

FEM SIMULATION OF COMPOSITE COLUMN DURING FIRE EXPOSURE BASED ON DIN EN 1991-1-2/NA:2010-03

Katarzyna Ostapska

Faculty of Civil Engineering, Warsaw University of Technology, Warsaw, Poland

The aim of this presentation is to provide step by step implementation of a numerical model for the benchmark problem provided by the DIN EN 1991-1-2/NA:2010-03 (CC.4.11 Beispiel 11 - Verbundstütze mit Kammerbeton). A composite column is considered with initial geometric imperfection during fire exposure. Initial deformation is parabolic with the peak value in mid-span. The column's section is bisymmetrical reinforced concrete with partially encased steel profile and four reinforcing bars placed symmetrically. The analysis is performed using FE programs Abaqus and Ansys in cooperation with dr. Ioan Both and prof. Raul Zaharia. Obtained results are compared with DIN data as well as with other numerical results. The main parameters taken into account for comparison are: fire resistance and mid-span displacement at selected time points after subjecting column to uniform heating on four sides. Performed numerical simulation helped to estimate initial parameters, materials types, and thermal analysis options and to verify assumed mesh for this DIN benchmark.