MAGCAD/ToPoliNano installing instructions

ToPoliNano is a framework and a CAD tool, invented and developed at the Politecnico di Torino. The framework is composed of two separate programs: ToPoliNano and MagCad. Details on these two programs can be found on the official website:

https://topolinano.polito.it

The procedure the download and install MagCAD/ToPoliNano is the following:

- Connect to the official website https://topolinano.polito.it/register/
- Register an account by inserting your personal information and a valid e-mail address;
- Once pressed the "Register" button, an e-mail will be sent to your inbox folder. Activate your account by pressing the link in the e-mail;
- If you didn't receive any e-mail, check your spam folder.
- Go to section Downloads of the website and obtain your free copy of MagCad and ToPoliNano for your OS. The installing should be immediate.

In case of troubles, the documentation can be found here: https://topolinano.polito.it/documentation/

In case you are a Linux Use and are not using a Fedora distribution, but Ubuntu or Debian or Mint distribution you can use the tool Alien to transform the .rpm in a .deb package and then you can install directly the package using *dpkg -i xxx.deb*

MAGCAD test

In order to verify whether the installing procedure has been correct please execute the following steps:

- Open MagCAD
- The window "Drawing settings" should show up, press Close;
- Ignore the possible warning "no technology selected" by pressing Close;
- Chose File ¿ Open, navigate to the directory where you saved the given files and select the file "inml test.qll". The iNML circuit should appear in the main Window, Figure 1.6;
- Select File ¿ Export Component. Do not change anything, just press Export;
- If everything is ok, you should see the message "Component has been correctly exported!", by pressing Close, a second message should show up: "Vhdl generation correctly performed". Press Close;
- Eventually, some *.vhdl files should have been created in your user folder, typi- cally similar to: C:/Users/USERNAME/MagCADFiles/VHDL/inml test/.

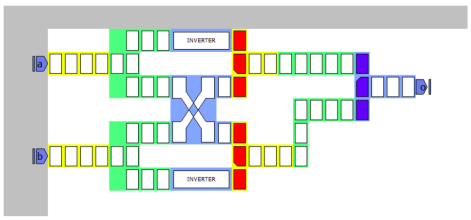

Figure 1.6: MagCAD circuit example

OPTIONAL: Modelsim installing

ModelSim is an environment developed by Mentor Graphics, for simulation of hardware description languages such as VHDL and Verilog. Details on this program can be found on the Intel official website:

https://www.intel.it/content/www/it/it/software/programmable/quartus-prime/model-sim.html
The procedure the download and install ModelSim is the following:

- Connect to the Intel Download Center for FPGAs website

  https://www.intel.com/content/www/us/en/programmable/downloads/download-center.html
- Select **Version 20.1 Lite Edition**
- In the new page check the version and the operating system, then click on the tab **Individual Files** and click on **ModelSim-Intel FPGA Edition**
- If you are not registered, fill the form with your personal data to create an account
- Download the ModelSim program
- Launch the executable file and follow the installation process

OPTIONAL: Modelsim test

In order to verify whether the installing procedure has been correct please execute the following steps:

- Open ModelSim;
- A window named “IMPORTANT Information” might appear, close it by pressing Close;
- Create a new Project with File > New > Project...
- Fill the appeared form by inserting the name “tb test” and the Location you prefer. A new window will appear, select “Add existing file”. Navigate to the folder where you saved the given files and select the file “tb test.vhd”, then press OK.
- The file will appear in the ModelSim window, right-click on it and do Compile > Compile selected.
- Now we have compiled the file, we have to simulate it. Choose from the nav Simulate > Start Simulation.
- The window Start Simulation will appear. Expand the work directory and select the tb test.vhd file. Then press OK
- In the bottom part of the ModelSim interface, there is a console (Transcript). Type “add wave *” and then “run 50 ns”
- Figure 1.7 reports what you should see after a few seconds.

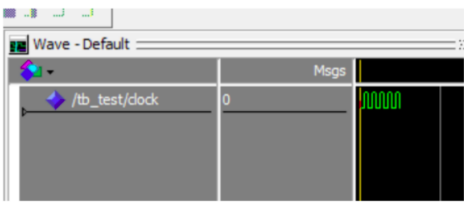

Figure 1.7: Modelsim output example