

# Mentor Graphics Quicksim II Tutorial

---

## Table of Contents:

1. Introduction
  2. Preparation
  3. Starting QuickSimII
  4. Simulation Procedure
    - o 4-1. Add/Edit Input Signal
    - o 4-2. Trace Output/Node Waveform
    - o 4-3. Run It !
    - o 4-4. Run simulation with assigned gate delay
    - o 4-5. Keep simulation setup
  5. Print the results
  6. Save the results and Exit
- 

## 1. Introduction

The Quicksim logic simulator (Gate Level) is a computer program that allows you to test a "software breadboard" of a digital hardware design. This tutorial illustrates the simulation procedure and is based on the 'full adder' schematic created in Mentor Graphics Design-Architect.

---

## 2. Preparation

Before you run Quicksim II, you need to create your schematic on Design-Architect and make sure that there are no errors in your design by executing

**Check -> Sheet**

in Design\_Architect. After correcting the errors save your schematic.

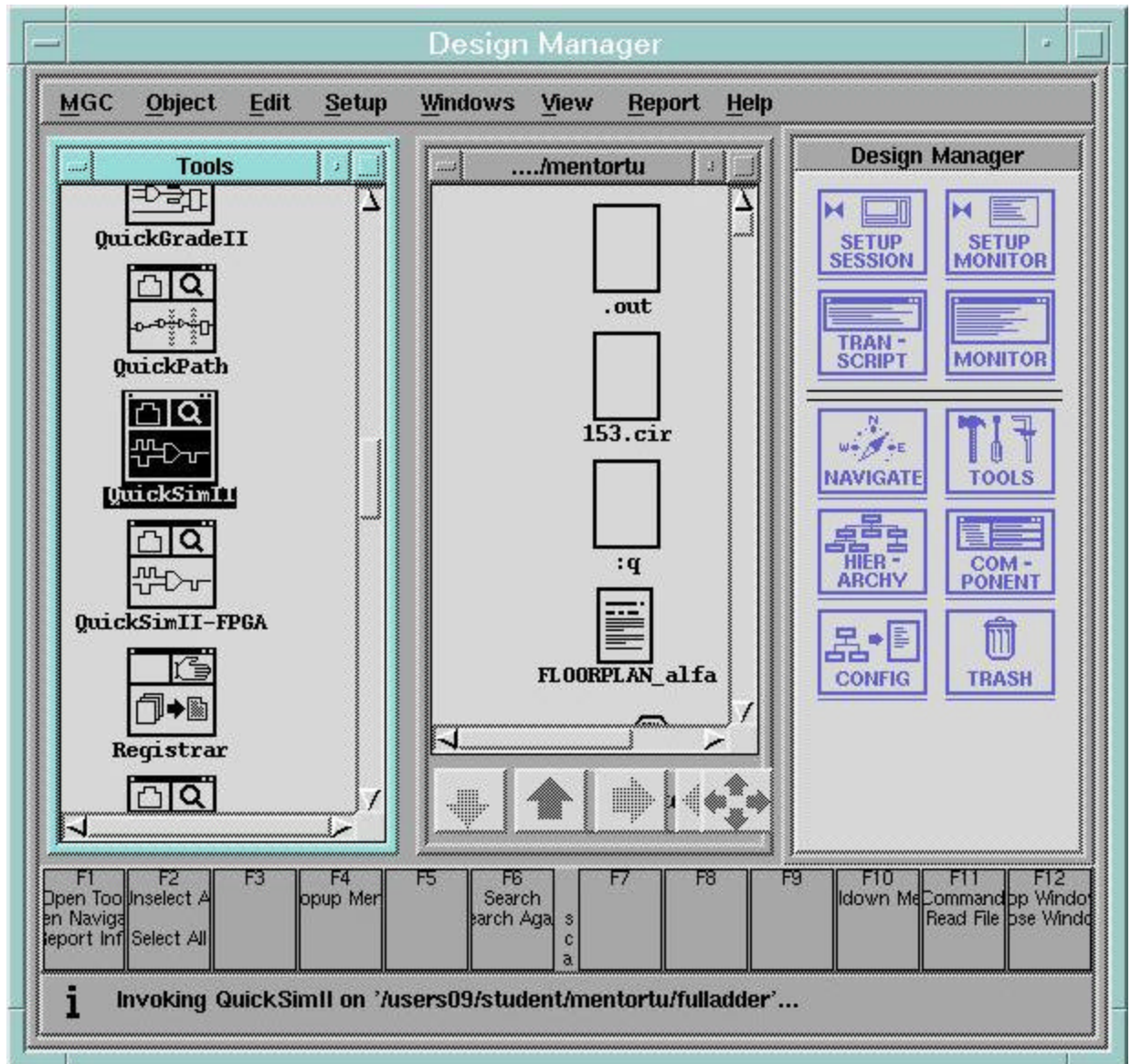
---

## 3. Starting QuickSimII

1. Invoke 'Design Manger' by typing at the shell prompt:

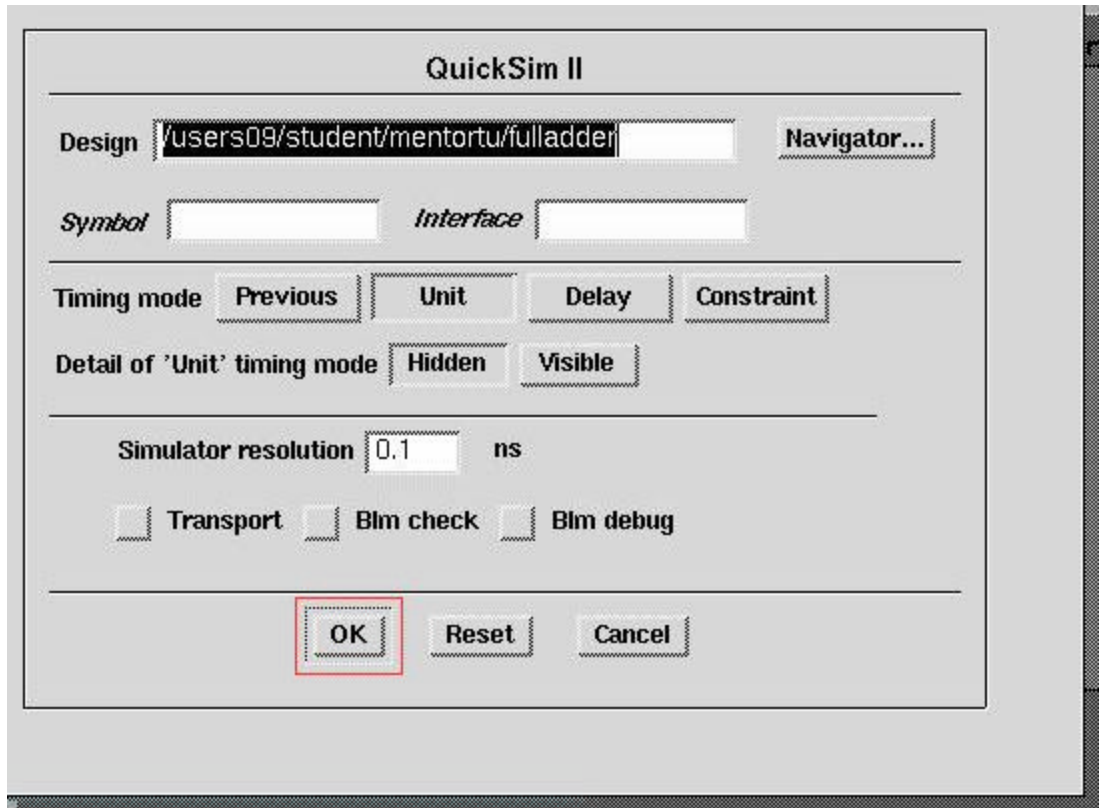
**\$ dmgr**

2. In the 'Tools' window as shown in Fig 3-1, double click the 'QuickSimII' icon to invoke Quicksim.



**Fig 3-1. Start QuickSimII from Design Manager**

3. QuickSimII dialog box will appear as be shown as in Fig 3-2. In the **Design** field, enter the path including the name of schematic or click the Navigator to select the schematic file you have created in Design Architect. Then click on **OK**



**Fig 3-2. QuickSimII Dialog Box**

**Note:** Another way to invoke **Quicksim II** is to type at the shell prompt

```
$ quicksim <design name>
```

4. Click on the '**OPEN SHEET**' icon in the '**Setup**' menu window, the schematic of your design will appear. The example we are using in this tutorial is a 1-bit full adder which is shown in Fig 3-3.

