REVISION			
REV	**DESCRIPTION**		**DATE**
1.1	Initial Revision		06.10.2005

DIGITAL CIRCUIT SIMULATOR

Project Proposal

Number of Pages								
Doc	**Appendixes**							
12	A	B	C	D	E	F	G	H
	–	–	–	–	–	–	–	–

STARSOFT
Middle East Technical University
06531 ODTÜ, Ankara-TÜRKIYE

Controlled Folder:					
TBD	**A4**				1

PREPARED BY			
NAME	**TITLE/ROLE**	**REV**	**SIGNATURE**
Elif SAYGI	Quality Engineer	1.1	
Halit DEVELIOGLU	Project Manager	1.1	
Serhat KOYULMUS	Software Developer	1.1	
Seniz SOZER	Configuration Manager	1.1	
Volkan ICEL	Software Developer	1.1	

CONTRIBUTIONS			
NAME	**TITLE/ROLE**	**REV**	**SIGNATURE**
Serkan BAL	Advisor	1.1	

CHECKED BY			
NAME	**TITLE/ROLE**	**REV**	**SIGNATURE**
Serkan BAL	Advisor	1.1	
Elif SAYGI	Quality Engineer	1.1	
Volkan ICEL	Software Developer	1.1	

TABLE OF CONTENTS

1. PROJECT TEAM

1.1. PROJECT MEMBERS

Elif SAYGI

Halit DEVELIOGLU

Serhat KOYULMUS

Seniz SOZER

Volkan ICEL

1.2. MEMBER ROLES AND RESPONSIBILITIES

Project Manager   **:** Halit DEVELIOGLU

Software Developers   **:** Serhat KOYULMUS, Volkan ICEL

Quality Engineer   **:** Elif SAYGI

Configuration Manager **:** Seniz SOZER

2. PROJECT

2.1. PROJECT TITLE

Our project title has been decided as "DigiArt". For this decision, each group member suggested lots of name options and this name is decided after lots of successive eliminations which are done by democratic voting.

2.2. PROJECT DESCRIPTION

DIGSIM project is implementation of a graphical tool which is used for editing digital electronics circuits and executing them. Graphical User Interface of DIGSIM enables users to design and edit combinatorial and sequential digital circuits easily. Validity of the designed circuit is checked and its operation is tested. All digital circuit elements are ready to use in our DIGSIM project library with their actual pin layouts.

2.3. PROJECT DOMAIN ANALYSES

There are many Digital Circuit Design and Simulation Tools which are similar to DigitalArchitect. We have investigated some of them such as Xilink, Tabspice, Logic Works, SimPlorer, SimDL, etc. The following paragraphs describes the basic features of them.

TopSPICE v7.0

TopSPICE is one of the price/performance leaders in true analog/digital/behavioral mixed-mode simulation for the PC. TopSPICE offers an advanced simulator in its price range, compatibility, and a quick and easy to use integrated design environment from schematic capture to graphical waveform analysis.

TopSPICE offers both an easy to use interface for beginners and maximum flexibility for expert SPICE users. Most simulation analysis commands can be specified by selecting menu options. Experienced SPICE users also have the option of entering commands in text form directly on the schematic drawing or a separate command file.
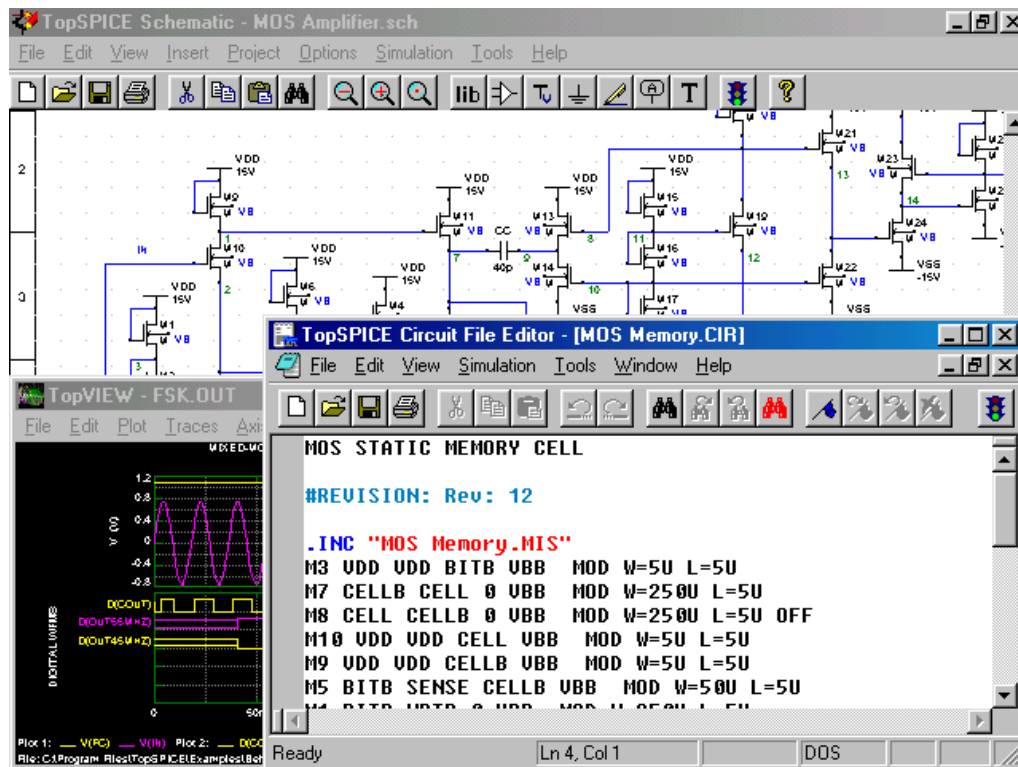

A snapshot for TopSPICE

Built-in event-driven logic simulator; SPICE like syntax for logic elements; nine-state logic simulation; separate analog and digital time step control; no limitations in analog/digital connections; user specified gate delays; user specified A/D interface models; glitch and race condition detection; digital state and waveform output options.

DEEDS (Digital Electronics Education and Design Suite)

Digital Electronics Education and Design Suite is a set of educational tools for Digital Electronics. Its approach is characterised by the "learn-by-doing" concept. It covers combinational logic networks, sequential logic networks, finite state machine design, micro-computer interfacing, micro-computer programming.
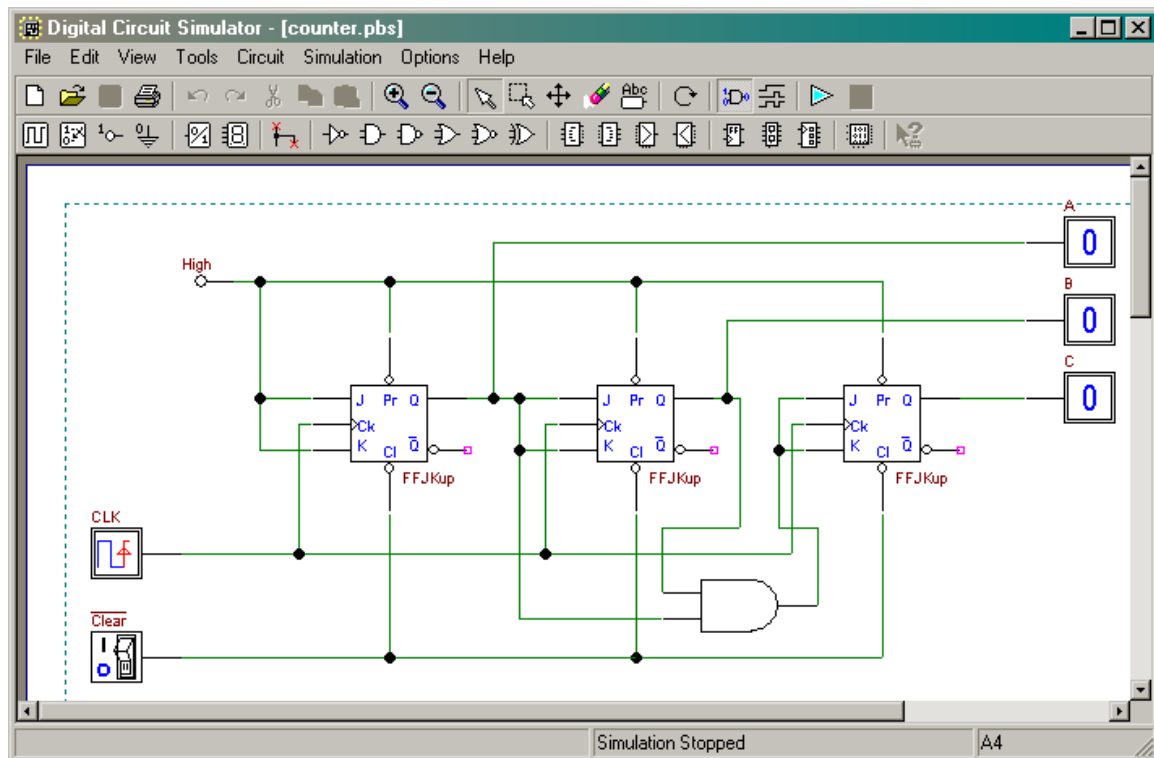
A snapshot for DEEDS

