Design & Simulation of Rectifier through Multisim

Udit Mamodiya, Deepak Purohit, Goverdhan Singh
Poornima College of Engineering, Jaipur

Abstract- The manuscript contained simulation of common physics experiment, Halfwave, Fullwave and Bridge Rectifiers using MULTISIM (Evolution mode) program. In the current paper circuit models designed in the computer program and analyze their wave forms (I/P & O/P) within the simulation computer program. Students can perform experiments inside the model and take out reading by varying Load. Multimeters and Oscilloscope provided in the simulation model for analyze purpose. The result obtained from simulation is compatible with the actual experiments. It is concluding that students who go through this simulation before actual experiments do not need faculty attention during the laboratory work.

INTRODUCTION
NI Multisim (formerly MultiSIM) is an electronic schematic capture and simulation program which is part of a suite of circuit design programs, along with NI Ultiboard. Multisim is one of the few circuit design programs to employ the original Berkeley SPICE based software simulation. Multisim was originally created by a company named Electronics Workbench, which is now a division of National Instruments. Multisim includes microcontroller simulation (formerly known as MultiMCU), as well as integrated import and export features to the Printed Circuit Board layout software in the suite, NI Ultiboard. Multisim is widely used in academia and industry for circuits education, electronic schematic design and SPICE simulation.

Semiconductor manufacturers provide SPICE models to facilitate simulation of their devices by customers. SPICE technology is commonly used to model discrete components such as BJTs, Operational Amplifiers, and MOSFETs. Simulation models are also increasingly being developed for integrated systems such as integrated power converters and switching controllers where SPICE simulation plays a critical role in the evaluation of system performance, power efficiency, conduction, switching losses, and overall thermal behavior. In addition engineers are using behavioral simulation models of high-density general purpose amplifiers to determine performance parameters such as gain and phase margins, noise behavior, and harmonic distortion. NI MULTISIM includes over 26,000 devices validated by leading semiconductor manufacturers such as Analog Devices, NXP, Infineon, Texas Instruments, ON Semiconductor, Microchip, Maxim, and many others.

NI ULTIboard is an electronic Printed Circuit Board Layout program which is part of a suite of circuit design programs, along with NI MULTISIM. One of its major features is the Real Time Design Rule Check, a feature that was only offered on expensive work stations in the days when it was introduced. ULTIboard was originally created by a company named Ultimate Technology, which is now a subsidiary of National Instruments. Ultiboard includes a 3D PCB viewing mode, as well as integrated import and export features to the Schematic Capture and Simulation software in the suite, MULTISIM.

MULTISIM & ULTIboard
NI MULTISIM is an electronic schematic capture and simulation program which is part of a suite of circuit design programs, along with NI Ultiboard. MULTISIM is one of the few circuit design programs to employ the original Berkeley SPICE based software simulation. MULTISIM was originally created by a company named Electronics Workbench, which is now a division of National Instruments. MULTISIM includes microcontroller simulation (formerly known as MultiMCU), as well as integrated import and export features to the Printed Circuit Board layout software in the suite, NI Ultiboard. MULTISIM is widely used in academia and industry for circuits education, electronic schematic design and SPICE simulation. Several separate tasks comprise the overall process of circuit simulation in MULTISIM. These include: (1) schematic capture, (2) device modeling, (3)
setting up and running the simulation, and (4) output analysis.

**SIMULATION**

A simulation is designed in the NI MULTISIM 10.0 evolution mode. Three different experiments designed in three different windows of MULTISIM. Identical (Similar to the actual experiment) components placed in the circuit utility board. Required junctions and wire.

**REVIEW PROCESS ADOPTED**

A literature review is necessary to know about the research area and what problem in that area has been solved and need to be solved in future. This review process were divided into five stages in order to make the process simple and adaptable. The stages were:

- **Stage 0: Get a “feel”**
  This stage provides the details to be checked while starting literature survey with a broader domain and classifying them according to requirements.

- **Stage 1: Get the “big picture”**
  The groups of research papers are prepared according to common issues & application sub areas. It is necessary to find out the answers to certain questions by reading the Title, Abstract, introduction, conclusion and section and subsection headings.

- **Stage 2: Get the “details”**
  Stage 2 deal switch going in depth of each research paper and understand the details of methodology used to justify the problem, justification to significance & novelty of the solution approach, precise question addressed, major contribution, scope & limitations of the work presented.

- **Stage 3: “Evaluate the details”**
  This stage evaluates the details in relation to significance of the problem, Novelty of the problem, significance of the solution, novelty in approach, validity of claims etc.

- **Stage 3+: “Synthesize the detail”**
  Stage 3+ deals with evaluation of the details presented and generalization to some extent. This stage deals with synthesis of the data, concept & the results presented by the authors.

