Install and Run Ivory

Step 1: Install Ngspice with shared library version (This is used for dynamic model simulation. If only the static model is used, you can directly skip this step)

1. Install dependency for Ngspice
   sudo apt-get install -y bison
   sudo apt-get install -y flex
   sudo apt-get install libxaw7-dev

2. Download Ngspice and uppack
   https://sourceforge.net/projects/ngspice/files/ng-spice-rework/old-releases/28/ngspice-28.tar.gz/download

   In the directory of ngspice-28.tar.gz
   tar -zxvf ngspice-28.tar.gz

3. Configure and install Ngspice
   cd ngspice-28
   ./configure --with-ngshared --enable-xspice --enable-cider --enable-openmp --disable-debug
   make clean
   sudo make install

4. Install a normal Nspice Configure and install Ngspice
   cd ngspice-28
   ./configure --enable-xspice --enable-cider --enable-openmp --disable-debug
   make clean
   sudo make install

5. Add Ngspice dynamic linked library path
   gedit ~/.bashrc
   Add "*LD_LIBRARY_PATH=/usr/local/lib:$LD_LIBRARY_PATH*" to the file.
   Save and exit

Step 2: Download and compile Ivory:

1. Download and unpack Ivory from Github

2. Make and run Ivory
   cd Ivory-Early-stage-design-space-exploration-tool-for-integrated-voltage-regulators-master
   make clean
   make
   ./main