Digital Simulator has a toolbar of digital circuit elements, including logic gates, flip-flops, switches, and indicators. Drawing a circuit with Digital Simulator is like using a paint program. You click the element's icon on the toolbar, then click where you want the element to go. You use a similar procedure to draw wires and indicate connections.

It has emulation of wide range of devices, logic analyzer, hypertext online help and limitable memory and disk space usage.

DLSim

There are two digital circuit simulation tools developed by the company SourceFORGE; DLSim and Digital Simulator. DLSim is a java based digital logic simulator, designed to be used as an educational tool. Uses a simple GUI to allow users to quickly and easily create and visualise circuits from simple components. By this tool users are allowed to create custom components besides known gates such as AND, OR, NOT. Here, intended audience is end users, namely desktops. Since it is developed by JAVA, it is operating system independent. It has the ability of save and load file options with its own file format with extension "dsim". But it has no printing property. This is a simple tool with a very simple GUI and with reduced functionality.
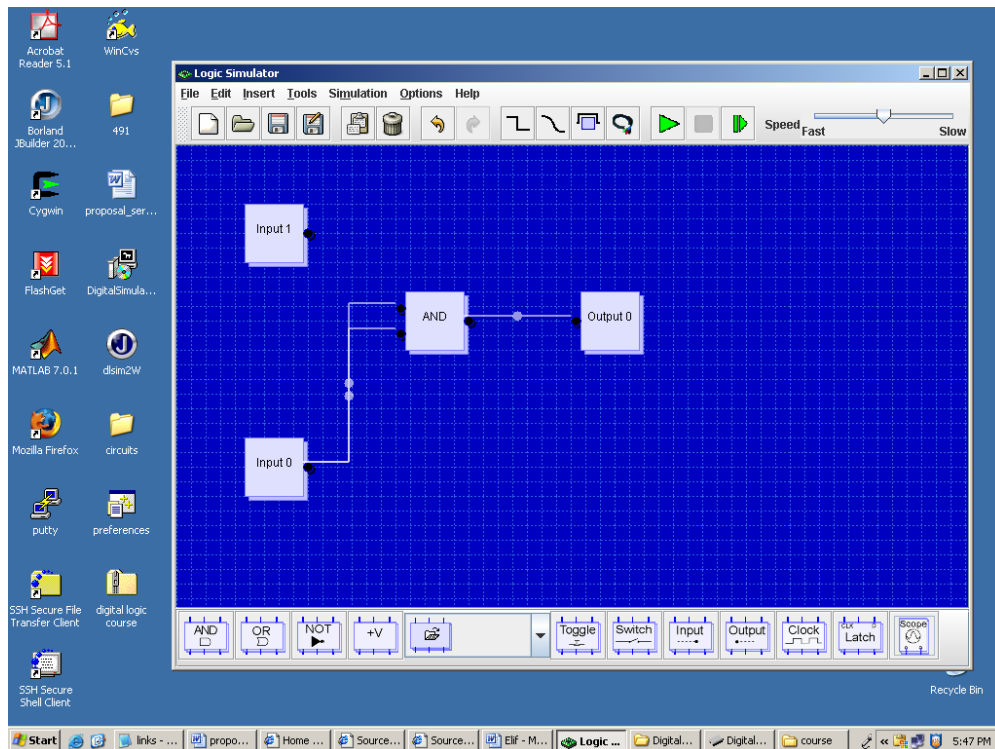
A screenshot for DLSim