**How to perform bridge rectifier circuit by multisim**

A bridge rectifier makes use of four diodes in a bridge arrangement to achieve full wave rectification. This is a widely used configuration, both with individual wired and with single components bridges where the diode bridge is wired internally.

**Component used are :**

1. Sinusoidal voltage source : 10V, 50Hz
2. 4 diodes: 1N4007
3. Resister : 100k ohms
4. Oscilloscope
Output:
After simulation of program we will get the output as –

Table:
A reading of Voltage across load resistor and current pass through load resistor were taken by varying load resistor within the simulation model.

<table>
<thead>
<tr>
<th>R (KΩ)</th>
<th>I (µA)</th>
<th>Volt (V)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>513</td>
<td>4.61</td>
</tr>
<tr>
<td>2</td>
<td>575</td>
<td>4.60</td>
</tr>
<tr>
<td>3</td>
<td>655</td>
<td>4.59</td>
</tr>
<tr>
<td>4</td>
<td>762</td>
<td>4.57</td>
</tr>
<tr>
<td>5</td>
<td>910</td>
<td>4.55</td>
</tr>
</tbody>
</table>

How to perform full wave rectifier by multisim:
Basically a rectifier is usually used to rectify the pulses or waveforms of AC to DC. Another name of Full wave rectifier is Bridge rectifier as is mentioned in the title of this post.

This circuit consists of a set of 4 diodes connected in a bridge like structure. At any instant of time, only 2 diodes will be functional and other 2 will remain idle. In the positive half of the cycle, 2 diodes which are in forward bias mode will form the circuit and will conduct. In the negative half cycle, other 2 diodes which were earlier in reverse bias mode will be in forward bias mode and other 2 will be in reverse bias mode.

Components used are:
1. Sinusoidal voltage source: 10V, 50Hz
2. 4 diodes: 1N4007
3. Resistor: 100k ohms
4. Oscilloscope

Output:
When we run the simulation, only 2 diodes will conduct in each half cycle and will give rise to a
voltage waveform which doesn’t contain any negative half of the original sinusoidal waveform. You can see the waveforms in the image given below:

**Table:**

A reading of Voltage across load resistor and current pass through load resistor were taken by varying load resistor within the simulation model.

<table>
<thead>
<tr>
<th>R (KΩ)</th>
<th>I (µA)</th>
<th>Volt (V)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1434</td>
<td>12.90</td>
</tr>
<tr>
<td>2</td>
<td>1613</td>
<td>12.89</td>
</tr>
<tr>
<td>3</td>
<td>1841</td>
<td>12.89</td>
</tr>
<tr>
<td>4</td>
<td>2148</td>
<td>12.88</td>
</tr>
<tr>
<td>5</td>
<td>2575</td>
<td>12.87</td>
</tr>
</tbody>
</table>

**How to perform half wave rectifier by multisim:**

In our day to day electronics applications, we need many circuits which enable us to rectify the AC waves to DC. In current electronics scenario, we need circuits as small as possible to minimize costs. Since rectifier circuit is one small part of any big electronics application, so we need to minimize the size of every circuit involved in any project. One such simple and small rectifier is a half wave rectifier which uses a PN junction diode. Though this is not any advanced circuit which can be used for high accuracy rectification with minimum ripple, but still this circuit forms the basis of rectifiers.

**Result:**

Here we are using a 10V, 60Hz AC voltage source, 100K ohm resistor and pn junction diode 1N4001 all connected in series with each other. To check the waveforms of the load resistor R1, I have connected an oscilloscope across the resistor. When the simulation starts, during the positive half cycle of the AC source, the diode is forward biased and hence act as a conductor, allowing the positive part of the sine wave to pass through. During negative half cycle, the diode is reverse biased and hence act
as an insulator, so the negative part of the wave appears as zero at the output. Hence the sinusoidal wave is rectified partially. Since it is partially rectified, so it is called Half wave rectifier. The output wave across the resistor R1 is shown in the snapshot shown below.

CONCLUSION:
Groups of students were performed experiments using simulation model and understand all the required aspect of experiment. After that they bring to the laboratory for actual experiment. Significantly, they performed actual experiment and found voltage regulation (%) without attention of faculty. It shows that students can better understand visualization of experiments compared to traditional teaching method.

FUTURE PROSPECT:
Lots of Physics experiments related to electronics background can be visualized by simulation model using MULTISIM. This will help teacher as well as students for better understanding of experiments and it all related theories.

Table:
A reading of Voltage across load resistor and current pass through load resistor were taken by varying load resistor within the simulation model.

<table>
<thead>
<tr>
<th>R (KΩ)</th>
<th>Halfwave Rectifier</th>
<th>1 (µA)</th>
<th>Volt (V)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>277</td>
<td>2.49</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>311</td>
<td>2.49</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>355</td>
<td>2.49</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>414</td>
<td>2.48</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>496</td>
<td>2.48</td>
<td></td>
</tr>
</tbody>
</table>

References: