FE-SFS PROJECT: Numerical analysis techniques for structural fire safety design of steel structures.

FE-SFS Model input files and Tutorial Videos

Description: This project aims to close the knowledge and skills gaps for using performance-based methods for structural fire safety of steel structures in Finnish design companies and research organizations in Finland. Hence, training material and tutorial videos were developed to provide training for performance-based structural fire safety design and relevant analysis techniques in this project. Here are the model inputs files and tutorial videos for selected fire test simulation in Abaqus. The selected fire test is a simple beam exposed to fire by Cong et al. (2005) and a restrained column by Ali and O'Connor (2001).

Model input files: Model input files are the ready-made input files to perform thermal and structural analysis of each fire test from previous periods, which were simulated and validated using FEM software Abaqus. A user can directly run the model input files in Abaqus solver. The model and input files contain input files for 2D thermal analysis and 3D structural analysis for Abaqus software. The input files developed during these projects are also stored in GitHub for selected fire tests used in making the video tutorials. The location of GitHub can be found below. Table 1 summarizes the name of mode input files and their descriptions.

Table 1   Model input files example explanation for case 1 simple beam exposed to fire by Cong et al. (2005)

Name of model input files	Software	Purpose
2D_temp.inp	Abaqus	To perform thermal analysis using shell element of case 1.
structural_beam.inp		To perform structural analysis using beam element of case 1.
structural_shell.inp		To perform structural analysis using shell element of case 1.
temperature_shell.odb		Result fire of thermal analysis which will be used in structural analysis using shell element.
simple_beam_mat_temp_data.py		A python code to write material and temperature from thermal analysis in Abaqus CAE.

The location of the input file on GitHub can be found below:

https://github.com/neudilip/FE-SFS.git

Tutorial Videos: Tutorial videos for a demonstration of the complete process of structural fire analysis, including thermal and structural analysis for a few fire tests, have been developed during this project. The tutorial videos were developed to perform thermal and structural analysis. For tutorial videos, the selected fire test is a simple beam exposed to fire by Cong et al. (2005) and a restrained column by Ali and O'Connor (2001).

The tutorial videos were developed as workshop tutorial videos. The training and tutorial PowerPoint materials, model input files, and tutorial videos are directly related to each other and are used in each tutorial video. For each selected fire test, three tutorial videos are developed to demonstrate how to perform thermal analysis, structural analysis using beam elements, and structural analysis using shell elements. The developed videos tutorials are published on the YouTube channel. The link to YouTube is illustrated in table 2. The following table summarizes the developed tutorial videos.

Table 2   Tutorial videos, workshop descriptions, and links to YouTube

Fire test	Types of Analysis	Description of tutorial videos	YouTube links
Simple beam exposed to fire by Cong (2005)	thermal	Thermal response analysis of simple beam exposed in fire	https://youtu.be/R7A2BA0StsI
	structural	Video tutorial of structural response analysis of simple beam structure exposed to fire using beam element.	https://youtu.be/HIS57PLdhto
	structural	Video tutorial of thermal and structural response analysis of simple beam structure exposed to fire using shell element.	https://youtu.be/FcHC1iZFKMk
Restrained Column by Ali and O'Connor (2001)	structural	Video tutorial of structural response analysis of restrained column exposed to fire using beam element.	https://youtu.be/UXJ0u9xuuY4
	structural	Video tutorial of structural response analysis of restrained column exposed to fire using shell element.	https://youtu.be/kEJUd1Kanf4

All the tutorial videos are published on YouTube in a YouTube Channel named FE-SFS. The single web address of the channel, which includes all the videos tutorials, is as follows.

https://www.youtube.com/channel/UCMNr9JcfCMxD0IalJGmm-VA

List of Publications: During this project, the following three research papers and a master thesis have been published, and they are related to the developed model input files and videos tutorials. The learner is advised to read the following publications for detailed understanding of fire test simulation.

1. Z. Ma, J. Havula, and M. Heinisuo (2019). Structural fire analysis of simple steel structures by using LS-DYNA. Rakenteiden Mekaniikka (Journal of Structural Mechanics), Vol. 52, No 1, 2019, pp. 1–22.

2. D. Neupane (2020). Structural analysis technique of simple steel structures exposed in fire using Abaqus. Master's thesis. LUT University. http://urn.fi/URN:NBN:fi-fe202102164978

3. Z. Ma, J. Havula, F. Wald, K. Cabova (2020). Temperature analysis of steel structures protected by intumescent paint with steel claddings in fire. Fire and Materials, 44(7): 897-908. Doi: 10.1002/fam.2890

4. Z. Ma, J. Havula, A. Joo, A. Lendvai (2021). FE model validations of steel structures with steel claddings at elevated temperatures. The 9th European Conference on Steel and Composite Structures, Sheffield, UK, September 2021. **https://doi.org/10.1002/cepa.1413**

References and details about fire test:

Ali, F., O'Connor, D. (2001). Structural performance of rotationally restrained steel columns in fire. Fire Safety Journal, 36: 679-691.

Cong, S., Liang, S., Dong, Y. (2005). Experimental investigation of behavior of simple supported steel beams under fire. Journal of Southeast University (Natural Science Edition), 35 (sup (I)): 66-68.