Digital Simulator

The second one Digital Simulator is a more featured tool that allows users to design, simulate and output your digital circuit board designs. It is developed by Java and C++ and runs on Windows systems. This tool also has save/load property with the same file extension "dsim" as DLSim. But additionally, Digital Simulator has printing functionality. It has a simple and functional GUI. As an additional property with respect to other simulation tools, you can document the circuits you design with the specified format.
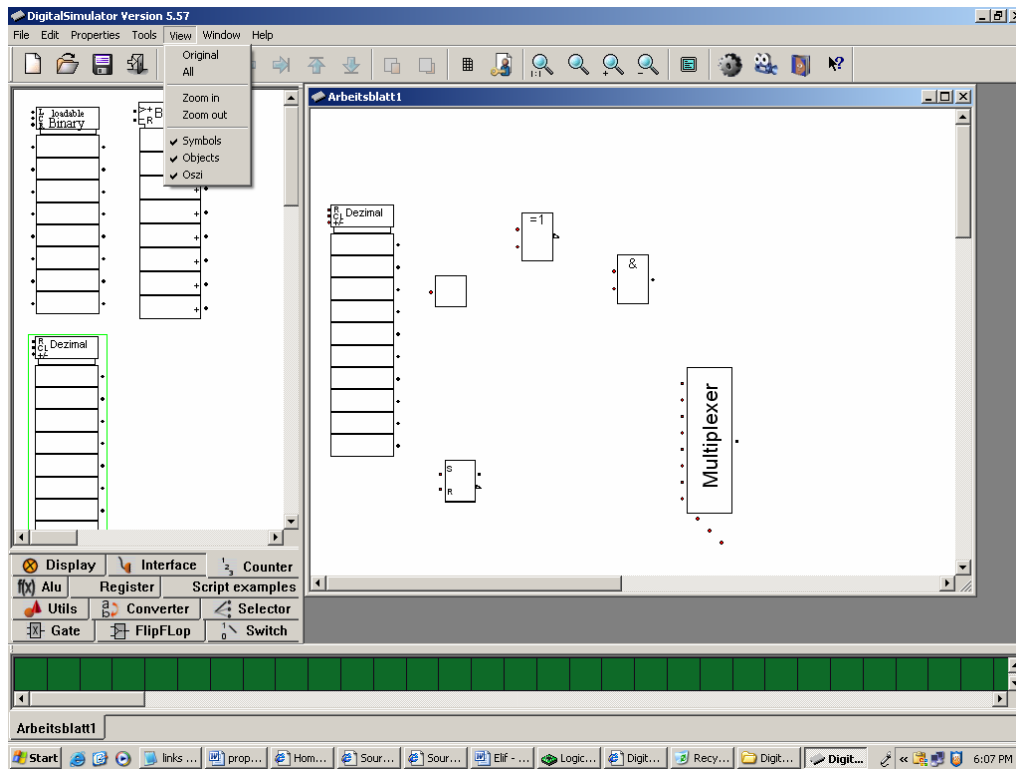
A snapshot for Digital Simulator

The Chipmunk System

The Chipmunk system is a collection of software tools for Unix systems and OS/2.

