Dear Students,

In the Fundamental of Circuit Design Course, you are given a project which illustrates the use of LTSPICE software to the analysis of a simple analog circuit. LTSPICE is a commercial software developed by Linear Technology, based on the widely used circuit simulator core, SPICE.

A similar software, MULTISIM developed by National Instruments is also available for your evaluation from the folllowing web site (needs registration):

"*www.ni.com/multisim/try/*"

In the followings, two simple examples (Examples 1 and 2) illustrating the use of MULTISIM in the analysis of analog circuit and of logic circuits are provided.

You may consider studying these examples while you are on vacation. You may also consider the presentation files prepared by Prof. Bitar from WPI.

Examples:

1- http://web.itu.edu.tr/ozoguz/logic_circuit_example.pdf

2- web.itu.edu.tr/ozoguz/analog_circuit_example.pdf

Presentation Files from Prof. Bitar's website

1- http://users.wpi.edu/~sjbitar/ece2799/lectures/Topic17%20-%20Using%20Simulation%20Effectively/NI%20MS%20Tut%20Pt%201.pptx
2- http://users.wpi.edu/~sjbitar/ece2799/lectures/Topic17%20-%20Using%20Simulation%20Effectively/NI%20MS%20Tut%20Sample%20Ckts.pptx