Timisoara, March 2nd , 2013

STSM Scientific Report

COST Action TU0904

STSM Reference code: COST-STSM-ECOST-STSM-TU0904-010313-024723

Visiting student:	Ioan BOTH Faculty of Civil Engineering "Politehnica" University of Timisoara Timisoara, Romania
Host:	Prof. Leslaw KWASNIEWSKI Faculty of Civil Engineering Warsaw University of Technology Warsaw, Poland

Period of STSM: 02/03/2013 – 30/03/2013

Purpose of the visit

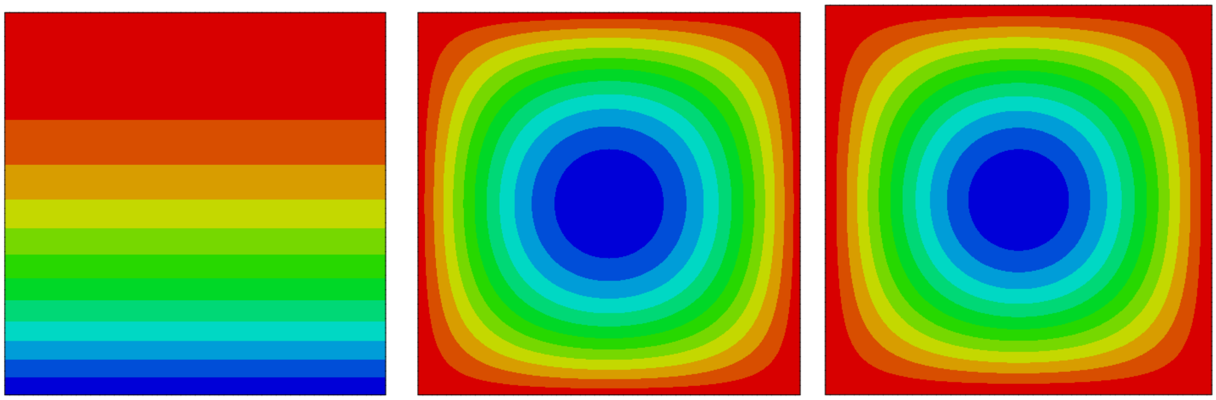
The main purpose of the STSM was the numerical simulation of thermal action on structural elements. The cases analyzed were chosen so that it begins with simple examples and tries to develop complex ones.

Description of the work carried out during STSM

The work started by performing simple heat transfer examples which are validated by the German National Annex of Eurocode 1: Actions on structures – Part 1-2: General actions – Actions on structures exposed to fire, DIN EN 1991-1-2/NA.

In order to calibrate the heat transfer, there were modelled 3 simple cases. These examples indicate the validity of the input data, material properties under thermal conditions as well as the validity of the thermal equation used by the FE software: Abaqus 6.11.

The first example takes into consideration a material that has an initial temperature of 1000°C and is sinked into an ambient temperature of 0°C.



a) Example 1 b) Example 2 c) Example 3

Fig. 1 Heat transfer analysis in Abaqus

The heat transfer takes into consideration only the convection. The results obtained are similar to the reference values provided by DIN EN 1991-1-2/NA (Table1)

Table 1.

Time	Reference level ΘO	.... ABAQUS			Limit deviation
		Calculated Θ'O	Deviation		
	Temperature	Temperature	[(Θ'O - ΘO)/ΘO] ×100	(Θ'O - ΘO)	
[s]	[°C]	[°C]	[%]	[K]	
0	1000	1000.0	0	0	± 1 %
60	999.3	998.3	-0.10	-1.0445	
300	891.9	891.6	-0.03	-0.3104	and
600	717.7	718.8	0.15	1.0811	
900	574.9	576.3	0.24	1.3721	± 5.0 K
1200	460.4	461.9	0.33	1.5034	
1500	368.7	370.2	0.41	1.5282	
1800	295.3	296.7	0.49	1.445	

Furthermore, there was performed a Grid Convergence Index (GCI) calculation. For the GCI there were compared 3 element sizes (h1=0.0125,h2=0.025, h3=0.05). The results show the precision with respect to the asymptotic solution ($f_{h=0}$) function of the mesh density (Table 2).

Table 2.

