Cvetanka Chifliganec, from Ss. “Cyril and Methodius” University in Skopje, Faculty of Civil Engineering – Skopje to ETH Zurich (6th to 18th of May 2018)

- **The Purpose** of this STSM was (1) to do a state-of-the-art on numerical modelling of timber-concrete composite structures in fire and (2) to develop a new numerical model of a TCC slab with screwed connection in fire in the computer program SAFIR, which could calculate the slip between the subcomponents of the system.

- **Short summary of performed STSM**: The Host institution`s digital library was used for collecting appropriate literature in the field of research. After performing literature review, a state-of-the-art on numerical simulations of timber-concrete composite structures exposed to fire was done. Visit to the laboratory at ETH Zurich and attendance at one fire test was done. Also, visit to the Empa’s Fire Technology Lab was done. A new numerical model for analysis of the TCC slab with screwed connections in fire in the computer program SAFIR was developed.

- **Description of the main results obtained**: The state of the art showed that numerical modelling is done only for small number of tested TCC slabs in fire. The numerical model underestimated the fire resistance and the deflection of the TCC structure. After several different parameters were varied in the developed model, it was concluded that the response of the analysed TCC beam in fire was mainly governed by the behaviour of the timber beam in fire and not the connections.

- **Future collaboration with the host institution**: No particular plans for future collaborations are made for now.

- **Foreseen publications/articles** resulting from the STSM: The state of the art on numerical modelling of TCC in fire and a step by step description of the modelling procedure in the computer SAFIR are to be included in the guidance document “Numerical modelling of timber-concrete composite structures in fire” prepared by the WG2 TG2 of the COST action FP1404, which will form part of the background documentation for the revision of EN 1995-1-2.

- Confirmation by the host institution of the successful execution of the STSM is attached (Annex 1)
Annex 1 – Confirmation by Host Institution

Confirmation from host

Date: 2018-05-18
Reference: Visiting research period
Page: 1 (1)

ETH Zürich
Institute of Structural Engineering (IBK)
Timber Structures
Stefano-Franscini-Platz 5
8093 Zürich
Switzerland

Cvetanka Chiffliganec – Visiting research period at ETH Zürich – Confirmation letter

In the name of the host institution, ETH Zürich, I the undersigned Dr. Michael Klippel confirm the successful completion of the visiting research period by Mrs. Cvetanka Chiffliganec.

The visiting research period took place between the 6th of May until 18th of May 2018.

The purpose of this research period was to contribute to the guidance document about the numerical modelling of timber-concrete composite structures in fire.

We thank Mrs. Chiffliganec for her work and are looking forward to future cooperation.

Yours sincerely,

ETH Zürich
IBK – Timber structures

Dr. Michael Klippel
Numerical modelling of timber-concrete composite slabs exposed to fire

Scientific report of STSM at ETH Zurich

06.05-18.05.2018

Author:
Cvetanka Chifliganec,
Ss. “Cyril and Methodius” University in Skopje,
Faculty of Civil Engineering – Skopje
# Table of Contents

Summary.........................................................................................................................................................1

Annex 1..........................................................................................................................................................2

Annex 2..........................................................................................................................................................3

Table of Contents..........................................................................................................................................4

1. Introduction .............................................................................................................................................5

2. Overview of performed work.....................................................................................................................6

3. State of the art on numerical modelling of TCC in fire................................................................. 8

3.1. TCC slab with screwed connections from Frangi [4] ................................................................. 9

3.2. Description of the model TCC floor system from O`Neill at al. [8] ..................11

3.3. Timber steel fibre–reinforced concrete floor slabs from Caldová et al. [12].13

3.4. Timber steel fibre–reinforced concrete floor slabs from Blesák et al. [16]...15

3.5. Wood-concrete composite deck under fire conditions in Meena et al. [19] 17

4. Numerical model of the TCC slab in SAFIR ................................................................. 19

4.1. Description of the model ...................................................................................................................... 19

4.2. Material models....................................................................................................................................23

4.3. Results from the numerical analysis and discussions ..................................................................25

5. Conclusions...............................................................................................................................................27

References....................................................................................................................................................28
1. Introduction

Modelling the behaviour of timber-concrete composites (TCC) in fire is a very complex process. Numerical models should provide a realistic analysis of structures exposed to fire. They should be based on fundamental physical behaviour in such a way as to lead to a reliable approximation of the expected behaviour of the relevant structural component under fire conditions. They can be used for determination of: (1) charring depth, (2) development and distribution of the temperature within structural members (thermal response model), and (3) evaluation of the mechanical behaviour of the structure or of any part of it (structural response model) [1].

In general, to describe the behaviour of a TCC structure in fire a nonlinear numerical analysis has to be conducted. The nonlinearity of the problem comes from: the changes in material properties by high temperatures (mechanical and thermal), the nonlinear temperature distribution in the element’s cross-section and the continuous change of internal forces in the structure. Also, as the structural behaviour of TCC is mainly governed by the shear connection between the timber and the concrete, proper simulation of the connector’s behaviour is of great importance for delivering accurate results.

Nowadays, there are several computer programs capable of calculating the thermal and mechanical behavior of TCC structures in fire. The validation of the advanced numerical calculation models in these programs should be made based on comparison with experimental results, analytical solutions or other computer codes. But, even though an increased number of large-scale and small-scale testing of TCC structures in fire are performed in recent years (see section state-of-the-art), numerical analyses are conducted only for a small number of the them.

For the Expert Meeting of WG2 TG2 of COST Action FP1404, held in Skopje in October 2017, I presented one numerical model of a TCC slab in fire in SAFIR. Several analyses of the TCC slab were conducted in order to compare the numerically achieved results in SAFIR with the experimental data and the results obtained by the simplified calculation method (based on the calculation model for mechanically jointed beams with flexible elastic connections given in EN 1995-1-1 [2] and the reduced cross
section method given in EN 1995-1-2 [3]). In that model, the TCC slab was treated as simply supported beam. Two different 2D models of the slab with different cross-sections (T and TTTT) were made. The steel connectors were incorporated in the 2D models, but the connection between the timber beam and the concrete slab could not be treated as flexible and the slip between the subcomponents could not be calculated.

Since improvement of the numerical models was necessary, this STSM was the perfect opportunity to do that at the institution where the fire tests of this particular TCC slab were conducted. The aim of this STSM was to collect all necessary input data and to conduct a numerical analysis of the TCC slab with screwed connection in fire tested by Frangi et al [1] [4] [5]. The numerical analysis was conducted in the computer program SAFIR, a special purpose computer program for the analysis of structures under ambient and elevated temperature conditions, developed at the University of Liège, Belgium by Jean-Marc Franssen and Thomas Gernay [6] [7].

2. Overview of performed work

During this STSM, a visit to the laboratory of the ETH Zürich, Institute of Structural Engineering (IBK) was done. There, I was presented with: the testing instruments and equipment, some previously tested specimens and elements, and the ongoing timber experimental setups. I attended to one experimental testing, conducted by Lars Ackermann and Joachim Schmid, for the purpose of Ackermann’s master thesis “Design fires - Description of charring in non-standard fires” (Figure 1). Also, I visited Empa’s Fire Technology Lab (EMPA - Swiss Federal Laboratories for Materials Science and Technology) (Figure 2).
Figure 1. Look of the test specimen through the fire chamber window at a time moment during the 20 minutes’ fire exposure

Figure 2: Empa’s Fire Technology Lab furnaces for fire testing of slabs, walls, beams, columns etc.

Also, a state of the art on numerical analysis of TCC in fire was done, presented in Section 3 of this STSM report. This state of the art will be revised and completed by other members of the WG2 and afterwards it is intended to be incorporated in a guidance document for numerical analysis of timber-concrete composite structures in fire.

A new numerical model for analysis of the TCC slab with screwed connections in fire in the computer program SAFIR was developed. This numerical model is presented in Section 4.
3. State of the art on numerical modelling of TCC in fire

Modelling of the behaviour of timber-concrete composites in fire is a very complex process. Numerical models should provide a realistic analysis of structures exposed to fire. They should be based on fundamental physical behaviour in such a way as to lead to a reliable approximation of the expected behaviour of the relevant structural component under fire conditions. The can be used for determination of: charring depth, development and distribution of the temperature within structural members (thermal response model), evaluation of structural behaviour of the structure or of any part of it (structural response model) [3].

