HotSpot Thermal Simulator

1 HotSpot's Inputs

The thermal model is bundled as a trace-level simulator that takes a power trace file and a floorplan file as inputs and outputs the corresponding transient temperatures onto a temperature trace file. There is also an option to output the final steady state temperatures onto a file. As inputs, HotSpot mostly requires:

- A configuration file ("-c <fileName.cfg>"): This file contains general configuration parameters for the simulation, e.g., thermal conductivity values (for the chip, the heat spreader, the heat sink, etc), the size of the heat sink, the sampling time of the transient simulations, etc. The default configuration file included in the distribution is called "hotspot.config".

- A floorplan file ("-f <fileName.flp>"): This file contains the floorplan of the chip considered for the simulations. Every line in this file represents a unit (or block) in the chip. Each line, separated by tabs, includes: the name of the unit (no spaces), the width of the unit, the height of the unit, the x coordinates of the left-bottom corner of the unit, and the the y coordinates of the left-bottom corner of the unit. The user decides the level of granularity of the floorplan. For example, every line of the floorplan could represent a complete core, and other lines could represent some other big units, like L2 caches. Contrarily, we could have a floorplan file in which each line corresponds to a functional unit of every core in the chip. An example floorplan file included in the distribution is called "ev6.flp". The floorplan can be viewed using a FIG file viewer like 'XFig' and the perl script 'tofig.pl' which converts it into the FIG format. Alternatively, there is also a command to convert a floorplan into a PDF using fig2dev and ps2pdf is, specifically, "tofig.pl floorplan.flp — fig2dev -L ps — ps2pdf - floorplan.pdf".

- A power trace file ("-p <fileName.ptrace>"): The power trace file contains the power information of each unit through time. The first line of every power trace file should include the names of every unit, matching those in the floorplan, separated by tabs. Then, each line in the power trace file contains the power consumption (in Watts) of every unit (according to the corresponding column) during one time instant. Different lines correspond to different time instants. The power trace file has no timing information associated to it. The duration of every time instant in the power trace file is configured in HotSpot's configuration file ("hotspot.config"), specifically, by parameter "-sampling_intvl" (in seconds). An example floorplan file included in the distribution is called "gcc.ptrace". However, power trace files are normally generated by performance and power simulators (e.g., gem5+McPAT, or Sniper+McPAT), which are then parsed in order to match HotSpot's power trace file format.

- An initial temperature file (optional, "-init_file <fileName.init>"): When no initial temperature file is provided, HotSpot assumes that all thermal nodes in its underlying thermal model (there are more nodes in the thermal model than units in the floorplan) have the same initial temperature as indicated in the configuration file by parameter "-init_temp". However, for a chip that was already powered on, this initial temperature is not a representative value of reality. Therefore, HotSpot can read the initial temperatures for all thermal nodes from an initial temperature file. The standard method for generating this file is to first run a "warm-up" simulation, for which we generate an output file with the resulting steady-state temperatures for a given power trace file (note that the initial temperatures have no impact for steady-state computations). This steady-state file can then be renamed and used as the initial temperature file for the next "true" simulation.

2 HotSpot's Outputs

- Steady-state temperatures file (optional, "-steady_file <fileName.steady>"): By using command option "-steady_file <fileName.steady>" HotSpot will output the resulting steady-state temperatures to the indicated file. When the "-steady_file" file command option is not included, HotSpot will output the steady-state temperatures to the screen. Each line in this file will contain the name of the unit (matching the floorplan) and the resulting steady-state temperature (in Kelvin) for the given power trace file. Note that it also contains additional lines with the steady-state temperatures of all other thermal nodes in HotSpot's thermal model (refer to HotSpot's publications for more details). This steady-state file can be renamed and used as initial temperature file for a future simulation. With regards to how the temperatures are computed based on the power trace file, note that if the power trace file contains more than one line, the resulting steady-state temperatures outputted to the file correspond to average temperatures. Because of this reason, if we are interested in computing the steady-state temperatures for having some specfici power consumption throughout the chip, it is good practice to generate steady-state temperature files by using power trace files that have only one line.

- Transient temperatures file (optional, "-o <fileName.ttrace>"): By using command "-o <fileName.ttrace>" HotSpot will output the resulting transient temperatures to the indicated file. The output transient file format is similar to the input power trace file format. That is, every column corresponds to a unit in the floorplan, and every line contains the transient temperatures (in °C) for each unit associated to a time instant. The duration of every time instant in the transient temperature file is configured in HotSpot's configuration file by parameter "-sampling_intvl" (in seconds). If no transient file is indicated, HotSpot will only compute the steady-state temperatures.

3 Using the grid model

The above runs used the default, fast but less accurate "block" thermal model. HotSpot offers the choice of another more accurate but relatively slower model called the ”grid” model (for the computations, not necessarily for the outputs). The ”-model_type grid” command line option switches HotSpot to the grid model. The trade-off between speed and accuracy of the grid model can be controlled by specifying the grid model's resolution through the command line options ”-grid_rows <num>” and ”-grid_cols <num>”. The default grid size is 64x64 (as can be seen from the ”hotspot.config” file). Grid dimensions are restricted to powers of two for algorithmic convenience. In addition to the ”-steady_file <file>” option that provides temperatures at a per-block granularity, the grid model provides an additional means to access the finer grid of steady state temperatures. Since the grid model aggregates the finer grid temperatures into per-block temperatures, HotSpot provides a choice to the user in the mode of aggregation. The user can select the mapping between the grid and block temperatures of the grid model through the command line option ”-grid_map_mode <mode>”. The four mapping modes supported are: ”min”, ”max”, ”avg” and ”center”. The first three options respectively mean that the block's temperature is computed as the minimum, maximum, or average temperature of the grid cells in it. The ”center” option means that the block's temperature is given by the temperature of the grid cell at its center. Finally, in order to have output files that show grid model results, the command line option ”-grid_steady_file <file>” outputs the internal steady-state grid temperatures directly (without aggregating them into per-block temperatures). This can help in learning how temperatures vary within a block. Also, the perl script ”grid_thermal_map.pl” can produce an SVG format color image of these temperatures with a superposed drawing of the floorplan for easy viewing. If native viewer programs (e.g. Adobe SVG viewer) are not available for viewing an SVG file, conversion tools like ”ImageMagick convert” can be used to generate alternate formats.