	ABAQUS (2D)			Temp	refinement ratio (r)	Order of convergence (p)	Asymptotic solution $f_{h=0}$	ε	GCI12 [%]	GCI23 [%]	$GCI23/r^p$ GCI12
sec		mesh	size	ref point							
900	Thermal	1	0.0125	576.27	1.41E+00	4.00E+00	5.76E+02		-3.62 E-04	-1.45 E-03	1.00E+00
		2	0.025	576.28				-8.68E-06			
		3	0.05	576.297				-3.47E-05			

The second example presents the heat transfer for a material taking into consideration convection and radiation. The material's initial temperature is 0°C and it is introduce in an ambient temperature of 1000°C. The results are very close to the reference values provided by DIN EN 1991-1-2/NA.

Table 3.

Time [min]	Reference level ΘO Temperature [°C]	.... ABAQUS .... Calculated Θ'O Temperature [°C]	Deviation [(Θ'O - ΘO)/ΘO] ×100 [%]	 (Θ'O - ΘO) [K]	Limit deviation
30	36.9	35.28	-4.4	-1.62	for t ≤ 60 min
60	137.4	136.34	-0.8	-1.06	± 5.0 K
90	244.6	245.01	0.2	0.41	
120	361.1	364.6	1.0	3.5	for t > 60 min
150	466.2	470.66	1.0	4.46	± 3 %
180	554.8	560.1	1.0	5.3	

Also the GCI computation show that the finer mesh the closer the results (Table 4).

Table 4.

	ABAQUS (2D)			Temp	refinement ratio (r)	Order of convergence (p)	Asymptotic solution $f_{h=0}$	ε	GCI12 [%]	GCI23 [%]	GCI23/r^{p} GCI12
min		mesh	size	ref point							
		1	0.0025	243.89							
90	Thermal	2	0.005	244.37	1.41E+00	3.54E+00	2.44E+02	-1.98E-03	-1.02 E-01	-3.49 E-01	9.98E-01
		3	0.01	246.02				-6.74E-03			

The third example was performed to investigate the behavior of two different materials in contact. The model was created using one part, dividing it into cells to which there were assigned different section properties. The heat transfer analysis takes into consideration convection and radiation as specified by the parameters in DIN EN 1991-1-2/NA. The results are very close to the reference values recommended by DIN EN 1991-1-2/NA (Table 5).

Table 5.

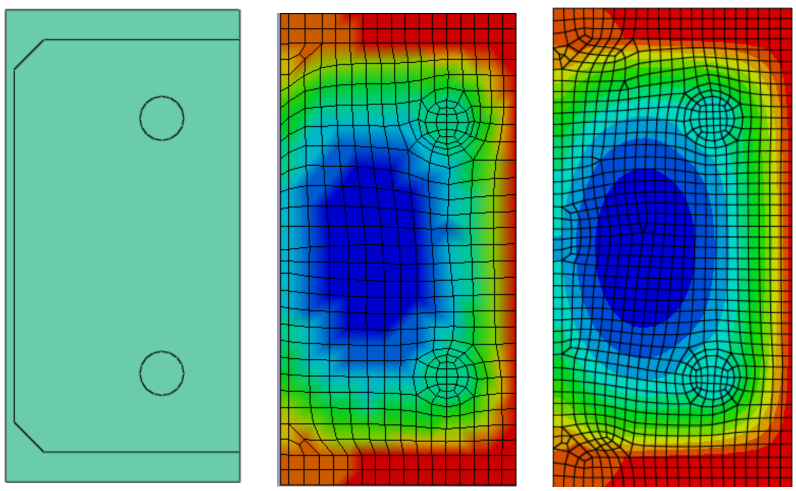
Time [min]	Reference level ΘO Temperature [°C]	.... ABAQUS .... Calculated Θ'O Temperature [°C]	Deviation [(Θ'O - ΘO)/ΘO] ×100 [%]	 (Θ'O - ΘO) [K]	Limit deviation
30	340.5	339.6	-0.3	-0.92	± 1 %
60	717.1	721.0	0.5	3.89	
90	881.6	884.5	0.3	2.9	and
120	950.6	952.2	0.2	1.6	
150	979.3	980.2	0.1	0.9	± 5.0 K
180	991.7	991.8	0.0	0.11	

The different element size used in the model show the precision of the results through GCI (Table 6).

Table 6.