In general, to describe the behaviour of a TCC structure in fire nonlinear numerical analysis has to be conducted. The nonlinearity of the problem comes from: the changes in material properties by high temperatures (mechanical and thermal), the nonlinear temperature distribution in the element cross-section and the continuous change of internal forces in the structure. Also, as the structural behaviour of TCC is mainly governed by the shear connection between the timber and the concrete, proper simulation of the connector’s behaviour is of great importance for delivering accurate results.

Nowadays there are several computer programs capable of calculating the thermal and mechanical behavior of TCC structures in fire. Validation of the advanced numerical calculation models in these programs should be made based on comparison with experimental results, analytical solutions or other computer codes. But, even though increased number of large-scale and small-scale testing of TCC structures in fire are performed in recent years, numerical models are made only for a small number of the tested TCC.

Table 1 presents a summary of the state-of-the-art on numerical modelling of TCC structures exposed to standard fire in recent years. These numerical analyses were performed for the purpose of delivering results for particular experimentally tested TCC structures. The table lists: the types of TCC structures analyzed, the computer programs used for numerical analysis, the research groups and the corresponding
literature references. In the following subsections, more detailed descriptions of each numerical analysis is given.

Table 1: Summary of the state-of-the-art on numerical analysis of TCC structures exposed to fire in recent years.

<table>
<thead>
<tr>
<th>Reference</th>
<th>Year</th>
<th>Computer program</th>
<th>Type of numerical analysis</th>
<th>Verification with experiment in reference No.</th>
<th>Materials of TCC</th>
<th>Connection type</th>
</tr>
</thead>
</table>

3.1. TCC slab with screwed connections from Frangi [4]

Short description of the TCC slab:
- Span 5.21 m and width 2.8 m, exposed to standard ISO fire until failure.
- Slab consists of a glulam beam GL24h with cross-section dimensions of 180 mm x 240 mm, a 20 mm thick timber board and an 80 mm thick concrete slab.
- The screws in the connection have 45° of inclination. The distance between the screws, with 100 mm of its length embedded in the timber and 50 mm in the concrete, is 120 mm, and the side cover of the screws is 50 mm.

- The mechanical load is kept constant during the fire test and the slab is designed to survive 60 minutes of fire exposure.

Several researchers did numerical analysis of the TCC slab with screwed connections experimentally tested by Frangi [4].

In [4], a thermal numerical analysis of the timber sections was done, using the program SAFIR. The results of the numerical analysis (the temperature fields) were in good correlation with the experimental results.

In [20], the finite element program ANSYS [15] was used for a 3D sequentially coupled thermal-mechanical analysis of the above mentioned TCC structure. The data for the thermal and mechanical properties of concrete and timber were adopted from the EN 1994-1-2 [21] and EN 1995-1-2 [3], respectively. First a thermal analysis was conducted to determine the temperature field distribution of the TCC beam under fire. The element type used in the thermal analysis was Solid 70. Boundary conditions on the surfaces subjected to fire were set by radiation and convection according to the ISO 834 standard fire curve. Then, the mechanical analysis of TCC beam was made by importing the results of thermal analysis into the structural model. During the mechanical analysis, the thermal element for timber and concrete was replaced by Solid 185 and Solid 65, respectively. The shear connectors were set by the element Combination 39. The results for the temperatures in the cross section and the deflection of the structure were in good agreement with the experimental results from [4], but this model did not calculate and comment the slip between the concrete slab and the timber beam.
For the TCC structure with crewed connections from [4] the guidance document in preparation will give an overview of the complete modelling process in several computer programs: ABAQUS, ANSYS, SAFIR and one self-developed program by Tomaž Hozjan.

3.2. Description of the model TCC floor system from O’Neill at al. [8]

The composite floor, presented in Figure 4, is a semi-prefabricated system comprising of “M” panels that are built with laminated veneer lumber (LVL) beams and sheathed with a thick plywood interlayer, which acts as a permanent formwork for the concrete. The plywood interlayer has holes cut into it to accommodate the notched form of connection being used between beams and the concrete slab. The concrete slab is both ways reinforced. In the test two specimens were tested. The first floor specimen tested was the 300 mm beam floor, which was tested to destruction. Failure occurred at 75 minutes under the ISO 834 design fire and applied design load. The side with notched connections failed first, and the testing was terminated. The second floor was the 400 mm beam floor and this test was stopped shortly after 60 minutes to assess the damage [8].

![Image of the floor system](image)

**Figure 4:** LVL-concrete composite floor system from O’Neill et al. [6]

In [8], a 3D sequentially coupled thermo-stress analyses in ABAQUS is conducted in order to determine the effects of a fire on floor assemblies under load. This involved a thermal analysis to determine the temperature profile of the floor assemblies for the
duration of modelling, and then a stress analysis using the temperature profile as an input into the structural model.

For the 3D thermal modelling of the floor, the temperature distribution in the cross section was computed as an uncoupled heat transfer analysis using 8 node linear solid elements, DC3D8.

The floor beams were subjected to 3 sided exposure to the ISO834 fire as a standard temperature input into the models, and this is also applied to the underside surfaces of the concrete floor slab. This was applied via surface film conditions and surface radiation to the exposed surfaces, and ambient free convective surface conditions were modelled on the top of the slab. The convection coefficient and emissivity were assumed to be 25 W/m2K and 0.8, respectively. The cross section was discretized into a 5 mm mesh, and along the length of the floor the mesh was left very coarse at 300 mm.

In the subsequent 3D structural modelling, the same mesh from the thermal analysis was imported into the structural model (a cross sectional mesh size of 5 mm, lengthwise of 300 mm), and the element type used was an 8 node 3D linear solid element, C3D8R for both the timber and concrete portions of the floor. To consider reduction in the mechanical properties of timber with temperature, the values for reduction of the strength and modulus of elasticity were taken from the EN 1995-1-2 [3]. The timber is considered to behave in a brittle manner in tension, and to exhibit elasto-plastic behaviour in compression. The material model being used to characterize these stress-strain relationships for timber is the concrete damaged plasticity model, which allows for separate stress strain curves to be defined for both compression and tension. Plasticity is accounted for in the timber in compression via specifying arbitrary values of plastic strain at the maximum compressive stress, however in tension it is assumed to behave in a brittle manner and a strength reduction of 99% is imposed once the maximum tension stress is reached. The values for the thermal and mechanical properties of the concrete slab were taken from the EN 1992-1-2.

The modelling of the floor structure was conducted in two phases. Firstly, the concrete slab was left cold, and secondly it was heated. This was to define the influence of heating the concrete slab on the behaviour of the floor as a whole.
For the 300 mm beam floor the failure was recorded at 75 minutes, and from the Figure 5a it can be seen that this is predicted well by the hot concrete model which specifies a failure of approximately 74 minutes. For the 400 mm beam floor, the cold concrete model predicted failure at approximately 85 minutes, while the hot concrete model is slightly more conservative predicting failure at 81 minutes (Figure 5b). The displacement response form the numerical analyses confirmed that the deeper joist floor is much stiffer. From the analyses, it is found that the prediction of fire resistance by displacement compares relatively well to the experimental results as long as the thermal response of the concrete slab is incorporated.

3.3. Timber steel fibre–reinforced concrete floor slabs from Caldová et al. [12]

The experimentally tested and numerically analyzed composite timber fibre-reinforced concrete floor by Caldová et al. [12], presented in Figure 6, is composed of timber frame (glulam timber), two secondary beams (glulam timber), and a concrete slab connected to floor joists. As connectors, screws inclined 45° to the beam axis in two rows were used. The distance between the screws in one row was 0.1 m.

The 3D coupled thermo-mechanical numerical analysis was conducted in the FE program ANSYS. The numerical model represented a symmetrical portion of the structure and FE discretization with solids was used.

In the thermal analysis, the heat transfer in the structure was determined as a transient nonlinear problem with an implicit Newmark’s integration. The boundary conditions on the surfaces exposed to fire used radiation and convection based on measured furnace temperatures from the experiments and the surfaces outside the furnace had a
constant temperature of 12°C. To obtain the thermal properties (thermal conductivity, specific heat, and heat transfer coefficients) of SFRC and timber that were used in the numerical analysis, starting values in the material data from EN 1992-1-1 [22] for plain concrete and EN 1995-1-1 [2] for timber were used. The results obtained for these starting data sets were afterward improved and calibrated in order to determine the best fit to measured data from the experiments [12].

The material model used for the timber was the transversally isotropic material law with isotropic plasticity. The strength characteristics for glue laminated timber GL24h (modulus of elasticity parallel to grain 11 600 MPa, modulus of elasticity perpendicular to grain 390 MPa, shear moduli 720 MPa and tensile strength parallel to grain 16.5 MPa) were adopted from EN 1995-1-1 [2]. The charring of timber was taken into consideration through reduction of the modulus of elasticity and the tensile strength, according to the EN 1995-1-2 [3]. For the coefficient of thermal expansion a constant value of $4.5 \times 10^{-6} \degree C^{-1}$ was used.

For the steel fibre–reinforced concrete in fire, the microplane material model (ANSYS) was used. This model is suitable for modelling of damage due to extensive loading and thermal degradation. The material data for the microplane model were calibrated to measurements obtained from four-point bending tests at ambient and elevated temperatures and standard cube compressive tests. The thermal expansion of SFRC was taken as for siliceous concrete from EN 1992-1-2 [3].

The shear behavior of the screw connection was simulated based on push-out test. The model of the screw connection assumed that the screws are smeared to a special bonded contact model, which can carry shear load with a defined bilinear model of strength, 1.1 MPa, for 0.0012 sliding distance [13].

The mechanical analysis of the composite floor slab was performed as a geometrically nonlinear static analysis. The Newton-Raphson method was applied to determine the solution of the nonlinear static analysis. The results from the numerical analysis were verified with the measured displacements and the fit to the measurement was satisfactory. The failure of the structure was not observed in the analysis because of the residual strengths of SFRC.

The numerical simulations by Calová et al. [12] showed once again that the use of adequate material models is essential to obtain accurate simulations.
3.4. **Timber steel fibre–reinforced concrete floor slabs from Blesák et al. [16]**

In Blesák et al. [16], the behaviour of one of the tested slabs: ELE-2-100/160, described in Caldová et al. [12], was numerically simulated using the software Atena Science [17] [18].

The overall numerical analysis was based on the outputs of two analyses – thermal and static.

In the thermal analysis, the properties of timber were modified in compliance with [23] and considered constant. The thermal properties of concrete were taken from [24] but modified for the purpose of the analysis. The fire curve ISO 834 was used to fire-load the structure from the bottom edge and the temperature on the top edge was defined.
to be 25°C, the same as the initial temperature of all the used materials. The convection heat transfer coefficient was considered 50 W/(°C.m2), resulting emissivity factor of the radiation source and heat transfer 0.7 [24]. As the thermal gradient across the slab thickness is non-linear, three layers of shell elements placed one on another were used in the numerical simulation. The thermal data were transported into the static calculation considering the surface points of the shell elements (Figure 8).

In the structural analysis, only one quarter of structure was modelled applying the appertaining symmetry conditions. The structural system is point-supported as depicted in Fig. 8 b). Point support enables the supporting frame elements to rotate when being affected by the slab deformation and so torsion stiffness variation along the frame element`s length was taken into account.

The mechanical properties of glued laminated timber were considered to be decreased to zero when reaching temperature 300° C [23], except the decrease of the Young`s modulus of elasticity which was considered to decrease linearly from its initial value down to 50 GPa. The author`s explanation for this is that taking zero value into account would cause divergences in the numerical model. The burn out of timber was considered by applying an effective Young`s modulus of elasticity. The Young`s modulus of elasticity of timber was controlled in time, not as a function of temperature, which showed to be sufficient and proper for the numerical analysis. Charring rate of 0.5 mm.min⁻¹ [23] is taken into account in the frame of the analyses.

The element sizes were opted taking the computational efficiency into account – computational time and results precision. For the slab, shell elements with thickness 20 mm and ground-plan dimensions 150 x 150 mm were applied; for the internal beam, four solid elements in height and two in width were applied. For the timber material SOLID 3D elements were used and for the SFRC material SHELL 3D elements are used. The SHELL 3D elements in the software Atena Science are numerically defined as shells but may be connected to SOLID 3D elements, which enables many specific operations to be applied [17] [18].
The numerical analysis by Blesák et al. [12] gave the maximal vertical deflection of the coupled timber-SFRC system and the major cracks opening pattern, both in good correlation with the experimental results (Figure 9).

3.5. Wood-concrete composite deck under fire conditions in Meena et al. [19]

Meena et al. [19] conducted experimental investigations on a small section of a wood-concrete composite deck exposed to fire, presented in Figure 10. The timber beams were made of GL24h (dimensions: 160/80) and the concrete deck was of self-consolidating concrete (C60/65). The connection of the slab to the beams, penetrating 30 mm into the slab, was with 8 grooves. The specimen was subjected to the ISO-834 standard temperature-time curve with the concrete slab exposed to fire.
In order to obtain the temperature profiles in the materials of the specimen under fire conditions, Meena et al. [16] conducted only thermal numerical analysis. For the numerical analysis, the commercial software package Ansys Workbench v12 was used. An isometric view of the model can be seen in Figure 11. Only a portion of the test specimen was modelled, as the temperature variation across the bottom surface of the system was determined to be nearly identical during the conducted fire tests. Contact sizing, with element size 10 mm, was used to refine the mesh at the contact surface of the timber beam and the concrete slab.

When describing the thermal properties of the materials the author just points that thermal conductivity and specific heat of the different materials for the modelling purpose are taken from the literature [25] and gives the data from Figure 12.
4. Numerical model of the TCC slab in SAFIR

4.1. Description of the model

The subject of analysis was the TCC slab with crewed connections in fire tested by Frangi et al. [1] [4] [5]. The cross-section and the longitudinal section of the slab are once again presented in Figure 13.

![Figure 13: Cross section of the TCC beam](image)

The nonlinear 2D numerical analyses were conducted with the computer programme SAFIR specialized for structural fire analysis [6] [7]. The analysis of the structure exposed to fire consist of two steps. The first step involves predicting the temperature distribution inside the structural members, termed as ‘thermal analysis’. The second part of the analysis, termed as ‘structural analysis’, is carried out for the main purpose of determining the response of the structure due to static and thermal loading.

In the structural analysis, the TCC beam was treated as a simply supported Virendeel truss. The top chords represent the concrete slab, the bottom chords represent the timber beam and the verticals/webs represent the connections (Figure 14). The vertical steel beam elements are placed at the exact position of the screws through the length of the tested TCC beam.

![Figure 14: Structural system of the modelled TCC slab and loads](image)

The cross sections of the concrete slab, the timber beam and steel verticals are presented in Figure 18a, Figure 19a and Figure 19b, respectively. The interlayer between the concrete slab and the timber beam (timber board with thickness of 20 mm) was not modelled but it’s positive influence on the temperature profile in the
concrete slab was taken into consideration by assigning a temperature-time curve to describe the evolution of temperature at the bottom side of the concrete slab.

The main goal in this numerical model was to properly model the connections. According to Frangi [4], one pair of the 45° inclined screws has: characteristic load-carrying capacity $F_{v,\text{rk}}=12.6$ kN and slip modulus $K_{\text{ser}}=18$ kN/mm. According to ETA-13/0699 [26], these screws have: $F_{v,\text{rk}}=18.38$ kN and $K_{\text{ser}}=17.21$ kN/mm. In the numerical model, the steel verticals representing the connections are modelled as steel beam elements with the same load carrying capacity as the 45° inclined screws in Frangi [4].

The model in the structural analysis consisted of 168 beam elements. In each time step, SAFIR uses an iterative procedure to converge to the correct solution. A precision value of 0.0001 and the modified Newton-Raphson convergence procedure were used in the analyses. For solving the system of equations the method of Paradiso, based on a space matrix solver, was used. In SAFIR there is no deflection criterion for defining the failure point, therefore the calculation goes on until the failure of the whole structure happens. Local failure of a structural member that does not endanger the safety of the whole structure is handled by the means of the dynamic analysis [6].

In the thermal analysis, first, the cross-section of the TCC beam was analysed as a T-section with standard fire exposure (ISO 834) on the bottom side (Figure 15). The main purpose of this analysis was to calculate the temperatures in the screws and in the concrete slab which is protected from direct fire with timber board. Afterwards, the temperature-time curve which describes the evolution of temperature in one particular node of the screw (Figure 16) was applied to the outer boundary of the steel verticals. This node is positioned at about half of the length of the screw in the timber and it’s circled with red in Figure 15c. For the concrete slab, the node with the highest temperatures in time was used to describe the evolution of temperatures due to fire in the slab (rounding of isotherms occurs near the corners at the contact between the concrete slab and the timber beam and nodes there have lower temperatures). This temperature-time curve (Figure 17) was then applied at the bottom frontiers of the concrete slab presented with green in Figure 18b.
Figure 15: a) Cross section of the TCC beam modeled as a T-section b) Temperature field in the cross-section at the moment of failure of the structure (56 min) c) Temperature-time curves in some particular nodes along the length of the screw in the timber

Figure 16: Comparison of temperatures around the screw calculated with: SAFIR for one particular node and the expression for calculation of temperature profile in a wood member subjected to ISO fire exposure from one side given by Frangi et al. [5] for $x=50$ mm
Figure 17: Temperature-time curve describing the evolution of temperature at the bottom side of the concrete slab.

Figure 18: a) Cross section of the TCC beam (dimensions in meters) b) Fire exposed surfaces c) Temperature field at the time of failure (t=56 min) and applied mesh.
In the thermal analysis, SOLID elements were used to model the cross-sections. The number of solid elements was: 1424 for the concrete slab (Figure 18c), 3884 for the timber beam (Figure 19a) and 260 for the steel verticals (Figure 19b). After several analyses were conducted, it was concluded that the increase of the solid elements i.e. decrease of the mesh size in the cross-sections did not influence the results of the structural analysis.

4.2. Material models

For concrete, steel and wood, the thermal models are based on the corresponding Eurocodes.

For concrete - SILCON ETC [27], the material model is based on the laws of EN 1992-1-2 [22] except that in the ETC model the transient creep strain treated by an explicit term in the strain decomposition.

For structural carbon steel (used for the steel verticals), the material model, based on EN 1993-1-2 [27], is elastoplastic with a limiting strain for yield strength of 0.15 and an ultimate strain of 0.20.

For reinforcing carbon steel (used for the reinforcement in the concrete lab), the material model is based on the EN 1992-1-2 [28].
For describing the temperature-dependent mechanical behavior of wood the uniaxial material model of Annex B of EN 1995-1-2 [3] is used. The strength and stiffness start to decrease as soon as the temperature exceeds 20°C and they reduce to zero at 300°C. Hence, the charring depth can be estimated based on the position of the 300°C isotherm in the section. In the range 20-300°C, different reduction factors apply to tension and to compression for the strength and modulus of elasticity [6]. It is worth mentioning that this is the only predefined material model for wood in SAFIR.

The input data describing the mechanical and thermal characteristics of the materials used in the numerical analysis are presented in Table 2 and Table 3.

Table 2: Mechanical characteristics of materials

<table>
<thead>
<tr>
<th>Material</th>
<th>Mechanical characteristics</th>
<th>Experiment</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete</td>
<td>E [N/mm²]</td>
<td>37000</td>
<td>37600 (Silicon ETC)</td>
</tr>
<tr>
<td></td>
<td>f_c [N/mm²]</td>
<td>47</td>
<td>47</td>
</tr>
<tr>
<td></td>
<td>f_t [N/mm²]</td>
<td>?</td>
<td>2.73</td>
</tr>
<tr>
<td></td>
<td>ρ [kg/m³]</td>
<td>?</td>
<td>2400</td>
</tr>
<tr>
<td>GL24h</td>
<td>E [N/mm²]</td>
<td>10620</td>
<td>same</td>
</tr>
<tr>
<td></td>
<td>f_c [N/mm²]</td>
<td>24</td>
<td>same as in experiment [5]</td>
</tr>
<tr>
<td></td>
<td>f_t [N/mm²]</td>
<td>16.5</td>
<td></td>
</tr>
<tr>
<td></td>
<td>ρ [kg/m³]</td>
<td>431-445</td>
<td></td>
</tr>
<tr>
<td></td>
<td>ω [%]</td>
<td>13</td>
<td></td>
</tr>
<tr>
<td>Reinforcement steel</td>
<td>E [N/mm²]</td>
<td>?</td>
<td>200000</td>
</tr>
<tr>
<td></td>
<td>f_y [N/mm²]</td>
<td>?</td>
<td>500</td>
</tr>
<tr>
<td></td>
<td>d [mm]</td>
<td>6</td>
<td>6 /100/100mm</td>
</tr>
<tr>
<td>Connection</td>
<td>E [N/mm²]</td>
<td>?</td>
<td>210000</td>
</tr>
<tr>
<td></td>
<td>f_y [N/mm²]</td>
<td>?</td>
<td>400</td>
</tr>
<tr>
<td></td>
<td>d_{inner} [mm]</td>
<td>4.3</td>
<td>d=40 mm</td>
</tr>
<tr>
<td></td>
<td>d_{outer} [mm]</td>
<td>7.3</td>
<td></td>
</tr>
</tbody>
</table>
Table 3: Thermal characteristics of materials