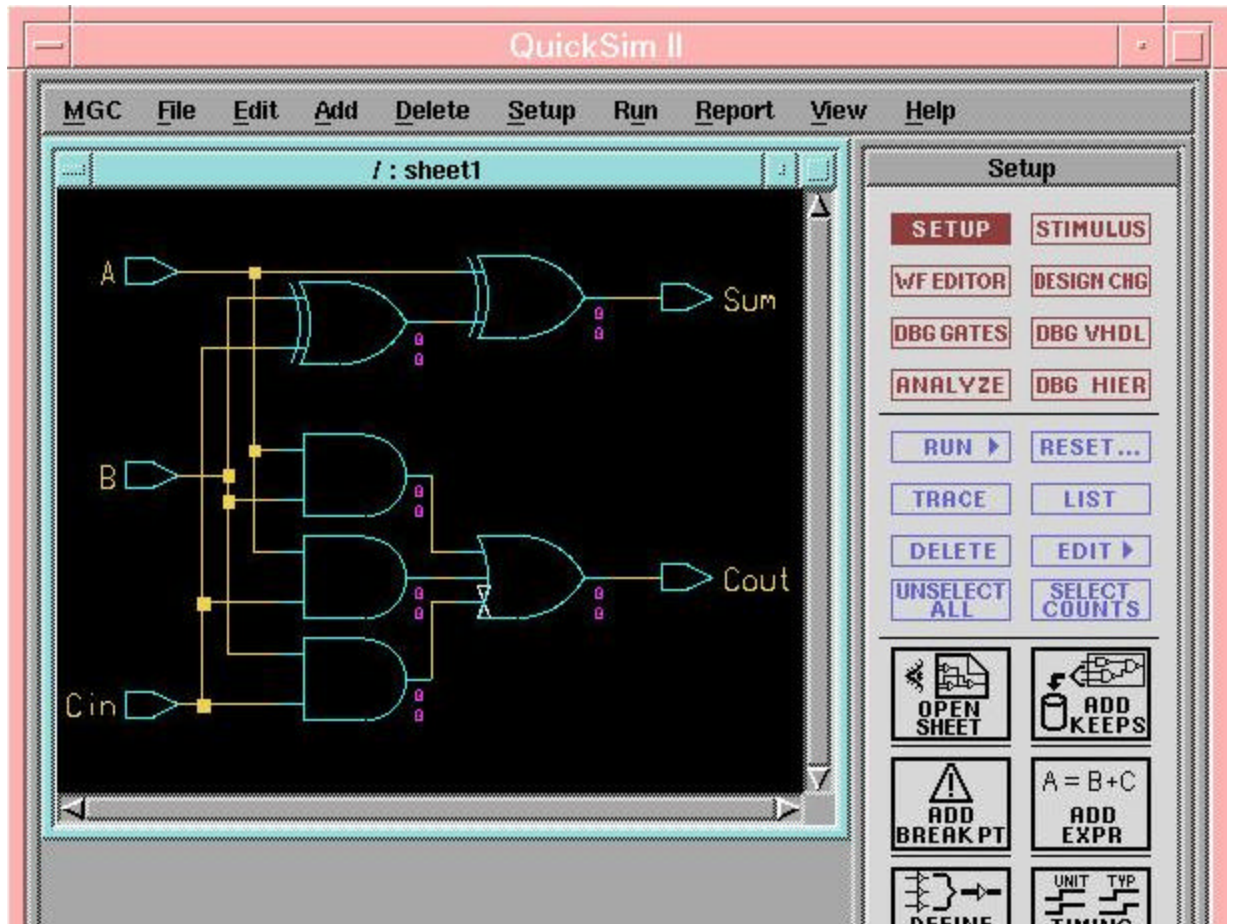


Fig 3-3. Open your design in QuickSimII

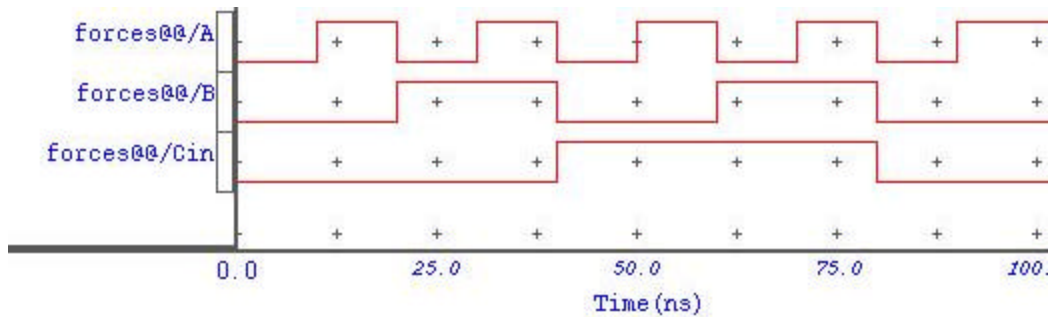
---

## 4. Simulation Procedure

---

