**Simulation using HSPICE**

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

**1)** Place the **HSpice-based netlist file** (i.e. “\*.sp” format) inside a certain directory.

**2)** Enter the following command for running simulation:

**Command**: hspice \*.sp > out.txt

**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 = HSPICE (\*.tr,\*.ac,\*.sw,\*.ft)
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)** 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.