Faculty of Electrical Engineering & Computer Science
Master Degree Course Electronics Engineering, MSc EE

HSB
Hochschule Bremen
City University of Applied Sciences

Master 2022

MD Moin Uddin Biswas

EMI Filter Design and Verification using RF Simulation Tools and Measurements.

ABSTRACT - Masterthesis

Modern technologies are depended on efficiency that includes reduction in size, cost, supply power of a system. All these efficiencies come with the obvious generation of noise. The more a device is noise-free the more it becomes efficient. All the electronic devices are produced considering the effect of electromagnetic interference. This interference is defined as the noise that travels through the electronic components like resistors, inductors, capacitors, wires, printed circuit boards, or through the overall design. This form of electromagnetic interference (EMI) has a serious impact on malfunctioning. To reduce this interference a filter is used named electromagnetic interference (EMI) filter.

Ideally, an EMI filter is a low pass filter that blocks the high frequencies as well as the noise flow while passing through the input. It will also reduce the amplitude of high-frequency signals (which are greater than the cutoff frequency). This filter is constructed with two lumped elements inductors and capacitors. There are several orders of this filter depending on the component's alignment and value.

This thesis work will deal with an EMI filter constructed with basic components. We will measure the unintended radiated electromagnetic wave from the test kit and the disturbance, also how much radiation actually produced. We will apply different valued inductors, capacitors, and chocks to measure the filter effect first and then we will synthesize the disturbances. Microwave Office software (AWR) will be used to calculate S-parameters to simulate the actual device. We are going to perform linear and nonlinear measurements to observe the filter performance as well. The simulation of this filter circuit will also be done both in frequency and time domain and at the end we will compare the performance analysis between the manufacturers' circuit board and simulated filter.

Before starting the lab work, several filters will be designed and tested in the AWR platform to identify the least disturbance circuit. Hardware simulation will be performed after designing the proper filter schematic, and the printed circuit board accordingly.