### 4-1. Add/Edit Input Signal

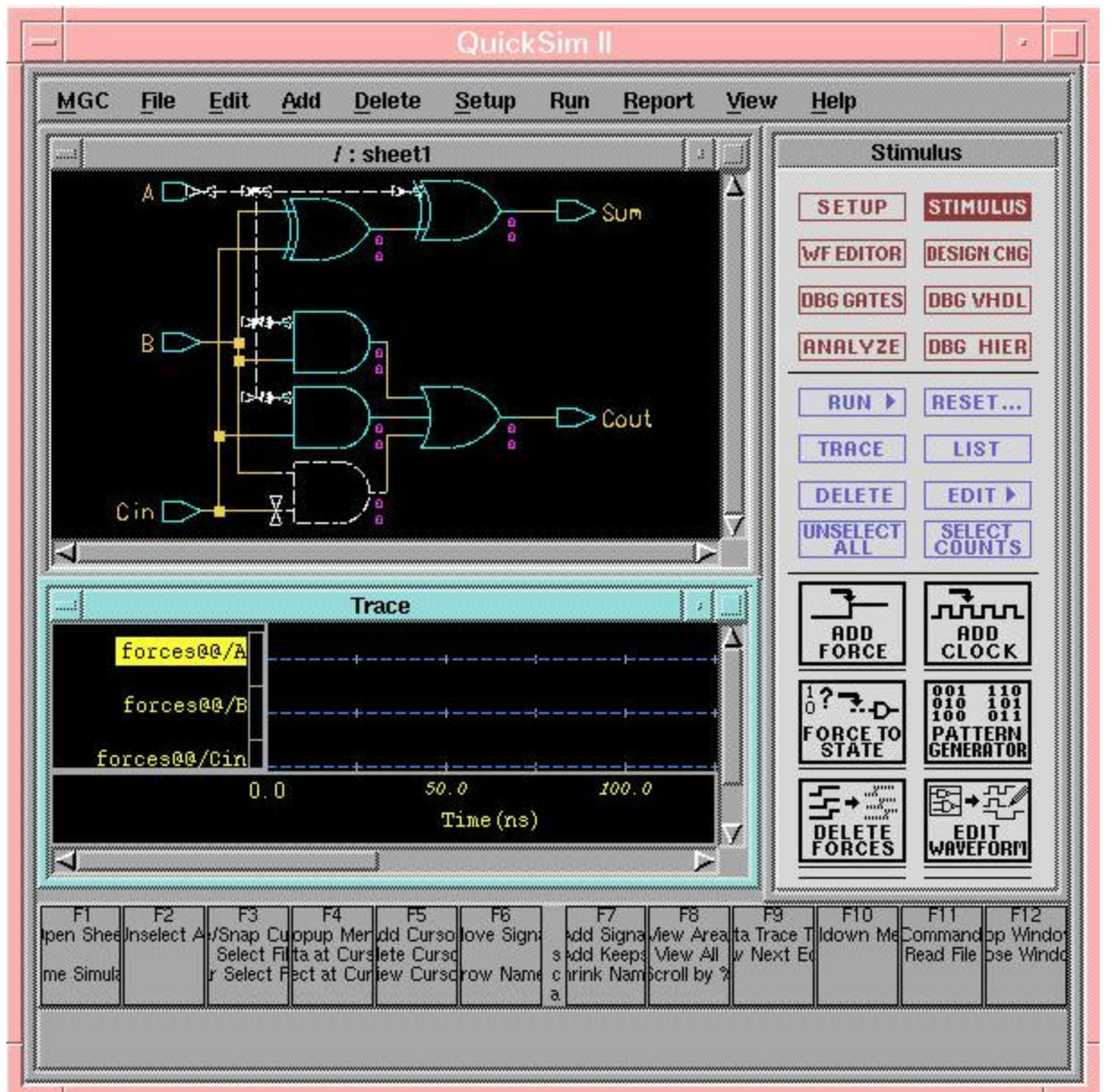
The input signals that we will use to test the design are shown in Fig 4-1:



**Fig 4-1. Input signals**

Steps for editing these input signals are listed below:

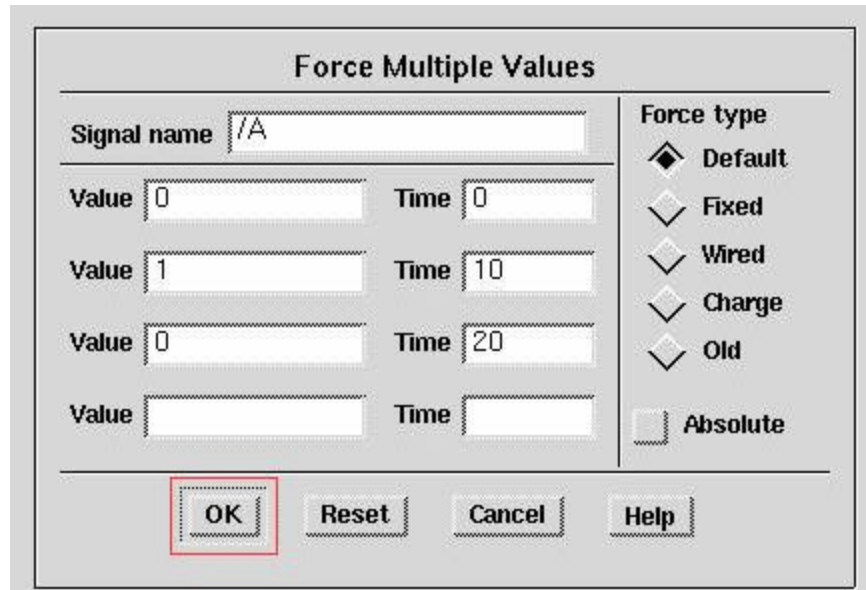
1. Select all '**portin**' symbols in your schematic . In our example, they are A,B and Cin.
2. Select '**WF EDITOR**' in Setup palette.Then select '**EDIT WAVEFORM**' in Waveform Editor palette.
3. A trace window will appear as shown in Fig 4-2. Please size and position the windows such that you can see both, the schematic and wave trace.
4. Go to '**Trace**' window. Unselect all selected symbols by clicking on '**UNSELECT ALL**' in the edit palette.
5. Select '**STIMULUS**' from the edit palette.
6. Go to 'Trace' window. Select one port at a time. The selected port name will reverse in yellow. (Signal 'A' is selected here and is shown in reverse video in Fig 4-2)