<table>
<thead>
<tr>
<th>Material</th>
<th>T characteristics</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete</td>
<td>Convection coefficient on hot surfaces $\alpha_{\text{hot}}$ [W/m(^2)K]</td>
<td>25</td>
</tr>
<tr>
<td>Concrete</td>
<td>Convection coefficient on cold surfaces $\alpha_{\text{cold}}$ [W/m(^2)K]</td>
<td>4</td>
</tr>
<tr>
<td>Concrete</td>
<td>Emissivity $\varepsilon$</td>
<td>0.7</td>
</tr>
<tr>
<td>Steel</td>
<td>Convection coefficient on hot surfaces $\alpha_{\text{hot}}$ [W/m(^2)K]</td>
<td>25</td>
</tr>
<tr>
<td>Steel</td>
<td>Convection coefficient on cold surfaces $\alpha_{\text{cold}}$ [W/m(^2)K]</td>
<td>4</td>
</tr>
<tr>
<td>Steel</td>
<td>Emissivity $\varepsilon$</td>
<td>0.7</td>
</tr>
<tr>
<td>Timber</td>
<td>Convection coefficient on hot surfaces $\alpha_{\text{hot}}$ [W/m(^2)K]</td>
<td>25</td>
</tr>
<tr>
<td>Timber</td>
<td>Convection coefficient on cold surfaces $\alpha_{\text{cold}}$ [W/m(^2)K]</td>
<td>4</td>
</tr>
<tr>
<td>Timber</td>
<td>Emissivity $\varepsilon$</td>
<td>0.8</td>
</tr>
</tbody>
</table>

4.3. Results from the numerical analysis and discussions

This new numerical model for the TCC slab in fire was able to calculated the slip between the subcomponents of the composite (concrete and timber), but it overestimated its value. Also, the model in SAFIR underestimated the fire resistance and the deflection of the TCC slab.

The main results from the analysis are presented in Table 4. Comparison of deflection curves and slip curves between the numerical model and the experiment are presented in Figure 20 and Figure 21, respectively.

Table 4: Results from the numerical analysis

<table>
<thead>
<tr>
<th>Results from numerical analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fire resistance [min]</td>
</tr>
<tr>
<td>Deflection at failure [mm]</td>
</tr>
<tr>
<td>Slip at failure [mm]</td>
</tr>
</tbody>
</table>
Figure 20: Comparison between the calculated deflection in Safir, the measured deflection in the experiment and the calculated deflection with the simplified method proposed by Frangi et al. [5]

Figure 21: Comparison between the calculated relative slip at the end of the beam in Safir, the measured slip in the experiment and the calculated slip with the simplified method proposed by Frangi et al. [5]
Several attempts were made to improve the results of the numerical analysis.

First the varying parameter was the mesh size. It was concluded that the mesh size neither of the cross sections in the thermal analysis, nor of the structure in the structural analysis had an effect on the results.

In order to see if the stiffness of the steel verticals in the truss have an influence on the results, another parametric study was conducted. It was concluded that increasement of the diameter of the steel vertical above 40 mm just decreased the initial deflection, off course, but did not change the calculated fire resistance of the TCC beam.

5. Conclusions

The state of the art on numerical modelling of timber-concrete composite structure in fire showed that numerical modelling is done only for few experimentally tested TCC slabs in fire. This points out the need for more extensive research in this field.

A new numerical model for analysis of timber-concrete composite beam in fire was developed in SAFIR, but it underestimated the fire resistance and the deflection of the TSS structure and overestimated the relative slip at the end of the beam. After several different parameters were varied in the model, it was concluded that the response of the analysed TCC beam in fire was mainly governed by the behaviour of the timber beam in fire and not the connections. One possible explanation for the lower fire resistance of the model in comparison to the experiment could be that SAFIR uses the constitutive model for wood according to EN 1995-1-2, where linear stress-strain relations until failure for compression and tension are assumed. This has yet to be verified.

Improvement of the numerical model is already in progress.
References


