manager using the following command before opening the Oscilloscope for example.

SWEEP_MANAGER_OPEN

It opens Sweep Manager. The Sweeps are saved inside the design.

SWEEP_MANAGER_CLOSE

Closes Sweep Manager

Example - Sweeps

Sweeps with Oscilloscope:

```
SWEEP_MANAGER_OPEN
SCOPE_OPEN
SCOPE_SET_HORZ, 1ns
SCOPE_SET_VERT, 1v
NET_SET_EDGE, RISE
SCOPE_START
SCOPE, SAVECSV, TL_IBIS_rise
SWEEP_MANAGER_CLOSE
```

DDRx Wizard

DDR_WIZARD, <action>

It runs the DDRx Batch-Mode Wizard. The value for <action> can be:

- **OPEN** = Opens the DDRx Wizard. It must be the first DDRx command you run.
- **LOAD, <DDRx_setting_file>** = Imports the DDRx Settings file (.ddr). Save the setup data from the Wizard to create this file.
- **RUN** = It runs the DDRx simulation.

Example – DDRx Wizard

Run DDRx simulation on a Multi-board project:

```
LOAD_PJH_FILE, NewMultipleBoardProject.pjh
DDR_WIZARD, OPEN
DDR_WIZARD, LOAD, NewMultipleBoardProject.ddr
DDR_WIZARD, RUN
```

Terminator Wizard

TWIZ_START

Runs the Terminator Wizard. It corresponds to menu selection “Simulate SI/Optimize Termination.” In BoardSim select a net before running this command.

TWIZ_SAVE_FILE, <filename>

It saves the result of the Terminator Wizard as <filename>.HypTwiz in the [Output directory](#).

General Batch Simulation

BOARD_WIZARD, <gbs_file>

This command will run the general batch mode simulation on the loaded board in BoardSim using the settings provided in the gbs file. A gbs file can be saved from the General Batch Wizard interactively and is also saved in results folder with each simulation.

Example – Batch Mode

```
LOAD_HYP_FILE, DEMO.hyp
BOARD_WIZARD, test.gbs
```

BOARD_WIZARD_AUDIT

This command will run the general batch audit on the loaded board.

SerDes Simulation

SERDES_WIZARD, <sws_file>

It runs the SERDES Batch Wizard using the settings from the .sws file both in LineSim and BoardSim. The Wizard allows interactively saving a .sws settings file.

Example – SerDes Batch

SERDES_WIZARD, Test.sws

SERDES_COMPLIANCE, <sws_file>

It runs the SERDES Wizard in Compliance Mode only using the settings from the .sws file both in LineSim and BoardSim. The Wizard allows interactively saving a .sws settings file.

Example – SerDes Compliance Mode

SERDES_COMPLIANCE, Test.sws

SERDES_MEASURED, <sws_file>

It runs the SERDES Wizard in Compliance Mode using Measured S-parameters the settings from the .sws file both in LineSim and BoardSim. The Wizard allows interactively saving a .sws settings file.

Example – SerDes Measured Mode

SERDES_MEASURED, Test.sws

FastEye/IBIS-AMI

FASTEYE_ANALYSIS, <few_file>

It runs a Fasteye Channel Analysis based on the setting from the .few settings file. In BoardSim select a net before running this command. The <few_file> is the settings file and can be saved from the Wizard. The result files, like the .tps file or statistical contour .scd file will be in the [Output directory](#).

AMI_ANALYSIS, <few_file>

It runs an IBIS-AMI Channel Analysis based on the setting from the .few settings file. In BoardSim select a net before running this command. The <few_file> is the settings file and can be saved from the Wizard. The result files will be saved in the [Output directory](#).

Example – IBIS-AMI analysis

```
LOAD_FFS_FILE, Mentor_Tx_channel_Mentor_Rx.ffe  
AMI_ANALYSIS, Mentor_Tx_channel_Mentor_Rx.few
```

3D Via Model - LineSim

VIA_PROP, <action>

It creates a 3D via model. This command is only available in LineSim.

