1-1-2022

Numerical Simulation on the Fire Performance of Reinforced Concrete Flat Slab

Elias Ali
University of Alabama in Huntsville

RCEU 2022 Project Proposal

Project Title

Numerical Simulation on the Fire Performance of Reinforced Concrete Flat Slab

Faculty Information

Name: Elias Ali

Status: Lecturer

Department/Program: Civil and Environmental Engineering

College: College of Engineering

Phone: 2568246856

UAH Email: elias.ali@uha.edu

Proposal ID RCEU22-CEE-EYA-01

Instructions are on the last page.

RCEU 2022 Project Proposal

I. Project Description

This project has the aim of providing students an insight and research experience to tackle current civil engineering problems using available computational tools. The project will investigate the fire performance of a reinforced concrete flat slab under standard fire using the available computational Finite Element Package, ABAQUS. Three analysis stages for structural members under fire conditions will be considered. 1) Heat transfer analysis, 2) Structural stability analysis and 3) Structural collapse analysis

To investigate the structural behavior of any structural members under fire, one has to understand and obtain the temperature distribution on the cross-sections at different temperature-time intervals, for which, heat transfer analysis will be first performed using ABAQUS.
Numerical simulation and models for structural buckling behavior will later be performed on finite element software ABAQUS, and obtain elastic buckling modes (yield lines), and finally, the non-linear collapse capacity of the slab understudy will be evaluated using an analytical approach considering the reduction in material properties during a fire.

II. Project Objectives

Recent advances in computing have enabled computational methods to become important, less expensive, and efficient tools than large-scale experiments to investigate real-life engineering problems. Yet many undergraduates are not positioned to take advantage of this valuable resource. This research experience is thus designed to take advantage of computational tools such as ABAQUS for simulating fire loading on structures.
The primary objectives of this project are:

- To expose students from primarily undergraduate Civil, Mechanical, and Aerospace Engineering students' to leveraging computational methods for analyzing extreme loading, such as fire on structures
- To improve students understanding, level of computational and simulation skills using Finite Element Package, ABAQUS
- To enable students, make more informed career and graduate or professional school choices. Upon program completion, students will be better prepared to achieve a high level of success in their chosen pathways.

III. Project Outcome

Upon completion of the project, the following outcomes are expected:

- Perform heat transfer analysis and obtain a reduction in material properties with elevated temperature
- Identify the influence of temperature distribution on the stability of the structural element
- Compute the strength of structural element at elevated temperature based on the result obtained from computational analysis

- Propose the computational approach used to predict the strength of other similar structural members of the same material and geometry

IV. Student Duties, Contributions, and Outcomes

Finite element analysis (FEA) using ABAQUS will be used to perform the heat transfer, structural stability, and collapse analyses of the structural element under standard fire conditions. To achieve the project goal, selected students will have the following specific duties, contributions, and outcomes:

a. Specific Student Duties

The selected student will expect the following duties

- Write literature reviews on the research topic
- Familiarize him/herself with finite element software package, ABAQUS
- Perform heat transfer analysis using ABAQUS
- Perform Stability and Collapse Analysis
- Provide preliminary research and computational outputs

b. Tangible Contributions by the Student to the Project (10% of Review)

The selected student will be expected to provide preliminary research and computational outputs and present the results of the research at a Poster Session at the start of the Fall 2022 semester.

c. Specific Outcomes Provided by the Project to the Student (30% of Review)

- The selected student will gain first-hand experience on the use of computational methods for analyzing extreme loading on structures.
- The selected student will gain an understanding, and improve the level of computational and simulation skills using FEA
- The selected student will gain experience and be more informed on the future career and graduate or professional school choices.
- The selected student will gain the opportunity to demonstrate strength and explore other potentials

V. Student Selection Criteria

This project is open to Civil and Environmental Engineering (CEE), and Mechanical and Aerospace Engineering (MAE) students at ALL-ACADEMIC RANKS. However, this project will primarily focus on recruiting students from underrepresented groups, women, and first-generation college students to evasion their talent, dream, and competitiveness in the STEM field.

VI. Project Mentorship *(30% of Review)*

To maximize the outcome of this research experience, the selected student will work under the close supervision of the mentor (me). I will meet with the mentee in monitoring the progress of the research at least twice a week in person. Details of the mentorship are provided below:

- Mentor will train the mentee Finite Element Analysis software, in this project ABAQUS
- Mentor will provide the mentee hands-on support in using MATLAB for computational analysis
- Mentor will provide necessary documents, articles, and resources to the mentee
- Twice a week in-person meetings will be held at the mentor office, to discuss and monitor the research progress
- Mentor will arrange additional online meetings as appropriate

VII. Safety and Contingency Plan

For any change to health and safety guidance that restricts or limits face-to-face contact, an alternative online meeting will be arranged by the mentor. Since this proposal is of computational research entirely, the change in health and safety guidelines will have no to minimal impact on the progress and outcome of the project.