**Fig 4-2. Selected port will reverse in color**

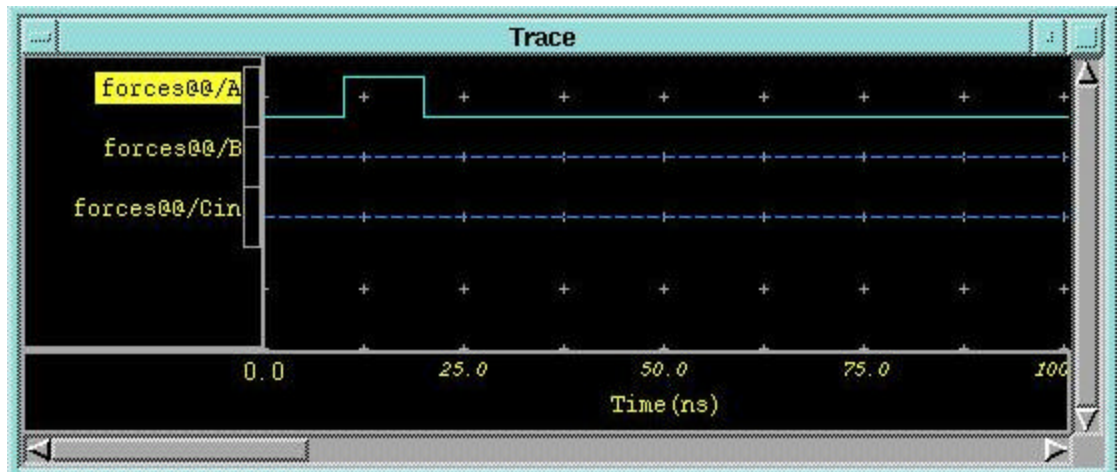
There are several ways to add or edit input signals:

- Select 'ADD FORCE' icon, then input the values '0' or '1' at certain time specified. In this example we specify the input signal as indicated in Fig 4-3:



**Fig 4-3. specify the first period of input signal A**

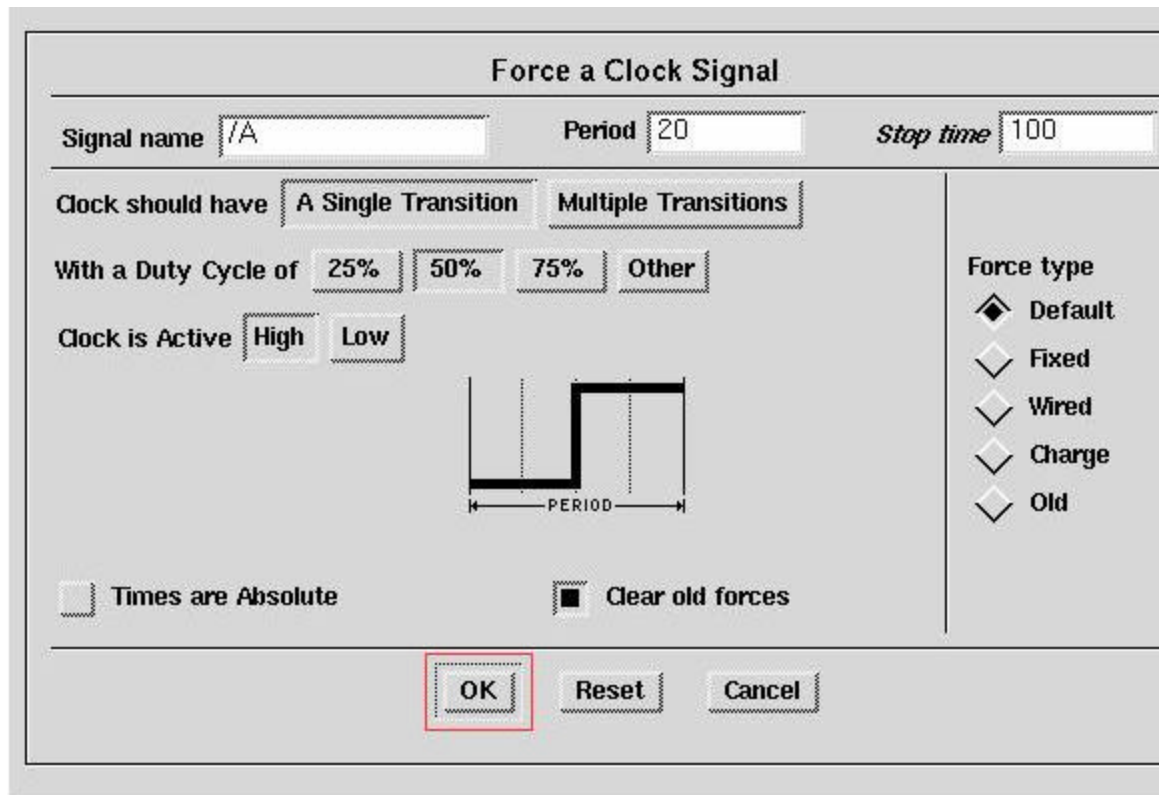
The values given for signal 'A' will generate a waveform shown in Fig 4-4:



**Fig 4-4. specify the first period of input signal A**

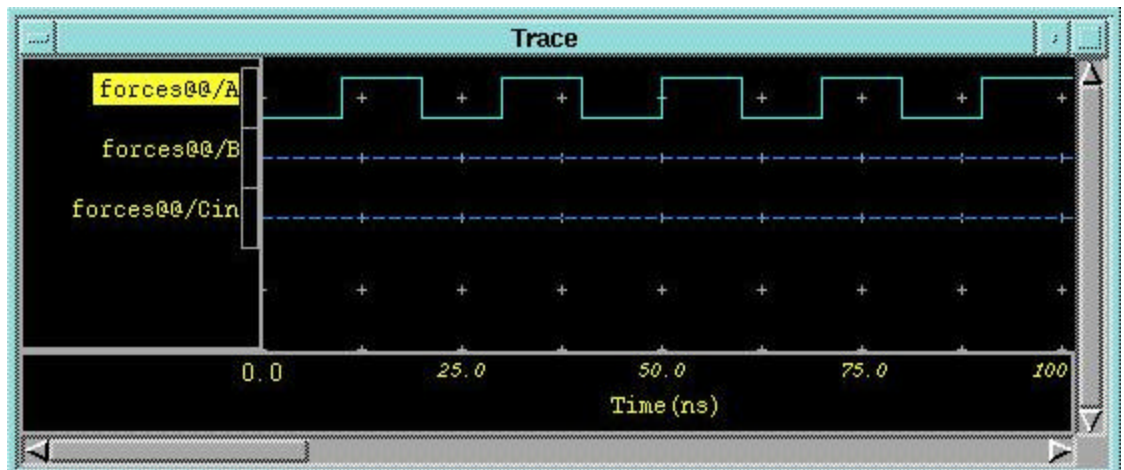
- Another way is to specify the input signal is as a 'clock' signal. This method is very convenient especially when the input signal is periodic. To specify the input signal as a clock follow the steps given below:

Select 'STIMULUS' from the edit palette. Select 'ADD CLOCK' icon to setup the clock. Enter the '20' for period and '100' for stop time in the dialog box shown below. Then click 'OK'



**Fig 4-5. ADD CLOCK dialog box for signal 'A'**

Signal 'A' will have the waveform shown below:



**Fig 4-6. Waveform for signal 'A'**

7. Repeat the steps above to get all the other input signals.

---

## 4-2. Trace Output/Node Waveform

1. Select the outputs in the schematic. Here we select **Sum & Cout** in this example.
  2. Click '**TRACE**' in the '**command window**'. (Now, all the ports are listed)
- 

## 4-3. Run It!

1. Select '**RUN**' in the right hand side palette. Select '**For Time**' in the pop up menu and then specify a time interval for running. Then click '**OK**'. The output waveform will be traced as shown in Fig 4-7.

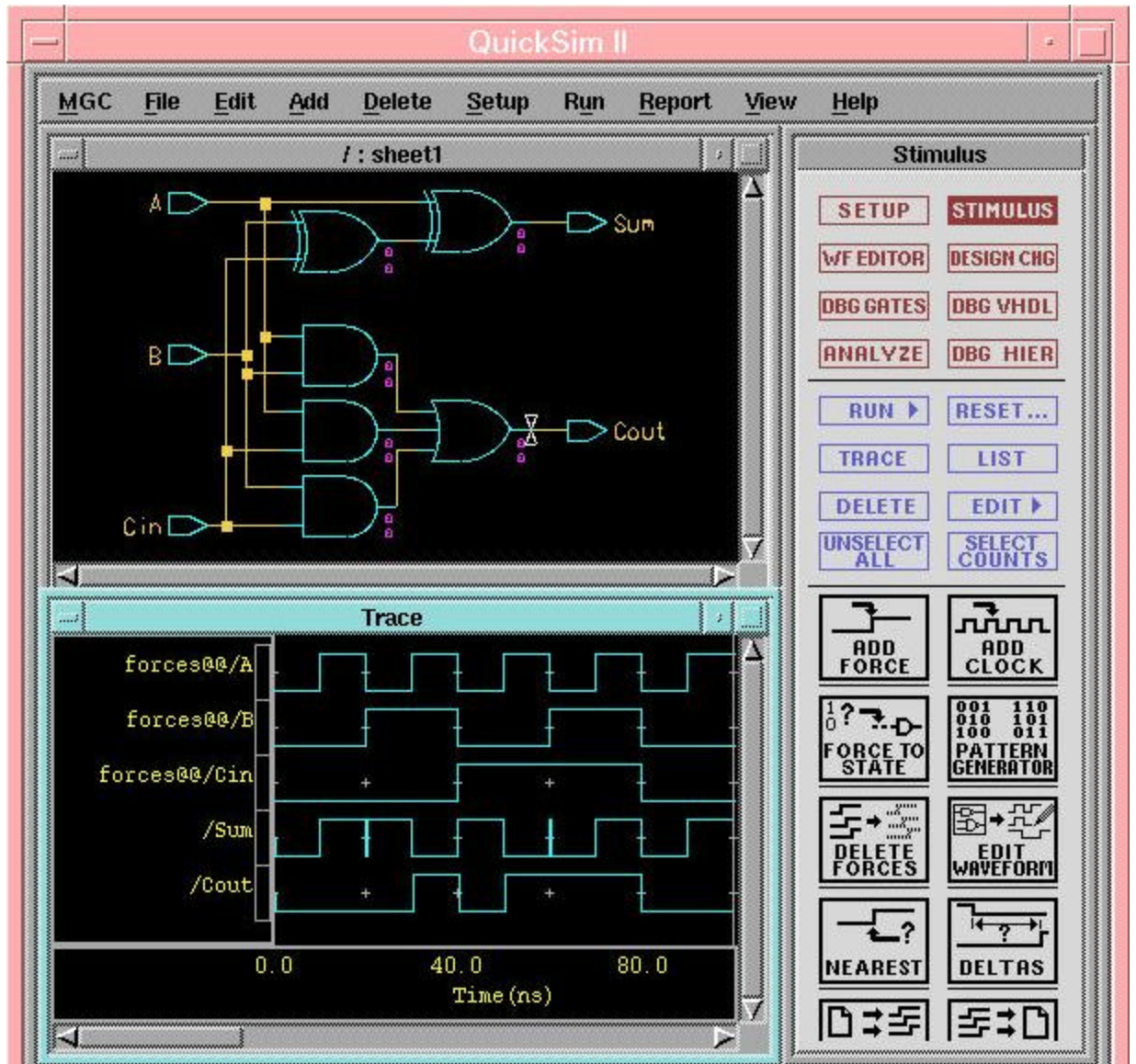
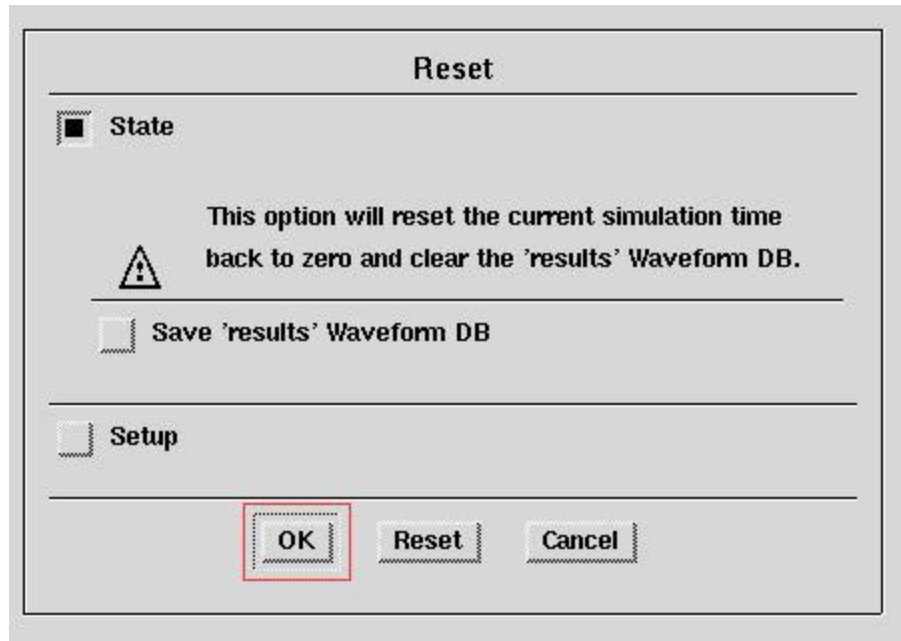