<action> can be the following:

- **OPEN, <via name>** = Open Via Property for further 3D simulation
- **3DPROJ [, OPEN]** = Opens 3D EM Project File, which is a .v3d file with all the settings for the model creation
- **3DPROJ, SIMULATE** = Starts the 3D EM simulation
- **3DPROJ, CLOSE** = Close HyperLynx 3D EM Project dialog
- **CLOSE** = Close the Via Properties dialog

Example – 3D via

```
LOAD_FFS_FILE, simple.ffe  
VIA_PROP, OPEN, V1  
VIA_PROP, 3DPROJ  
VIA_PROP, 3DPROJ, SIMULATE  
VIA_PROP, 3DPROJ, CLOSE  
VIA_PROP, CLOSE
```

PI Commands

DC Drop

DC_DROP, <action>

Command used to run DC Drop simulations. It is available in both BoardSim and LineSim. The values for <action> can be:

- **OPEN** = It corresponds to Simulate PI/Run DCDrop Simulation. It opens the DC Drop Analysis dialog. It must be the first DC Drop action before being able to run the following ones.
- **SELECT_NET, <net_name>** = Selects a Power/Ground Net to Analyze by name.
- **REF_ON** = It enables the “Include Reference net(s)” check-box from the DC Drop Analysis dialog. Is it available in LineSim?
- **OUTLOG, <log_file_name>** = Sets the name of the report file, which will contain the DC Drop analysis results. It’s location will be relative to the [Output directory](#).
- **RUN** = Run the DC Drop simulation
- **CLOSE** = Close the DC Drop windows

Example – DC Drop

Running a DC Drop simulation:

```
DC_DROP, OPEN
DC_DROP, SELECT_NET, 3.3V
DC_DROP, OUTLOG, DC_Drop_result.txt
DC_DROP, RUN
DC_DROP, CLOSE
```

BATCHDC_DROP, <action>

Allows running Batch DC Drop Simulation. It is available in BoardSim only.

- **OPEN** = It corresponds to “Simulate PI/Run DC Drop Batch Simulation” menu selection. It opens Batch DC Drop Simulation dialog. It must be the first Batch DC Drop action before being able to run the following ones.
- **LOAD, <file_name>** = It loads a DC Drop Batch Session (.dcs) file, which sets the selected nets and voltage and current thresholds.
- **REF_ON** = It corresponds to enabling the “Include Reference Nets” checkbox. The state of this checkbox can be saved and loaded from the .dcs file too.
- **SCOPEDATA_ON** = It enables the “Create PowerScope Data” checkbox. The state of this checkbox can be saved and loaded from the .dcs file too.
- **SPREADSHEET_ON** = Enables “Create Spreadsheet Reports” option
- **POWERMAP_ON** = Enables “Write power-map files for FloTHERM” option
- **RUN** = Runs simulations. The Batch DC Drop Simulation dialog is automatically closed when the simulation is done.

Decoupling Wizard

DECOUPLING_WIZARD, <action>

Sets up and runs the Decoupling Wizard and generates a Z-Parameter file as a result.

- **OPEN** = Opens the Decoupling Wizard. It corresponds to “Simulate PI/Analyze Decoupling” menu selection. It must be the first action before being able to run the following ones related to Decoupling Wizard – the Wizard must be first opened in order for the following commands to work.
- **LOAD, <dao_filename>** = It loads a .dao configuration file, which will do all the settings for the Decoupling Wizard. The Wizard can create such setting file.
- **OUT, <filename>** = It specifies the name of the Z-parameter result file to be generated. Set this **ONLY** in the case of Lumped or Distributed Analysis. The specified filename is assumed to be relative to the [Output directory](#) and should not begin with a slash.
- **OUTDRC, <filename>** = It specifies the name of the .CSV file to be generated. Set this **ONLY** in the case of Quick Analysis. The specified filename is assumed to be relative to the [Output directory](#) and should not begin with a slash.
- **RUN** = Run the Decoupling Analysis
- **CLOSE** = Close the Decoupling Wizard

