Ministry of Higher Education
and Scientific Research
Al-Muthanna University
College of Engineering
Department of Civil Engineering

Numerical Simulation of the Behvioar of Composite Columns Subjected to Fire and Axially Compressive Load

A thesis submitted to the council of the College of Engineering at Al-Muthanna University in partial fulfilment of the requirements of for the degree of Master of science (M.Sc.) in Civil Engineering

Zainab Hayder Kassim

Supervisor

Asst. Prof. Dr. Alaa Salam Al-Husainy

July 2024 Dhu Al-Hijja 1445

Summary

Columns represent one of the most important structural parts in building, and the composite column is one of the important types of columns due to its ease of construction, speed of work, and higher strength compared to a reinforced column with the same dimensions. Therefore, it is one of the most important types of columns used in multi-story buildings, especially the concrete filled steel tube column. In modern society, human activities have increased inside buildings, multi-story buildings and skyscrapers have also increased, and the rate of fires of varying severity has increased. Therefore, it is necessary to study the behvioar of structural members under the influence of fire and their duration of fire resistance, especially the column, which represents a vital member of the structure, to know the factors affecting the critical time before the building collapsed to help save as many people as possible. Because of the difficulty of studying columns under fire conditions, especially the full size of the columns, engineers moved to further study them and delve into them more deeply through simulation programs, including the Abaqus program, as in this research. The Abaqus program is one of the most common numerical simulation programs for representing structural elements under various conditions, which follows the finite element method with a relatively large number of elements and multiple shapes and types of elements. In this study, simple solid elements were used to model the column, and a sequential thermal analysis was used, dividing the model into two parts. The first is a pure heat transfer analysis in which the temperatures were simulated and its transfer from the atmosphere to the surface of the model and then to the rest of the parts of the model were simulated in the form of Contact surface. In the second part, an stress analysis was used, in which the results of temperature from the first model were recalled and static

loads were applied to simulate complex loads as in reality. After creating an integrated model and obtaining results that were similar to the behvioar of the element in the practical model, from the transfer and distribution of temperatures to the failure mode and vertical displacement, a parametric study was created in which the effect of changing the strength of concrete and steel was tested. It was found that the critical time was reduced by 15% with a decrease in strength of concrete by a quarter, while the effect of the yield strength of steel is almost imperceptible. The effect of the length to diameter ratio, temperature location, and installation conditions was also studied, and it was concluded that the closer the fire is to the installation point, the greater the effect of the fire and decrease the critical time. The effect of adding an additional steel section inside the concrete to become a double-skin column was studied. Adding an additional steel section inside the column greatly increases its strength and its heat tolerance period, especially if the additional section is in the form of a tube, as it will give double the confining force to the inner concrete core.