	ABAQUS (2D)			Temp	refinement ratio (r)	Order of convergence (p)	Asymptotic solution $f_{h=0}$	ε	GCI12 [%]	GCI23 [%]	$\frac{GCI23/r^p}{GCI12}$
min		mesh	size	ref point							
		1	0.0025	884.48							
90	Thermal	2	0.005	884.21	1.41E+00	3.95E+00	8.85E+02	2.99E-04	1.27 E-02	5.02 E-02	1.00E+00
		3	0.01	883.17				1.18E-03			

After the validation of thermal analysis by obtaining similar results with DIN EN 1991-1-2/NA, for the first three examples, there was performed a thermal analysis of a composite cross-section of a column. The cross section is made of an HEB300 steel profile with reinforced concrete between the flanges. The cross section is subjected to an ISO fire . In order to compare results, the analysis was performed using two different FE software: Abaqus 6.11 and LS-DYNA.



a) b) c)

Fig. 2 a) model, b) LS-DYNA results, c) Abaqus results

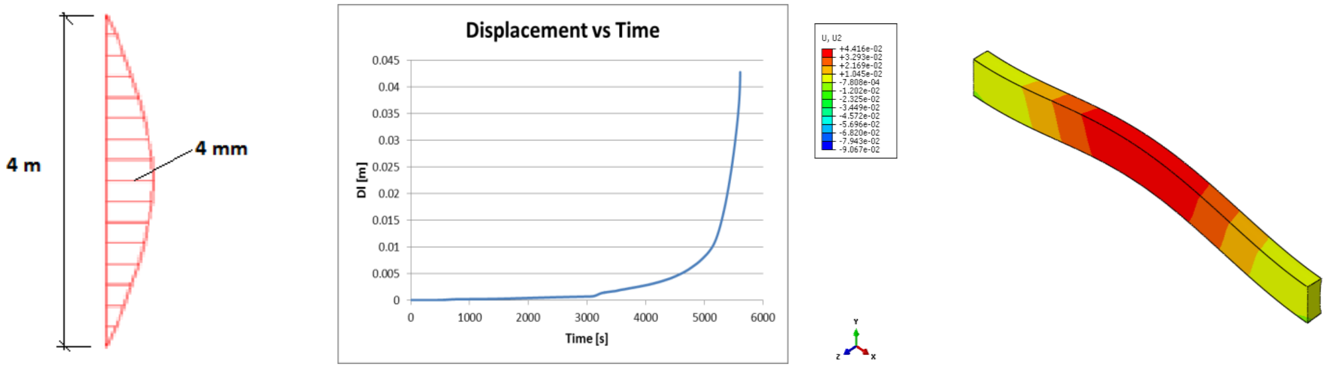
Comparing the temperature in the reinforcement and in the centroid of steel section it was observed that the results are close (Table 7).

Table 7.

Temperatures	Abaqus	LSDYNA
HEB	451	453
Rebar	512	539

Having the thermal analysis, it was intended to perform the structural analysis for a column subjected to a concentrated force and having the previous described cross-section. The analysis can be performed either by sequentially coupled analysis (obtain the node temperatures in the elements and then applying them, as a "predefined field" in the structural analysis) or by fully coupled thermal displacement analysis (the structural load and the thermal load is applied in the same time). Both analysis were performed for a simple case, in order to compare the results. Because the results were similar the following analysis were performed using the coupled thermal displacement analysis .

The geometry of the column is given in the following figure. The structural properties of the materials used, were defined as temperature dependent values. A big convergence problem is the concrete property definition. The concrete was defined first as a plastic material with same behavior for tension and compression. Secondly, the material behavior for concrete was modeled with *Concrete Damage Plasticity.



a) Initial geometry of column b) Displacement in midspan c) Deformed shape of column

Fig. 3 Coupled thermal displacement analysis (Abaqus results)

Description of the main results obtained

As a conclusion, the main achievements of the STSM are the ability to perform heat transfer analysis on structural elements as well as structural analysis on structural elements subjected to fire. Also the previous cases can be performed in a coupled thermal displacement analysis.

During the STSM, I attended a FE course for LS-DYNA. The course introduced me the basic notions of performing analysis using this software and making a thermal model for analysis of a composite cross-section.

Future collaboration with host institution (if applicable)

The collaboration with Warsaw University of Technology and especially with Prof. PhD. Leslaw Kwasniewski was a real support in understanding the analysis procedures and developing the knowledge on the thermal-structural analysis. For the future, collaboration may continue by comparing results for different structures subjected to thermal loadings, analyzed by FE software: Abaqus and LS-DYNA.

Foreseen publications/articles resulting or to result from the STSM

Dissemination of the results obtained in the current STSM may refer to the material definition in order to obtain convergence in case of composite steel-concrete sections. Benchmarks for the examples of heat transfer and structural analysis.

PhD. eng. BOTH Ioan