



Using ANSYS for Civil Engineering Purposes

By: Shaurya Gupta ; BE/10630/15

Department of Civil & Environmental Engineering

Project Guide: **Prof. James Thomson (Ph.D)**

Carnegie Mellon University, Pittsburgh, PA.

ANSYS, Inc. is an American public company based in Canonsburg, Pennsylvania. It develops and markets engineering simulation software. ANSYS software is used to design products and semiconductors, as well as to create simulations that test a product's durability, temperature distribution, fluid movements, and electromagnetic properties. ANSYS develops and markets finite element analysis software used to simulate engineering problems. The software creates simulated computer models of structures, electronics, or machine components to simulate strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes. ANSYS is used to determine how a product will function with different specifications, without building test products or conducting crash tests. For example, ANSYS software may simulate how a bridge will hold up after years of traffic, how to best process salmon in a cannery to reduce waste, or how to design a slide that uses less material without sacrificing safety.

Most ANSYS simulations are performed using the ANSYS Workbench software, which is one of the company's main products. Typically ANSYS users break down larger structures into small components that are each modeled and tested individually. A user may start by defining the dimensions of an object, and then adding weight, pressure, temperature and other physical properties. Finally, the ANSYS software simulates and analyzes movement, fatigue, fractures, fluid flow, temperature distribution, electromagnetic efficiency and other effects over time. ANSYS also develops software for data management and backup, academic research and teaching. ANSYS software is sold on an annual subscription basis.

My Project Goals at the Carnegie Mellon University included, learning how to use ANSYS Mechanical interface and Workbench to solve static and dynamic structural problems and to write a handbook to help students use ANSYS. I ran multiple simulations on bridges, trusses, frames, beams and spring mass systems to explore and analyze the software. I and my teammate were able to determine and report a few bugs in the software to the ANSYS team which we were informed would be corrected in the 19.2 version of the software. Some screenshots from my analysis are depicted below:

