**Simulation using Nanosim**

There are three candidates for simulation of the Spice-based netlists using Nanosim tool:

* Spice-based Netlist (**\*.sp**) 🡪 Shows the circuit structure.
* Configuration File (**\*.cfg**) 🡪 Contains the simulation and output commands.
* Vector File (**\*.vec**) 🡪 Contains the test vectors (a.k.a. drivers) for input signals.

Notice 1: Depending on the desired output results (e.g. **waveform information** or **standard delay format**), different commands can be utilized inside a configuration file.

Notice 2: There can be different types for a vector file, such as:

* **nsvt** 🡪 Having a fixed time interval between applying different test vectors to

input signals.

* **vec** 🡪 Having different time intervals between applying different test vectors to

input signals.

Notice 3: For more information, refer to the example configuration and vector files.

\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\* **Linux Operating System** \*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*

**1)** Place the related files inside a certain directory.

**2)** Enter the following commands for running simulation and generating output results:

**Command 1**:

nanosim -n Example.sp -c Example\_Option\_1.cfg -nvec Example.vec -out fsdb

**Command 2**:

nanosim -n Example.sp -c Example\_Option\_2.cfg -nvec Example.vec -out out

Notice: Refer to the “**Example\_Run.txt**” file for further information regarding the commands and their arguments.

**3)** Open Synopsys CosmosScope to view the signals:

**Command**: scope

**4)** Open one or more Plotfile(s):

1. File -› Open -› Plotfiles ...
2. Files of type = FSDB (\*.fsdb)
3. Appearance of **Signal Manager Window** and **Plotfile Invocation Window**

Notice 01: **Signal Manager Window** 🡪 It includes all of the opened plotfiles.

Notice 02: **Plotfile Invocation Window** 🡪 It includes the signals of a single plotfile.

**5)** Select all of the signals in the **Plotfile Invocation Windows** and click on **Plot**.

**6)** **Displaying Signals as A Bus**

In the **Graph Window**, select the name of the desired signals, and then right click on them and choose “Create Bus”. A bus shows the transitions of the chosen signals and the value that all of them construct together.

**7)** **Axes Adjustment**

Right click on the **X-Axis** and select “Attributes …”. Then, choose an appropriate range (for the example, **120 ns**) and click on “Apply”.

**8)** Save an image of the graph:

1. File -› Export Image ...
2. Enter the "File name", choose "Postscript (\*.ps,\*.eps)" for "Files of Type" and save the image.

\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\* **Windows Operating System** \*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*\*

**1)** Open **GSview**.

**2)** Choose the appropriate page size from "**Media**".

Example: A2 Page Size

**3)** Go to “File 🡪 Open ...” for opening the "\*.ps" file.

**4)** Go to “File -> Convert ...”.

**5)** Select the related options and save the file as "File\_Name.pdf".

**6)** Open **Briss** application and crop the image from the whole page.