Fig 4-7. Run the design simulation 'A'

- To simulate another input waveform, you have to first reset the simulator to time zero by clicking on '**RESET...**' in the simulation palette. Select only '**State**' and disable the '**Save results Waveform DB**' in the dialog box shown in Fig 4-8, then click '**OK**'.



**Fig 4-8. Restart the simulation from time '0'**

3. Click 'LIST' in the simulation palette to get output data in text form after the simulation is done.

---

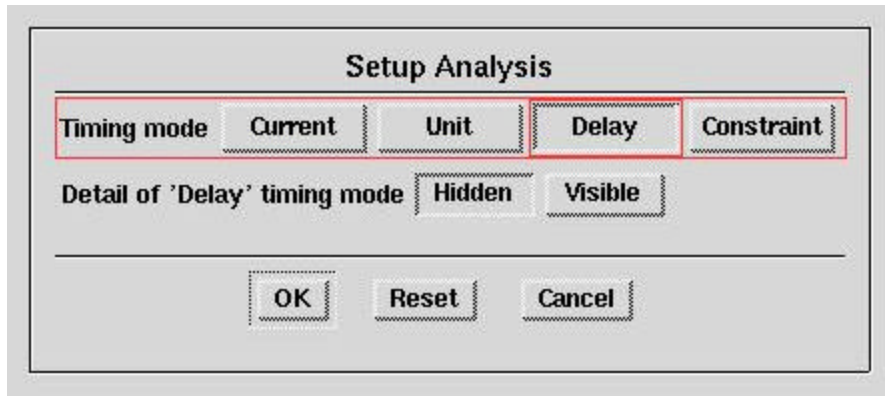
#### **4-4. Run simulation with assigned gate delay**

Sometimes you may want to simulate your design with the gate delays you assigned in '**Design Architect**'. (Please refer to [a quick-guide on how to set up a delay for gates/FFs.](#))

1. From the main pull down menu bar, select:

**Setup -> Kernal**

An '**Analysis**' dialog box will appear as shown in Fig 4-9. Click on '**Delay**' for timing mode. Then click '**OK**'



**Fig 4-9. Select timing analysis with gate delay**

2. Run simulation as explained in section 4-3. **Run it!**

---

#### **4-5. Keep simulation setup**

To save simulation setup means to save the whole environment settings, inputs, trace lists ...etc. so that it can be used whenever required . This is quite convenient when you repeatedly go back and forth between Design\_Architect and QuickSim for minor design changes.

1. To save setup, select from the main menu bar:

**File -> Save -> Setup**

A '**Save Setup**' dialog box will appear. Give a unique name for this environment setup or click on '**Navigator**' to place to another subdirectory. Click the '**Replace**' button, then Click on '**OK**'

2. Next time, when you get into QuickSim, just restore the setup file you have saved. To restore setup, select from the main menu bar:

**File -> Restore -> Setup**

A '**Restore Setup**' dialog box will appear. Enter the setup file name with complete path name, or click on '**Navigator**' to locate your setup file. Select '**Restore setup without confirmation**', then click on '**OK**'.