The software tools in the Chipmunk system perform a wide variety of tasks: electronic circuit simulation and schematic capture, graphics editing, and curve plotting, to name a few.

The Log System:

Log is a circuit schematic capture tool and simulation environment, written by Dave Gillespie. Highlights of the Log system include:

- Schematic capture

- Netlist generation

- Simulation data visualization

- Analog circuit simulation

- Digital circuit simulation

- Custom gate creation

View:

View is a data manipulation and plotting tool written by Dave Gillespie. View is primarily useful for processing and displaying large collections of one-dimensional data curves. Volume and surface visualizations are not supported. Programs in View language can be interactively entered at a shell prompt, or interpreted as batch files. Features of View include

- Data manipulation

- Data plotting

- Libraries

- Compatability

- Instrument Control

Until:

Until is a graphics editor written by Glen Gribble, incorporating code from an earlier graphics editor written by John Wawryznek and Telle Witney. The primary purpose of the tool is to generate publication-quality graphics: all of the figures in Carver Mead's book Analog VLSI and Neural Systems were drawn in Until, and imported into LaTex. Features of Until include

- Many drawing modes

- Input compatibility

- Output compatibility

- Edit-extend mode

Wol and Wolcomp:

Wol is a graphical environment for IC mask layout. Wolcomp is a simple cell compiler, embedded in C, for the compilation of leaf cells generated by Wol into a complete chip. The original versions of Wol and Wolcomp were written by Carver Mead; the version included in the Chipmunk package was primarily written by Massimo Sivilotti, with major contributions by Glenn Gribble. The name Wol originates from the children's book Winnie the Pooh. Features of Wol and Wolcomp include

- Cell editing

- Cell composition

- Netlist extraction

- Rudimentary DRC

	DigArt Project Proposal	**DA001**	**1.1**

Micro-Cap 8

Micro-Cap 8 is an integrated schematic editor and mixed analog/digital simulator that provides an interactive sketch and simulate environment for electronics engineers. Since its original release in 1982, Micro-Cap has been steadily expanded and improved. Micro-Cap 8, the eighth generation, blends a modern, intuitive interface with robust numerical algorithms to produce unparalleled levels of simulation power and ease of use. Nothing else comes close.
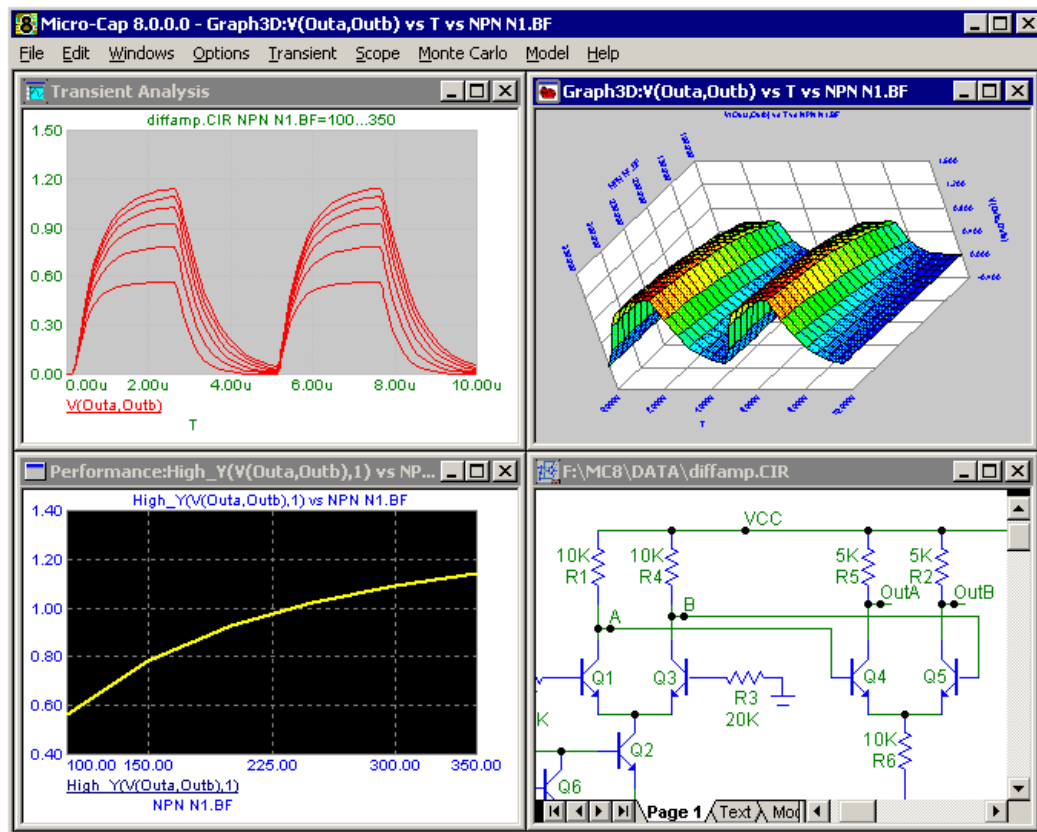