Example – Decoupling Wizard

Lumped or Distributed Decoupling Analysis (use OUTDRC command for the Quick Analysis):

```
DECOUPLING_WIZARD, OPEN
DECOUPLING_WIZARD, LOAD, cap11a.dao
DECOUPLING_WIZARD, OUT, cap11a.zNp
DECOUPLING_WIZARD, RUN
DECOUPLING_WIZARD, CLOSE
```

Advanced Decoupling Wizard

ADV_DECOUPLING_WIZARD, <action>

Sets up and runs the Advanced Decoupling Wizard, which generates a Z-Parameter file as a result.

- **OPEN** = Opens the Advanced Decoupling Wizard. It corresponds to “Simulate PI/Advanced Decoupling Analysis” menu selection. It must be the first action before being able to run the following commands related to Advanced Decoupling Wizard.
- **LOAD, <dao_filename>** = It loads a .dao configuration file, which will do all the settings for the Advanced Decoupling Wizard. The Wizard can create such setting file.
- **OUT, <filename>** = It specifies the name of the Z-parameter result file to be generated. The specified filename is assumed to be relative to the [Output directory](#) and should not begin with a slash.
- **RUN** = Run the Analysis
- **CLOSE** = Close Advanced Decoupling Wizard

Bypass Wizard

BYPASS_WIZARD, <action>

Sets up and runs the Bypass Wizard and generates a Z-Parameter file as a result.

- **OPEN** = Opens the Bypass Wizard. It corresponds to “Simulate PI/Analyze Signal-Via Bypassing” menu selection. It must be the first action before being able to run the following commands related to Bypass Wizard.
- **LOAD, <dao_filename>** = It loads a .dao configuration file, which will do all the settings for the Bypass Wizard. The Wizard can create such setting file.
- **OUT, <filename>** = It specifies the name of the Z-parameter result file to be generated. The specified filename is assumed to be relative to the [Output directory](#) and should not begin with a slash.
- **RUN** = Run the Analysis
- **CLOSE** = Close Bypass Wizard

In BoardSim for the Bypass Wizard we need to select the signal via. First we set the selection filter to only vias in the board viewer and then we use a couple of mouse commands to select the via. See the following example.

Example – Bypass Wizard

This is an example of running the Bypass Wizard in BoardSim. Notice the extra commands used to select the via:

```
* set the selection filter to vias only
SET_SEL_FILTER, VIA
BYPASS_WIZARD, OPEN
BYPASS_WIZARD, LOAD, test.dao
* move mouse and select a via
MOUSE_MOVE, 0.0362, 0.0512
MOUSE_DBLCLICK
BYPASS_WIZARD, OUT, Test1.zNp
BYPASS_WIZARD, RUN
BYPASS_WIZARD, CLOSE
```

Plane Noise

PLANE_NOISE, <action>

It is used to run Plane Noise Analysis. The value for <action> can be:

- **OPEN** = Open Plane Noise dialog in BoardSim
- **SELECT_NET, <net_name>** = Select net in Plane Noise dialog in BoardSim
- **OUTLOG, <log_filename>** = Set the name of the log file. Otherwise the default log file name is SSN.log
- **RUN [,<time(ns)>]** = Run the analysis. The <time> parameter is optional and it is the stop time in ns – it corresponds to Simulation Time in the Plane Noise dialog.
- **SAVE, <tps_filename>** = Save the result as a .tps file from the PowerScope
- **CLOSE** = Close Plane Noise dialog and PowerScope

Thermal Commands

THERMAL, <action>

Runs Thermal Simulation on the loaded board. The output file names are not controlled and are generated with default names. Value of <action> can be:

- **RUN** = Run Thermal Analysis
- **CLOSE** = Close Thermal windows

BATCHTHERMAL, <action>

Runs PI/Thermal Co-simulation. The output file names are not controlled and are generated with default names. Value of <action> can be the following:

- **OPEN** = Open ThermalSim and Batch DC Drop simulation windows
- **LOAD, <session_file>** = Load an existing Session File (Batch DC Drop Simulation dialog)
- **SCOPEDATA_ON** = Enable Create PowerScope Data from DC Drop
- **RUN** = Run PI/Thermal Co-simulation
- **CLOSE** = Close Thermal windows

Export Commands

Export to FFS

EXPORT_TO_FFS, <action>

Export the selected net from BoardSim to LineSim. The result is an ffs file.

- **OPEN** = Opens the Export To LineSim Free-Form Schematic dialog. It corresponds to “Export / Net To / Free-Form Schematic” menu selection. It must be the first <action> before calling the following ones.
- **COUPLED_ON** = Enables “Export coupled segments” checkbox
- **COUPLED_OFF** = Disables “Export coupled segments” checkbox
- **VIA_ON** = Export vias
- **VIA_OFF** = Don’t export vias
- **EBD_INSIDE_ON** = Enables “Expand into EBD models” checkbox. This checkbox is only available if the selected net has a pin with an EBD model.
- **EBD_INSIDE_OFF** = Disables “Expand into EBD models” checkbox. This checkbox is only available if the selected net has a pin with an EBD model.
- **TPE_ON** = Exports to PDN Editor
- **TPE_OFF** = Doesn’t export to PDN Editor
- **SAVE [, <filename>]** = Save to <filename> if specified or otherwise save with default name, which is the same as name of currently selected net. <filename> is relative to the [Output directory](#). This command creates the .ffs file, so all the commands with the different options must be specified before.
- **CLOSE** = Close the Export To LineSim Free-Form Schematic dialog.

Example – Export to FFS

```
LOAD_HYP_FILE, Test.hyp
NET_SELECT_BY_NAME, 1N56
EXPORT_TO_FFS, OPEN
EXPORT_TO_FFS, COUPLED_ON
EXPORT_TO_FFS, TPE_OFF
EXPORT_TO_FFS, VIA_ON
EXPORT_TO_FFS, SAVE, 1N56.ffs
EXPORT_TO_FFS, CLOSE
```

Export to SPICE Netlist

SPICE, SAVE, <filename>

Corresponds to “Export / SPICE Netlist” in LineSim and “Export / Net To / SPICE Netlist” in BoardSim menu selections. It creates a SPICE netlist of the selected net.

<filename> specifies the name of the Touchstone file to be generated. The specified filename is assumed to be relative to the [Output directory](#).

Example – Export S-parameter

```
LOAD_HYP_FILE, Demo_Clk.hyp
NET_SELECT_BY_NAME, CLK
SPICE, SAVE, clk-out
```

Export Net to S-parameter

EXTRACT_S, <action>

It is used to extract the S-parameter model of a net. <action> can be the following:

- **OPEN** = Corresponds to “Export/Net To/S-parameter Model” menu selection. It opens the Extract S-Parameter Model dialog.
- **MAP, AUTO** = It automatically maps a Port index to the Pin names. Currently this is the only supported way.
- **FREQUENCY_RANGE, <startFrequency>, <endFrequency>**
Corresponds to Min and Max frequency under Modeling parameters. It specifies the frequency range for the S-parameter model. The frequency points can be specified using units (ex. MHz, GHz)
- **FREQUENCY_SPACING, <type>**
Corresponds to Sweeping Type. The value of <type> corresponds to the GUI selections and can be the following:
 - **LINEAR, <numOfPoints>**
 - **LOGARITHMIC, <numOfPointsPerDecade>**
 - **ADAPTIVE, LOW/MEDIUM/HIGH**
- **RREF, value** = Allows setting a custom Reference Impedance for the model
- **CREATE [, filename]** = If the file name is not specified, it is determined based on the name of the net.
- **CLOSE** = Close extract S-parameters section

Example – Export S-parameter

```
LOAD_HYP_FILE, test.hyp
NET_SELECT_BY_NAME, data-a4
EXTRACT_S, OPEN
EXTRACT_S, MAP, AUTO
EXTRACT_S, FREQUENCY_RANGE, 0.1MHz, 1GHz
EXTRACT_S, FREQUENCY_SPACING, LOGARITHMIC, 500
EXTRACT_S, CREATE
EXTRACT_S, CLOSE
```

Export board to EBD

HYP_TO_EBD, <ref_des> [, <filename>]

It corresponds to menu selection “Export / Board To / IBIS .EBD File.”

It generates an IBIS .EBD file from the currently loaded board. The first argument <ref_des> specifies the component on the board to be used as the reference connector for the EBD. The second argument <filename> is optional and specifies the filename (without path or extension) of the generated .EBD file. If no filename is specified, then the name of the currently loaded board will be used.

The generated .EBD file will be written in the [Output directory](#). If no output directory has been set, then the file will be written in the current working directory.

If an IBIS file (containing terminators) is written, then the name of that file will be the same as the .EBD file (<filename>), but with an .IBS extension.

Export board to ICX

EXPORT_TO_ICX, <action>

It corresponds to menu selection “Export / Board To / ICX NDD File.”

<action> can be the following:

- **OPEN** = Open Export To ICX dialog
- **SAVE, <filename>** = Save to <filename>, which is relative to the [Output directory](#).
- **CLOSE** = Close Export To ICX dialog

Export Signal-Via Model

VIAMODEL_EXTRACTOR, <action>

It corresponds to “Export / Model / Signal-Via Model” in LineSim and “Export / Signal-Via Model” in BoardSim menu selections. The difference between the two is that while in LineSim the selected via is saved to the .dao file, in BoardSim you have to select the vias using mouse commands ([MOUSE_MOVE](#), [MOUSE_DBLCLICK](#)).

<action> can be the following:

- **OPEN** = Opens the Via model Extractor dialog
- **LOAD, <dao_filename>** = Loads the saved configuration (.dao file)
- **OUT, <touchstone_file>** = Specifies the name of the Touchstone file to be generated. The specified filename is assumed to be relative to the [Output directory](#) and should not begin with a slash.
- **RUN** = Run analysis
- **CLOSE** = Close dialog

Example – Via-model

```
* BoardSim example
LOAD_HYP_FILE, MAX1626.hyp
* select vias only
SET\_SEL\_FILTER, VIA
VIAMODEL_EXTRACTOR, OPEN
VIAMODEL_EXTRACTOR, LOAD, MAX1626_VM.dao

MOUSE_MOVE, 0.08330, 0.05370
MOUSE_DBLCLICK

VIAMODEL_EXTRACTOR, OUT, MAX1626_VM.s1p
VIAMODEL_EXTRACTOR, RUN
VIAMODEL_EXTRACTOR, CLOSE
```

DIFFVIA_EXTRACTOR, <action>

It corresponds to “Export / Model / Signal-Via Model” in LineSim and “Export / Signal-Via Model” in BoardSim menu selections. The difference between the two is that while in LineSim the selected via is saved to the .dao file, in BoardSim you have to select the vias using mouse commands ([MOUSE_MOVE](#), [MOUSE_DBLCLICK](#)).

<action> can be the following:

- **OPEN** = Opens the Via model Extractor dialog
- **LOAD, <dao_filename>** = Loads the saved configuration (.dao file)
- **OUT, <touchstone_file>** = Specifies the name of the Touchstone file to be generated. The specified filename is assumed to be relative to the [Output directory](#) and should not begin with a slash.
- **RUN** = Run analysis
- **CLOSE** = Close dialog

Example – Via-model

```
* BoardSim example
LOAD_HYP_FILE, multi-pin_01.hyp
* select vias only
SET\_SEL\_FILTER, VIA
DIFFVIA_EXTRACTOR, OPEN
DIFFVIA_EXTRACTOR, LOAD, multi-pin-diffvia.dao
MOUSE_MOVE, -0.186, 0.0315
MOUSE_DBLCLICK
MOUSE_MOVE, -0.185, 0.0305
MOUSE_DBLCLICK
DIFFVIA_EXTRACTOR, OUT, multi-pin-diffvia.sNp
DIFFVIA_EXTRACTOR, RUN
DIFFVIA_EXTRACTOR, CLOSE
```

Export Stackup

EXPORT_STACKUP, <file_name.stk>

It corresponds to “Setup / Stackup / Export menu selection and it writes out the stackup details to an .stk file

Export PDN Model

PDN_WIZARD, <action>

It corresponds to “Export / Model / PDN & Channel Model” in LineSim and “Export / PDN Model” in BoardSim menu selections.

