Introduction To Mentor Graphics EDA Tools

ECE/Comp. Sci. 352 — Digital System Fundamentals

Charles R. Kime — Department of Electrical and Computer Engineering University of Wisconsin – Madison

EDA Tools: Why Use Them and What Will We Be Using?

In this course, we are studying the design of digital systems. The systems which a digital designer typically works on are very complex. Due to this complexity, the design process is typically performed using computer-aided design tools. Since digital systems are implemented by electronic circuits, the broad category of computer-aided design tools employed is referred to as *electronic design automation* tools or EDA tools.

In this course, we will be using a small subset of the EDA tools available from one of the major EDA tool vendors, Mentor Graphics. These tools are available on Sun and Hewlett-Packard Unix workstations at Computer-Aided Engineering in the College of Engineering. They are also available on Unix workstations in Computer Sciences, but we do not support the use of the tools on these stations. To use these tools at CAE, you must execute newuser for Unix workstations. In addition, if you have not had Unix before, you are required to attend a CAE Unix tutorial early in the semester.

Mentor Graphics Tools for ECE/CompSci 352

Among the broad spectrum of Mentor Graphics EDA tools available we will use just three: Design Architect, Quicksim, and Design Manager. The entry on the command line used to launch each of the tools follows the tool name in parentheses.

Design Architect

Design Architect (da) is used for drawing logic diagrams otherwise referred to as schematics. A *schematic* is made up of components or modules and interconnecting wires.

Bundles of wires that are related and referred to using a single name are called *buses*. In drawing a schematic, the components which represent various elements are selected from a component *library*. A component is added from the library to the schematic. After several components had been added, wires are drawn to connect the components together to form a logic diagram.

Quicksim

After a schematic has been completed, it is necessary to determine the correctness of the schematic. Depending on the methods used, this process is called *validation* or *verification*. We will be using a process called simulation for validation. *Simulation* consists of applying inputs to models of the circuit generated from the schematic and comparison of the output results to the correct outputs. The Mentor Graphics tool used for simulation is called Quicksim (quicksim). After schematic has been drawn, Quicksim is invoked on it. The signals which are to be inputs can be selected on a copy of the schematic available in a window within Quicksim. We can also select the signals which are to be outputs. Next, we enter a sequence of binary values called forces to be applied to the inputs. Once a sequence has been entered, we then run the simulator and both the inputs applied and the resulting outputs appear in a waveform window or in an output listing window. We can then compare the inputs and resulting outputs observed with the expected inputs and outputs. If they agree and we have applied inputs that are adequate to fully validate the circuit, then we can conclude that the circuit is functioning correctly. If the simulation outputs disagree with the outputs we expect, then we must correct the logic schematic or design. By examining the outputs that appear and determining the errors that occurred, we are often able to pinpoint the problems in the circuit.

One of most difficult problems in validation is to determine an adequate input sequence or set of input sequences to apply. For combinational logic circuits with a small number of inputs, it is possible to apply all possible input combinations for validation. This allows us to produce a listing or waveforms that represent the truth table of the circuit. For combinational circuits with a larger number of inputs, we must select a subset of combinations that constitutes a good validation sequence. This can often be a challenging process and, if done incorrectly, the validation will be incomplete.

Design Manager

Complex EDA tools are supported by specific databases that describe the components, circuits and their models, as well as inputs, outputs, and internal signals. Mentor Graphics employs a database referred to as the Falcon Framework. The overall suite of tools including the Falcon framework is based on object-oriented principles. Also, EDA tools typically provide a complex system that keeps track of versions of designs and timestamps. As a consequence, operations such as copying, moving and deleting objects cannot be performed by using simple command line or GUI operations on files. Instead, it is necessary to use a tool that is specifically designed to preserve and manipulate the objects. The Mentor Graphics tool for this purpose is Design Manager (dmgr). In addition Design Manager can be used to select the design to work on and to launch other tools such as Design Architect and Quicksim. Ordinarily, you will not use Design Manager, but it is helpful for cleaning things up and if you get into problems related to your Mentor Graphics files.