A snapshot for Micro-Cap 8

The graphical, user-friendly interface is simple to learn and use. Familiar SPICE models, plus extensions, are easy to apply. Over 500 warnings and messages help you through problems, when the error occurs, not later in a text file. Micro-Cap 8 plots waveforms during the run, not after the run, when it is too late to change critical values.

DigSim

DigSim is a Java applet that simulates simple digital circuits.There are several advantages of DigSim,Its features are:

- schematic editor,

- animated simulator

- logic analyzer

- save/restore

DigSim is free. The wires are colored to show which ones are high (red), which ones are low (green) and which ones are disconnected (gray). Digsim is useful for both learning digital logic and for testing advanced circuit ideas. When DigSim starts up, a palette of circuit elements and an untitled circuit window appear. You simply drag any of the elements over to a circuit window to add them to the circuit.
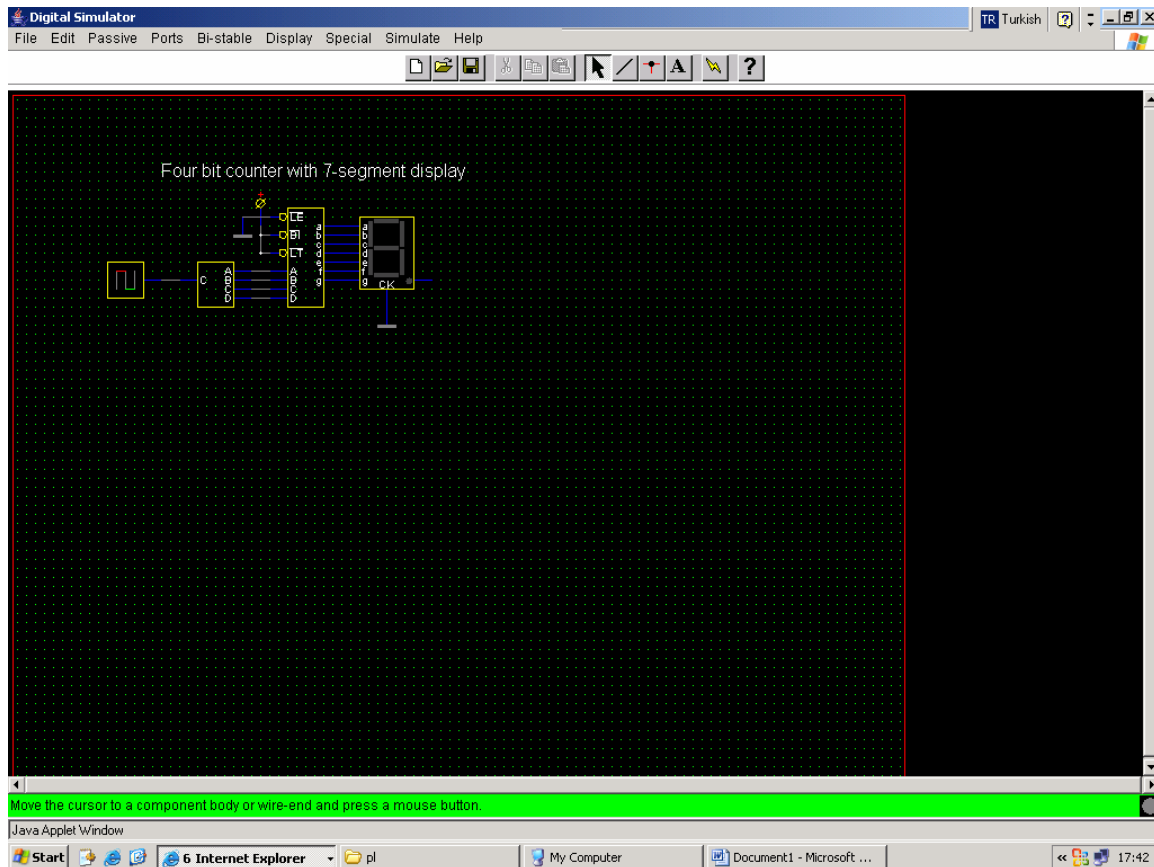

