

NUMERICAL STUDY OF THE BEHAVIOUR OF A SUB-FRAME ASSEMBLY IN A COMPOSITE FRAMED STRUCTURE EXPOSED TO FIRE

Naveed Iqbal

Division of Structural and Construction Engineering-Steel Structures, Luleå University of Technology, Sweden

Tel.: +46 920 492198; Fax: +46 920 492818; E-mail: naveed.iqbal@ltu.se

Abstract

When a building is exposed to fire, many changes in the internal forces of its various structural members are observed. These changes are basically due to two different reasons; first one is the increase in the internal forces due to restrained expansion and rotation of the a structural member exposed to a variable temperature gradient through its cross section depth and the other is the degradation of the material properties at higher temperatures.

The purpose of this study is to determine the response of the Finite Element model of a sub-frame assembly in a composite steel-concrete framed structure to different temperature distribution scenarios and assess the loading histories induced on the joints and the restraining effect of the surrounding structure on a heated member.

This study is part of the COMPFIRE project under which real fire tests are being planned to be conducted at the sub-frame level as well as complete structures. The data from these tests will be used in the validation of the Finite Element models developed in this study using commercial software ABAQUS, in order to perform further parametric studies. The conclusions from this study would help towards developing a better understanding of the robustness of composite structures in fire with better knowledge of the interaction between its various components.