# NUMERICAL ANALYSIS OF A COMPOSITE STEEL-CONCRETE COLUMN SUBJECTED TO FIRE USING ABAQUS

Ioan Both

University “Politehnica” of Timisoara, Romania

The presentation is focused on the numerical model for a composite column using solid elements. The input data for the model are taken from the German National Annex DIN EN 1991-1-2 which provides several results for thermal and structural analysis of the column.
The analysis was performed using the computer program Abaqus, during the STSM at Warsaw University of Technology, Poland, under the supervision of prof. Leslaw Kwasniewski. The issues of the analysis relate to the boundary conditions applied, mesh dimensions, material properties and interactions. The 3D analysis will highlight considerations that are not present in a 2D analysis using the beam element.

The parameters for the behavior of the materials, the numerical procedures, the boundary conditions, etc. considered in the numerical model, as well as the results, are discussed in comparison tothe work performed by Prof. Leslaw Kwasniewski and Katarzyna Ostapska, using the computer program Ansys.