<action> can be the following:

- **OPEN** = Opens the PDN Model Extractor dialog
- **LOAD, <dao_filename>** = Loads the saved configuration (.dao file)
- **OUT, <touchstone_filename>** = Specifies the name of the Touchstone file to be generated. The specified filename is assumed to be relative to the [Output directory](#) and should not begin with a slash.
- **RUN** = Run analysis
- **CLOSE** = Close dialog

Example – PDN Model

```
LOAD_HYP_FILE, demoPI.hyp
PDN_WIZARD, OPEN
PDN_WIZARD, LOAD, demoPI-pdn.dao
PDN_WIZARD, OUT, demoPI-pdn.sNp
PDN_WIZARD, RUN
PDN_WIZARD, CLOSE
```

Export Design Archive

EXPORT_TO_ARCHIVE, <action>

It corresponds to the “Export / Design Export” menu selection.

<action> can have the following values:

- **OPEN** = Open Export Design Archive dialog
- **SAVE [, <dirpath>]** = Save to <dirpath> if specified, which can be full path or relative path. The relative path will be the path relative to Output dir.
- **CLOSE**

Miscellaneous Commands

*** <comment>**

Lines that start with the * symbol are comment lines. Comments can't be placed on the same line with commands – the line has to start with * symbol for it to be considered a comment.

SET_SEL_FILTER, <filter>

Sets the selection filter in the board viewer to only the value of <filter>. This command is used for example in the case of Bypass Wizard in BoardSim to set the selection filter to vias only, so we can select a via using the mouse commands.

The value of <filter> can be:

- VIA = select vias only
- PIN = select pins only
- SEG = select segments only
- ALL = select anything

CENTER_VIEW, <cx>, <cy>

Specifies new center of the board viewer (in cm).

ZOOM_VIEW, <dx>, <dy>

Specifies new size of the board viewer (in cm).

ZOOM_VIEW, board

Makes board fit viewer window – corresponds to View/Fit to Window menu entry.

SET_VIEWPORT, <cx>, <cy>, <dx>, <dy>

Sets the center point of the board screen and the size of the visible part of the board (in cm).

MAXIMIZE_TPE

In LineSim maximizes PDN Editor and hides FFS Editor

MOUSE_MOVE, x, y

Moves mouse to the (x,y) position specified in metric coordinates.

MOUSE_DBLCLICK

Double-click the left mouse button at the current location. Use [MOUSE_MOVE](#) to move the mouse to the desired coordinate.

BREAK

Pop out of scripting to HyperLynx

FINI

Terminates the script

PAUSE

Pop up box for User

SLEEP [, N]

Sleep for 1 to 60 seconds. If N is not set it defaults to 5 seconds

TIME_STAMP

The current time is placed in the log file along with the time delta since the previous invocation of this command

TIME_STAMP_SET_FILE, <filename>

Set the name for the time stamping file. <filename> is relative to the [Output directory](#).

TIME_STAMP_TO_FILE [, <action>]

Writes a time stamp and the script line number of this command into the file specified by TIME_STAMP_SET_FILE. The time stamp is a duration (delta time) expressed in seconds.

The timer is zeroed out every time TIME_STAMP_TO_FILE or TIME_STAMP_SET_FILE commands are called.

Optionally specify the <action> parameter, which can be any of the following:

- INITIATE = Zeroes out the counter, but nothing will be written to the file
- <Comment> = Any comment could be specified here and this will be written out to the file on the same line with the duration and Script Line