A snapshot for DigSim

The analysis of the demo version shows that it is user friendly when compared to Diglog that we have used at the second year.Its most important feature is the simulator which we think to add in our software.By that we can easily see if the circuit is implemented correctly instead of arranging trying all the inputs manually and see the output as it is done in diglog.The product implements the latches,ports,flip-flops,buffer,inverter,decoder,oscillators.The product also has logic analyzer in its simulate menu.

B2LOGIC

B^{2}Logic is a high-level circuit schematic tool that allows the user to 'drag and drop' logic gates to form complex digital circuits. The figure below shows a screenshot of a sample circuit in B^{2}Logic.
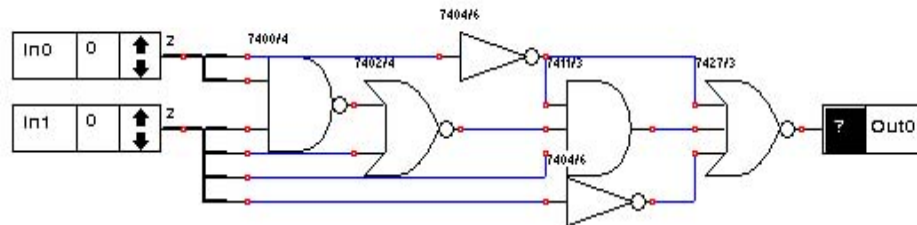
B^{2}Logic outputs a netlist file that is unique to each circuit design. This will allow easy identification and parsing of the specific parts and connections used in each design.

2.4. PROJECT FEATURES

- Full Cut, Copy and Paste of any group of circuit elements.

- Automatically detects signal line connections.

- Place your own text notations on the diagram.

- Automatically reroutes lines at right angles when you move symbols.

- Very simple diagram editing: most symbol placement and wire editing operations do not even require a menu selection.

- Special power and ground symbols automatically create connections and generate the correct simulation values.

- Pop-up menus give you direct access to common commands from anywhere on the diagram

- Easy to use point-and-click user interface.

- Create hyperlinks to move about in circuit or load other files.

- Diglog-like save file format.

- Control through the GUI.

- Suitable for simulation at gate or register transfer level.

- Single-step and clock-step simulator control.

- Basic gates (AND, OR, etc.)

- Alu components (Adders, shifters, multipliers)

- Memory elements

- Hierarchical design through user defined modules