---

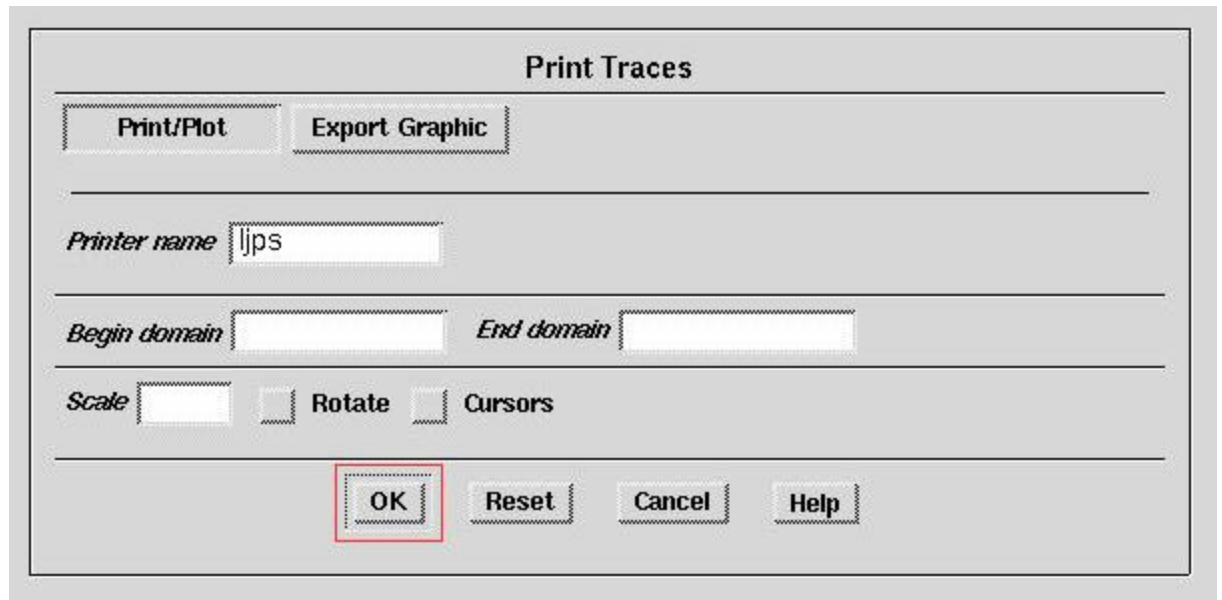
## 5. Print out the results

1. Make sure the window you want to print is the 'active' one. (Simply click on that window in any place to make that window 'active'.)
2. From the main pull down menu bar, select:

**File -> Print**

A dialog box will pop up as shown in Fig 5-1. Enter '**laser**' for '**Printer name**' box then click '**OK**'

**Note:** *if you are not using the tutorial at SCU Design Center, then find from your system manager the effective printer name.*

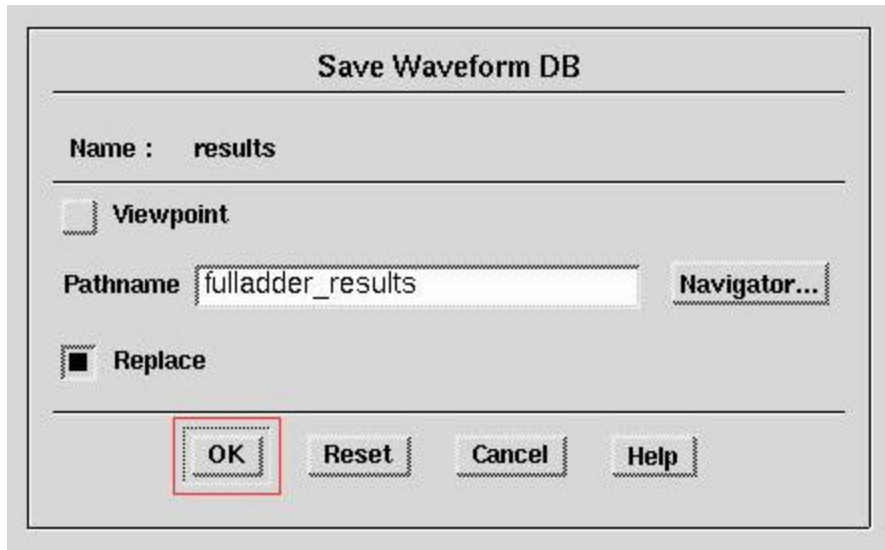


**Fig 5-1. Print dialog box**

---

## 6. Save the results and Exit

1. Double click the '-' button in the upper left corner on the window frame. An '**Exit Quicksim**' dialog box will appear. Click on '**OK**'. Another dialog box '**Save Setup**' will pop up. Accept all default settings by clicking '**OK**'.
2. Another dialog box '**Save Waveform DB**' will appear, enter the filename you want the output waveforms to be saved under. Enable the '**Replace**' button. Then click '**OK**'



---

**Fig 6-1.'Save Waveform DB' dialog box**