

sdmay22-39: Fast, Compact, High Strength Magnetic Pulse Generator

Weekly Report 2

Oct 04 - Oct 10

Team Members

Ben Newell

James Camp

Mohd Harith Arsyad

Abdulraheem Alquinas

Tom Zaborowski

Tyler Bolton

Weekly Summary

In the weekly Monday meeting, the group discussed and analyzed the previous groups' circuits. We discussed how every aspect of the circuit should work. The main components were the capacitor bank, the inductor and diode section, the MOSFET, and the voltage pulser section. After in-depth discussion on these topics, the group discussed documentation. The group is planning on drafting the weekly reports on a weekday instead of Sunday in order to send this to the group's academic advisor. The current task for this week is to simulate the circuit designs from the previous years and send in a report to the group's academic advisor.

Pending Issues

When doing the building the circuit for simulations in Multisim, we couldn't find some of the components used by the previous groups such as the diodes and MOSFETs. This could be because we need to import the specific components previous groups used. We have now been given resources on this topic but a few members are still running into error pop-ups when importing components into multisim.

Weekly Accomplishments

Ben Newell - Learned how to import SPICE models into multisim. Built and tested may17 and may20 circuits. Worked on project plan document and prepared for in class lightning talk.

James Camp - Created working version of circuit in Multisim. Have not tested with switching diode or with transistor used in previous design groups.

Mohd Harith Arsyad - Learned how to import MOSFETs into Multisim. Then I ran various analyses such as parameter sweeps on different MOSFET's to learn the role of each component in the circuit better.

Abdulraheem Alquinas - Learned how to import PSpice model into multisim, and simulated the circuit designs for (sdmay20-39) group.

Tom Zaborowski - Simulated the circuit designs from the 2016-2017, 2019-2020, and 2020-2021 groups via Multisim. Learned how to import components from other databases. Made a report on the findings in Multisim. Attended group meetings.

Tyler Bolton - Continued simulation of the previous circuit designs. Learned how to import components from PSPICE to Multisim.

Plans for Next Week

Ben Newell - Learn how to import specific component models into ADS. Set up a circuit model in ADS.

James Camp - Learn how to perform transient analysis in ADS and how to import Models (probably SPICE model) into ADS and Multisim. Design non-ideal version of circuit.

Mohd Harith Arsyad - Simulate the 2015 and 2020 circuits and start looking for ways to improve the design of the circuits. Also continue going through the research papers.

Abdulraheem Alquinas - Simulate the circuit designs of (may1740) and (sdmay21-42), continue reading the research papers, and modify the previous group circuit to get a better result.

Tom Zaborowski - Start designing a new circuit based on what was learned from this week and last week.

Tyler Bolton - Edit the previous circuit design or start a new circuit design that will meet the requirements.

Individual Contributions

Name	Individual Contribution	Hours
------	-------------------------	-------

Ben Newell	Multisim troubleshooting, simulations of May17 & May20 circuit, Project Plan Documentation	8.5
James Camp	Simulating in Multisim, Documentation	3
Mohd Harith Arsyad	Multisim simulation, Importing models ,and Documentation	7.5
Abdulraheem Alquinas	Simulations, Importing models in multisim, and Documentation	7.5
Tom Zaborowski	Multisim Simulations, Multisim import troubleshooting, and Documentation	9
Tyler Bolton	Multisim Simulations, Multisim Import, and Documentation	7.5