NUMERICAL MODELLING OF TIMBER CONCRETE COMPOSITE STRUCTURES IN FIRE - GUIDANCE DOCUMENT

Authors:

Chiara Bedon,
Department of Engineering and Architecture, University of Trieste, Italy

Cvetanka Chifliganec
Faculty of Civil Engineering, Ss. Cyril and Methodius University in Skopje, Macedonia

Igor Džolev,
Faculty of Technical Sciences, University of Novi Sad, Serbia.

Robert Pečenko,
Faculty of Civil and Geodetic Engineering, University of Ljubljana, Slovenia

Tomaž Hozjan,
Faculty of Civil and Geodetic Engineering, University of Ljubljana, Slovenia
Guidance on Numerical Modelling of Timber Concrete Composite Structures in Fire

COST Action FP 1404

“Fire Safe Use of Bio-Based Building Products”

N224-07

This publication is the result of work carried out within COST Action FP1404 “Fire Safe Use of Bio-Based Building Products”, supported by COST (European Cooperation in Science and Technology). COST is a funding agency for research and innovation networks. The Actions help connect research initiatives across Europe and enable scientists to grow their ideas by sharing them with their peers. This boosts their research career and innovation. More information available at www.cost.eu.

This guidance document may be cited as:


DOI 10.3929/ethz-b-000319551

Version 1 (2018)


This publication was created in WG2 and can be considered as state of the art-document. The information in the book was carefully selected-by experts from general technical knowledge, standards, most recent research results and building practice. The author(s), the editor(s) and publisher disclaim any liability in connection with the use of this information.

Neither the COST Association nor any person acting on its behalf is responsible for the use of the information contained in this publication. The COST Association is not responsible for the external websites or other references referred to in this publication.

No permission to reproduce or utilise the contents of this publication by any means is necessary, other than in case of images, diagrams or other material from the copyright holders. In such cases, permission of the copyright holder(s) is required.
Foreword

Timber-concrete systems (TCC) present one of the most effective bearing systems and have been extensively used in recent years. They can be used in construction of new buildings and bridges, as well as for rehabilitation and restoration of existing structures, where unsound material, damaged for various reasons, is replaced by a new composite system. While a lot is known about their behaviour at ambient conditions, knowledge about the fire resistance of TCC systems is less known and further research is needed. Aim of this document is to provide guidelines for numerical modelling of timber-concrete composite systems in various numerical software’s and thus, to enable further development and knowledge of behaviour of TCC systems in fire.

Acknowledgments

The authors gratefully acknowledge the European COST Action FP1404 on Fire safe use of bio-based building products. Authors would also like to thank other members of the action who participated at meetings of Task Group TG2 and their valuable comments.

Photo on cover page:


Bottom left: Numerical simulation of TCC system in ANSYS.

Bottom right: Thermal simulation of TCC system in HEATKO.
# Table of Contents

1. About the COST ACTION FP1404 ................................................................. 3
2. Introduction ................................................................................................. 4
3. Numerical modelling of TCC system in various software ..................... 6
   3.1 Short description of reference TCC system ...................................... 6
   3.2 Modelling of TCC in ANSYS ............................................................... 7
      3.2.1 Introduction of the ANSYS software ...................................... 7
      3.2.2 Numerical analysis of the reference TCC slab in ANSYS ........ 12
      3.2.3 Conclusion, recommendation and guidance ......................... 15
   3.3 Modelling of TCC in SAFIR ............................................................... 16
      3.3.1 Introduction of the SAFIR software ...................................... 16
      3.3.2 Numerical analysis of the reference TCC slab in SAFIR .......... 17
      3.3.3 Conclusion, recommendation and guidance ......................... 22
   3.4 Modelling of TCC in COMP-WOOD ............................................... 23
      3.4.1 Introduction of the COMP-WOOD software ....................... 23
      3.4.2 Numerical analysis of the reference TCC slab in COMP-WOOD 27
      3.4.3 Conclusion, recommendation and guidance ......................... 28
   3.5 Modelling of TCC in ABAQUS ......................................................... 30
      3.5.1 Introduction of the ABAQUS software .................................. 30
      3.5.2 Numerical analysis of the tensile test in ABAQUS ................. 33
      3.5.3 Conclusion, recommendation and guidance ......................... 37
4. Conclusions ................................................................................................ 39
1 About the COST ACTION FP1404

Bio-based building products have a very long history as timber structural members. However, due to its combustibility and several large fires in cities with timber buildings, timber as a structural material was banned for many years. When performance based design (PBD) was introduced and implemented in many building regulations, the market for timber structures opened again. And with it came the development of other bio-based building products. Still, the use of combustible building products is very limited in many countries. This is mainly due to the lack of knowledge on the properties of the materials, especially how the materials will influence the fire safety of the buildings. In addition, large differences in the regulations among various countries exist.

Modern living offers attractive, flexible buildings, and aims for cost efficient building techniques. In addition, the sustainability and environmental footprint of building products has become a very important issue. Consumers demand environmentally friendly and renewable products. At the same time, the Fire Safety of the end-product has to remain on a high level.

Fire Safety Engineering (FSE) has achieved large acceptance in the recent years. FSE allows a PBD with customized building solutions and novel materials. However, the available techniques and knowledge are often limited to non-combustible materials.

During the last decade, the portfolio of building products made from bio-based materials has increased enormously. The material properties that might affect a fire development vary, this has been confirmed in many projects, also European.

COST Action is a European co-operation for science and technology supported by the EU. COST Action FP1404 was created for building network on Fire Safety of Bio-Based Building Materials.

Intention of COST Action FP1404 was to create a platform for networking, exchange and collection of performance data, experiences, authority- and climate requirements, which affect the design with respect to the Fire Safe Use of Bio-based building products. Goal of working Group 2 – Structural Elements made of bio-based building materials and detailing – (WG2) within COST FP1404 Action was to provide guidance for the use and design of structural elements made from bio-based building products. Within WG2 several task groups were formed. Authors were members of Task Group TG2, which was dealing with fire design of timber-concrete composite systems and it is believed that this work represents a small contribution to a better understanding and faster development timber concrete systems in bio-based buildings as well as in other type of buildings.
2 Introduction

EN 1995-1-2 [1] is the European design standard for the design of timber structures exposed to fire. The current version of EN 1995-1-2 was published in 2004, and is now under revision. Fire design of timber-concrete (TCC) composites structures is not included in the current version of EN 1995-1-2.

In 2010, the European technical guideline Fire Safety in Timber Buildings was published. The guideline consists of new and updated design models for the separating elements and effective cross-section design for timber construction in fire, including timber-timber and timber-concrete (TCC) composites.

With respect to existing literature contributions, this guidance document provides the outcome of the research performed within the WG2 of the COST Action FP1404, including specific guidelines for the numerical analysis of TCC systems in fire conditions. The guidance document will form part of the background documentation for the revision of EN 1995-1-2.

Based on the literature review, the most thoroughly experimentally investigated systems are traditional TCC beams made from glued-laminated timber (glulam) or laminated veneer lumber (LVL) with screwed or metal plate connections. However, in recent years, TCC floor systems made of cross-laminated timber (X-Lam or CLT) or plywood became very popular. The experimental research in TCC is mainly focused on standard fire resistance testing of beams and slabs, while the behaviour of the connectors at elevated temperatures is rarely investigated. Among few, the most extensive study in this field is presented in [2],[7] where the behaviour of axially loaded self-drilling screws (tensile tests) and shear strength of the connection under fire conditions was investigated.

In Table 1 summary of the research related to most common TCC structures exposed to standard fire in recent years is given. The table lists: the material type, the connection type and the type of tests conducted by individual research groups.

<table>
<thead>
<tr>
<th>Reference</th>
<th>No. and type of tests</th>
<th>Material type</th>
<th>Connection type</th>
</tr>
</thead>
<tbody>
<tr>
<td>[2],[7]</td>
<td>23 large-scale tests</td>
<td>Glulam beam, concrete slab</td>
<td>Screwed and dowel connections with notches</td>
</tr>
<tr>
<td>[6]</td>
<td>15 small-scale, 1 large-scale</td>
<td>Glulam beam, concrete slab</td>
<td>Screwed connections</td>
</tr>
<tr>
<td>[3]</td>
<td>43 small-scale, 2 large-scale with 2 setups</td>
<td>LVL beam, concrete slab</td>
<td>Notched connection with steel screws and steel plates</td>
</tr>
<tr>
<td>[4],[5]</td>
<td>2 large-scale, 12 small-scale material tests for each SFRC mixture</td>
<td>Glulam timber frame, steel fibre reinforced concrete (SFRC) slab</td>
<td>Screwed connections</td>
</tr>
<tr>
<td>[8]</td>
<td>2 large-scale</td>
<td>Beech LVL, reinforced concrete slab</td>
<td>Notched connections</td>
</tr>
</tbody>
</table>
Numerical modelling of TCC structures in fire
Guidance document

The small, large and full-scale fire tests listed in Table 1 were conducted in standard fire conditions and represent an appropriate basis for the development of methods for the fire resistance calculation of TCC structural elements. It was found out that more experimental work still needs to be performed with focus on the behaviour of different types of timber-concrete connections at elevated temperatures. Currently plate connections are poorly investigated, whereas glued-in connections have been completely omitted. 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. As literature review revealed, numerical modelling for TCC beams exposed to fire was performed with several commercial software packages, but the reports are not so extensive and detailed. In general, the calculated deflections, failure times and failure modes agreed quite well with the experiments but, some specific phenomena like the slip between the subcomponents of the structure is not always investigated. Currently the field of numerical modelling of TCC structures still offers many opportunities for further development.

In what follows, procedures on numerical modelling of one particular TCC structure with the commercial software: ANSYS, SAFIR and self-developed software COMP-WOOD is given. Additionally, at the end of each subsection short summary including the recommendations and guidance for numerical modelling of the appropriate software is given in order to enable easier usage for researchers, students and engineers and to alert them to possible and common errors.

Validation study of all presented software is performed on the simply supported TCC beam with screwed connection exposed to ISO fire experimentally investigated in [7]. Details of the reference case are described in section 3.1 and validation results for each software are given in separate subsections. Additionally, at the end of section 3, guidelines for numerical modelling in ABAQUS are given, while example is focused on modelling the tensile test of connectors in fire as investigated in [6].

References


3 Numerical modelling of TCC system in various software.

3.1 Short description of reference TCC system

For the purposes of the numerical analysis and validation test, data and geometry of TCC beam-type slab with screwed connections are taken according to [1] (Fig. 1). The beam-type slab is composed of 4 glue laminated timber beams (w/h = 180/240 mm) and concrete deck (h = 80 mm) reinforced with K283 net reinforcement. The connection is made of VB-48-7.5x100 screws, which are inclined by 45° and placed in two rows as seen in Fig. 1. The beam-type slab is loaded with two point loads of F = 40 kN, thus each composite beam is loaded with F = 10 kN.

![Figure 1. Geometry and connector positions of TCC with screwed connection as in [1].](image)

References

3.2 Modelling of TCC in ANSYS

3.2.1 Introduction of the ANSYS software

ANSYS [5] is a general purpose finite element software, which can be used to simulate various engineering problems. The simulations are mostly performed using ANSYS Workbench platform [2], which consolidates different predefined Analysis System setups, based on the type of problem being analysed.

To assess the behaviour of timber-concrete-composite slabs subjected to fire, a nonlinear thermo-mechanical analysis needs to be conducted. Thermal and structural analysis can be coupled in a “weak” or “strong” manner. Weak coupling assumes that the temperature evolution inside the elements is independent on the deformations resulting from heating. Thermal analysis is conducted on undeformed structure and the results, in terms of temperature evolution in time for each finite element, are transformed to the subsequent structural analysis, as internal time-dependent body temperature. Strong coupling can be used to perform both thermal and structural analysis simultaneously, providing somewhat more realistic approach, but the limitations are related to the computational efficiency and the use of specific finite elements, which are not developed for certain types of materials, such as concrete. The project schematic workflow for “weak” or “one-way” coupling is presented in Fig. 2.

![Figure 2. Thermal-structural analysis procedure.](image)

Thermal and structural model are built independently. Each analysis system consists of individual cells:

- Engineering Data
- Geometry
- Model
  - Setup
  - Solution
  - Results
Numerical modelling of TCC structures in fire
Guidance document

Link connecting the solution from the thermal analysis to the setup of the structural analysis represent data transfer between two systems. The following chapters describe each type of analysis cell in detail.

**Engineering data**

Engineering data is used to define or access material models for use in an analysis. To perform the transient thermal analysis, density, specific heat and conductivity should be defined for the whole range of expected temperatures. In case of timber-concrete-composite (TCC) slabs, thermal properties for timber, concrete and steel can be adopted according to relevant Eurocode standards [9],[10], and transformed in the appropriate form.

For the structural analysis, the following mechanical properties need to be defined: material density, coefficient of thermal expansion, modulus of elasticity and Poisson’s ratio. There are many additional options for defining the material plasticity. Since large strains are expected and the loading is not cyclic by nature, bilinear or multilinear isotropic hardening plasticity is recommended. Not all material models can be accessed through the Engineering data. Some models, such as the concrete model, applicable with specific type of finite element, can be implemented only directly in the Model branch, using ANSYS Parametric Design Language (APDL) [4] command blocks. Workbench platform, therefore, has some limitations in advanced modelling using just graphic user interface (GUI) - additional functions need to be incorporated using APDL code. In case of TCC slab in fire, timber plasticity can be modelled as bilinear isotropic hardening. Steel and concrete plasticity can be defined as multilinear isotropic hardening plasticity, with additional features for concrete material, which include: shear transfer coefficients for open and closed cracks, uniaxial tensile cracking stress and uniaxial crushing stress as mandatory fields, and a few others, as optional. All of the above described thermal and mechanical properties of materials are to be defined as temperature-dependent.

**Geometry**

Geometry cell is used to create, import, edit or update the geometry model used for analysis. Although geometry is usually the same for thermal and structural analysis, it may also be modified. For example, non-bearing elements can be modelled to calculate the heat transfer in the thermal model, but can be omitted in the structural model. Geometry model can be prepared in any computer-aided design (CAD) system, and then imported, or it can be created using embedded software, such as ANSYS DesignModeler [3], which is fully parametric. The timber and concrete parts of the TCC slab should be modelled as solid bodies, while the screws and the reinforcement mesh can be modelled as line bodies. If the contact surface between the concrete and timber parts is modelled, the parts should be transferred as separate bodies to the Mechanical application, to allow for the definition of potential sliding of the surfaces in contact.

**Model**

The Model cell is associated with the Model branch in the Mechanical application and affects the definition of geometry, coordinate systems, connections and mesh branches of the model.
Previously defined geometry model consists of series of solid and line bodies, which need to be attributed to the appropriate material [1]. Lower or higher order elements can be used to define the finite element mesh, by keeping or omitting the element midside nodes. If lower ordered elements are chosen, the default solid and line elements used in thermal analysis are SOLID70 (three-dimensional 8-node element) and LINK33 (2-node line element), respectively, with a single degree of freedom, temperature, at each node. Connections between solid and line bodies can be established as coupled, using tie constraints, which are accessible through the APDL commands. To be able to tie the nodes belonging to solid and line bodies, the nodes should be coincident within an appropriate tolerance, which is important for the mesh generation. The coupling of nodes is implemented in the Model Setup branch via APDL command block.

In the structural analysis, default solid and line elements, in case of using lower order elements, are SOLID185 (three-dimensional 8-node element) and BEAM188 (2-node line element), respectively. While timber parts can be modelled using SOLID185 elements, the concrete material model is applicable only with SOLID65 elements. Conversion from SOLID185 to SOLID65 element is made through the APDL command block assigned to each concrete body in the Geometry branch of the Model. SOLID65 is a three-dimensional 8-node element with three degrees of freedom at each node: translations in the nodal x, y and z directions. The element is capable of cracking in tension and crushing in compression. Cracking is permitted in three orthogonal directions at each integration point. The presence of a crack at an integration point is represented through modification of the stress-strain relations by introducing a plane of weakness in a direction normal to the crack face. The open and close crack shear transfer coefficients, \( \beta_t \) and \( \beta_c \), present the amount of shear force transferred through opened and closed crack. Tensile behaviour of concrete before cracking is assumed to be linear elastic. The concrete material can be implemented using nonlinear constitutive model of William and Warnke [13], combined with isotropic hardening plasticity, following the stress-strain curves provided in EN 1992-1-2 [9]. The stress-softening after reaching the tensile cracking stress, is presented by a sudden reduction of the tensile stress to 60% and a linear descending to zero stress at a strain corresponding to 6 times the strain value at a maximum tensile stress.

**Model Setup**

Model Setup is used to define loads, boundary conditions and the numerical parameters to configure the analysis in the application.

A series of Construction Geometries can be created, including Paths and Surfaces, which can be addressed to in the post processing of the results, if the aim is to present the results along some specific path or cross-section. The definition of Path or Surface is related to specific Coordinate System that can be assigned through the Coordinate Systems branch. If the model geometry and loading scheme are symmetrical, leading to symmetrical response of the whole system, planes of symmetry can be introduced, scoped to specific Coordinate System and the Symmetry plane Normal. The use of symmetry is highly recommended, since it reduces the number of degrees of freedom, which greatly influences the overall computational time. The nonlinear transient analysis, both thermal and structural, is performed in multiple load steps, each comprised of additional substeps, which are iteratively solved. The system matrix inversion in each iteration is
computationally demanding and the optimisation of the matrix size is an important issue in assessing the practical functionality of the model, which is influenced by the geometry Mesh [6].

In the application of TCC slab in fire, the timber and concrete parts are meshed with solid elements, and the screws and reinforcement mesh are meshed with line elements. The quality of the mesh can be assessed by the Mesh Metrics, in terms of element quality, aspect ratio, Jacobian ratio, warping factor, parallel deviation, maximum corner angle, skewness and orthogonal quality. Since nonlinear analysis is performed, it is recommended that the initial mesh consists of high quality elements.

The structural self-weight can be calculated implicitly, based on the material density provided in the Engineering data, the standard Earth gravity acceleration and the structural volume. Additional external loading can be defined acting on a node, line or surface of the elements. The same also applies when defining the boundary conditions in terms of support.

The Analysis Settings provide the information on the solution options that will be implemented by the solver and is crucial to the effectiveness of the computation time. The overall analysis is performed in multiple load steps. Due to the nature of external thermal loading (ISO 834 fire curve), the initial steep temperature rise should be solved using smaller time steps. The size of time steps can be defined by the number of substeps per load step, or by the duration of substeps within the particular load step. A constant number can be assigned, or an upper and lower boundary values, enabling the solver to choose within the established boundaries, depending on the convergence issues. If the convergence criteria are not met at the end of particular substep and the substep number is defined by the boundary values, a bisection occurs, reducing the time step applied in the analysis, until the criteria are met. Failing to converge for the substep boundary value, the analysis will stop.

The heat transfer analysis can be performed by means of conduction, convection and radiation. Thermal loads, in terms of convection and radiation, are applied using SURF152 surface elements. Convection coefficients for exposed and unexposed surfaces, as well as emissivity factors can be assumed based on Eurocode standards, and an arbitrary heating curve can be assigned. To optimize the computational time, the overall fire duration should be divided into a number of load steps consisting of an optimal number of substeps. The aim is to calculate the response in as few steps as possible, provided that the solution converges in each substep of the analysis. If the time step is too large, additional iterations will be required to reduce the error below the criteria value, provided that the proposed limit number of performed iterations is sufficient for the solution to converge. On the other hand, if the time step is too small, a problem of the appearance of space oscillations of the solution, in case of thermal shock, can also occur [7]. In regions of severe thermal gradients during a transient analysis, the largest element size in the direction of the heat flow should be related to the smallest time step size, to provide accurate results. Using more elements for the same time step size will normally produce better results, but using more substeps for the same mesh could often lead to less accurate results.

Unlike the thermal analysis, with only one degree of freedom at each node (temperature), in the structural analysis, there are three degrees of freedom at each node of the element (translations:
x, y and z), adding more complexity to the solution. The concrete material model, with inherent nonlinearities, requests defining a large number of small sub steps to be implemented, in order for the solution to converge. The first convergence difficulties arise with the exceedance of the concrete tensile strength and the formation of cracks, which occurs already in the early stage of the analysis. The nonlinear iterative process of solving nonlinear equations uses the Newton-Raphson method, which could be improved by adding the Line Search option. Although the problem is essentially quasi-static, it is recommended to use the transient analysis to overcome the convergence difficulties. The transient structural analysis also requires the input of the Damping Controls, for which the Rayleigh damping can be used. The stiffness and mass coefficients can be calculated based on the modal response of the structure, for the range of dominant response frequencies, significantly contributing to the vibrations. To determine the frequency range of the structure, an additional Modal analysis should be performed.

**Model Solution**

Solution cell is used to access the Solution branch of the Mechanical application. Also, the cell is used to share solution data from one analysis system as input conditions to the other.

**Model Results**

The Results cell indicates the status of the analysis results. The post processing of the results depends on the type of analysis being performed. In case of thermal-structural analysis, the temperatures obtained in thermal analysis are specified as internal body temperature in the subsequent structural analysis. In the Solution branch of the structural analysis, results can be extracted in the form of displacements (total or directional), stresses, strains (which can be decomposed into elastic, plastic, total mechanical and thermal), internal forces along the previously defined cross section and many other, depending on the available data. Results can be specified for a particular time step during the analysis or in the form of time-history array, providing the desired type of result in each substep during the course of analysis.

**Results verification and validation**

During the process of developing the numerical model, it is essential to verify and validate the obtained model results. Verification aims at checking whether the mathematical model is solved correctly using proposed numerical procedures, while validation is usually performed by comparing the results obtained by calculation with the results of experimental analysis, in order to conclude whether the mathematical model is a good representation of the problem physics.

**Conclusion**

ANSYS software package provides a wide range of possibilities in simulating thermo-mechanical response of structures subjected to fire. The proposed modelling description aims at providing general guidelines for modelling TCC slabs subjected to fire. Detail instructions on specific model creation depend on the modelling objectives and the balance of the available computational resources against the desired precision of results.
3.2.2 Numerical analysis of the reference TCC slab in ANSYS

Thermo-mechanical numerical model is developed in ANSYS to simulate the bending test experiment on a timber-concrete composite slab exposed to standard ISO fire, performed by Franghi et al. [11] at the Institute of Structural Engineering of ETH Zurich.

Numerical model consists of the thermal and structural models, sequentially coupled. Thermal model is developed to calculate the heat transfer within the structural elements, due to standard ISO 834 fire, by means of conduction, convection and radiation, according to EN 1991-1-2 [8]. Structural response is analyzed in two steps. In the first step, gravitational load is applied, consisting of self-weight and additional live and permanent load. In the second step, external loads are restrained and internal body temperature is introduced, simulating thermal response of the structure during fire.

Thermal and mechanical properties of constitutive materials are adopted according to available data provided by the authors. Data which is not presented in the paper but is necessary to be implemented in the numerical model, is adopted according to Eurocode standards [9],[10], based on the recommendations for use in the advanced calculation method.

Finite element models consist of 3D solid (concrete slab and timber beams/boards) and 1D line elements (steel reinforcement mesh and screws). Different finite element meshes are used in thermal and subsequent structural analysis, based on the physical phenomenon. To reduce the computational time, only 1/4 of the structure is analyzed, taking into account a two-plane symmetry.

In the initial, simpler model, all connections between structural parts are modelled as bonded, providing stiffer response and structural failure after 75 minutes, with 10% longer fire resistance than reported during the experiment. A more realistic model, introducing a contact zone between the concrete slab and timber boards is established, providing bonded contact during thermal analysis and frictionless contact for a structural analysis, allowing free sliding of the surfaces in contact with a possibility of separation. Contact between steel reinforcement mesh and the surrounding concrete elements, as well as between screws and the surrounding concrete and timber elements, is modelled as bonded. A tie constraint is applied, preventing the screws withdrawal, i.e. providing no slip between line and solid elements. At the contact surface, a threaded part of the screws entering timber elements, is connected with the upper part embedded in concrete, by joints, fixing all translational and rotational degrees of freedom.

Thermal response of the structure (Fig. 3), compared to the test results provided by the authors, provides a slightly underestimated temperature rise in timber elements (Fig. 4). The maximum calculated temperature in the outer screw at the lowest point (Fig. 5) after 65 minutes of standard fire exposure is at the magnitude of 70°C (Fig. 6), which does not affect the mechanical properties of the screws. The highest calculated temperature in the reinforcement mesh, at the same time, is around 100°C (Fig. 7), also not affecting the load bearing capacity of steel elements. Maximum temperature calculated in the concrete slab, after 65 minutes of fire exposure is around 200°C, providing minimum reduction of the concrete compressive strength (5%), but more than 50% reduction of the Young’s modulus, thus softening the structure (Fig. 8).
Numerical modelling of TCC structures in fire
Guidance document

**Figure 3.** Thermal response of the TCC slab.

**Figure 4.** Temperature evolution in timber as a function of the distance from the exposed surface [11].

**Figure 5** Control points of the screw assembly.

**Figure 6** Temperature evolution in the screws.
Deflection is mostly governed by the thermal expansion, the reduction in mechanical properties of timber beams, which are directly exposed to fire, and at the later stage of fire, by the slip between concrete and timber parts of the structure. Comparison between calculated and measured deflection and slip are presented in Fig. 9 and Fig. 10, respectively.

In the simplest model, the TCC slab is modelled without screws, but with a bonded connection between timber and concrete. Adding screws with a perfect bond between the surrounding timber and concrete elements but preserving the bonded timber-concrete connection, does not affect the deflection-time curve and yields the same conclusion on the fire resistance. By introducing the frictionless contact, allowing sliding between timber and concrete parts, relatively good agreement with the measured data is provided, in terms of fire resistance, but slightly larger deflection is calculated than measured during fire.

Unlike in case of vertical arrangement, by arranging the connectors at ± 45° inclination, a virtual truss is formed with the timber and the concrete as girders and the connectors as diagonals, improving the stiffness by a factor of about 3 as reported in Timmermann and Meierhofer [12].
designing sufficient connector-cover thickness, providing insulation to the timber-concrete-screws assembly and its integrity during fire, as in the case of analyzed TCC slab, the fire resistance could be determined numerically by means of a simpler model. By implementing material nonlinearity according to Eurocode standards, and assuming bonded connection between concrete and timber members, not modelling the connectors explicitly, the calculated fire resistance could be determined relatively accurate, by 10% overestimation in time.

### 3.2.3 Conclusion, recommendation and guidance

As a general purpose finite element software, ANSYS provides a wide range of modelling possibilities. Unlike specialized software, developed for a limited field of application, but optimized in terms of computational efficiency and accuracy, ANSYS could be used to simulate thermo-mechanical response of various structural systems of arbitrary geometry and material composition. The main drawback, however, is the lack of validation of such numerical models, for which a more extensive experimental results database is required.

For modelling the TCC slab exposed to fire, the following remarks should be made:

- Thermal and mechanical material properties should be implemented as temperature-dependent, based on the test results or recommendations provided in relevant standards, such as Eurocode, for advanced calculation method. ANSYS material library is extensive, providing material models which can be implemented by the user.
- Computational efficiency is governed by the mesh density and the number of calculation substeps performed during the analysis. Denser mesh should provide more accurate results, but at a cost of longer computational time, which, given the material nonlinearities and complex connection geometry (inclined steel screws embedded in concrete slab and timber beams/boards), can be demanding in real time.
- For a stiff type of connection, as provided in the example, not modelling the inclined connectors and assuming bonded contact between timber and concrete elements, can provide global structural response which is not more than 10% different compared to the experimental results, in terms of fire resistance time, but on the unsafe side. This is valid only if the connection integrity in fire is provided, by designing sufficient timber cover thickness, as in the analysed case.
- More realistic behaviour can be achieved by modelling connections and the contact zone between the concrete and timber parts of the structure, allowing sliding of the surfaces in contact with a possible separation. Introducing a nonlinear contact problem in a 3-dimensional analysis, as well as taking into account material nonlinearity and degradation at elevated temperatures, is far more computationally demanding, leading to uncertainties and requiring additional verification and validation steps during the process of model development.

### References

Numerical modelling of TCC structures in fire
Guidance document


3.3 Modelling of TCC in SAFIR

3.3.1 Introduction of the SAFIR software

SAFIR [1] is a special purpose computer program for the analysis of structures under ambient and elevated temperature conditions. The software is based on the Finite Element Method (FEM), can be used to analyse one, two or three-dimensional problems for structures in fire made of concrete, steel, RC, wood, composite etc.

As a finite element software, SAFIR accommodates various elements for different descriptions, calculation procedures and various material models for incorporating stress-strain behaviour. The elements include the 2-D SOLID elements, 3-D SOLID elements, BEAM elements, SHELL elements and TRUSS elements.

SAFIR can be used for performing three different types of calculations, namely, thermal, torsional and structural analysis. The analysis of a structure exposed to fire may consist of several steps. The first step involves predicting the temperature distribution inside the structural members, referred to as ‘thermal analysis’. The torsional analysis may be necessary for 3-D BEAM elements, a section subject to warping and where the warping function table and torsional stiffness of the cross-section are not available. The last part of the analysis, known as ‘structural analysis’, is carried out with the main purpose of determining the response of the structure due to static and thermal loading [1].
3.3.2 Numerical analysis of the reference TCC slab in Safir

Description of the model

In the thermal analysis, the cross-section of the TCC beam was analysed as a T-section with standard fire exposure (ISO 834) on the bottom side (Fig. 11 a). SOLID elements were used to model the cross-sections (Fig. 11 b). Figure 12 gives the temperature fields in the cross section at different times. Figure 13 compares the temperatures around the screw calculated with SAFIR and the temperature calculated with the expression (Eq. 1) developed by Frangi et al. [3], based on the results from the fire tests conducted on wood members exposed to ISO fire exposure from one side. The temperature-time curve, which describes the evolution of temperature at point A of the screw, was applied to the boundary of the cross-section of the steel verticals representing the connection in the structural analysis in SAFIR.

\[
T(x) = 180\left(\frac{\beta t}{x}\right)^\alpha; \quad \alpha(t) = 0.025t + 1.75; \quad x = 50 \text{ mm}
\]  

(1)

Figure 11. Cross section of the TCC beam a) ISO time-temperature curve applied to describe the environment around the model b) FE mesh plot.

Figure 12. Temperature fields in the cross-section at different times a) t=30 min b) t=60 min c) t=90 min.

In the structural analysis, three different models were considered. The structural system of the TCC slab in each considered model is presented in Table 2.

In order to compare the response of the real TCC slab with flexible shear connection to a theoretical case of a TCC slab with rigid connection, Model I was created.

In Model II and Model III 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 represent the connections. The vertical steel beam elements are placed at the exact position of the
screws through the length of the tested TCC beam. The only difference between Model II and Model III is the height of the steel verticals.

Figure 13. Comparison of temperatures around the screw calculated with SAFIR and with the expression for calculation of temperature profile in a wood member subjected to ISO fire exposure from one side given by Frangi et al. [3]

The main goal of presented numerical models was to properly model the connections. According to Frangi [2], at T=20 °C one pair of the 45° inclined screws has a characteristic load-carrying capacity of $F_{v,rk}=15.6$ kN. According to ETA-13/0699 Error! Reference source not found., these screws have the following properties: $F_{v,rk}=18.38$ kN and $K_{ser}=17.21$ kN/mm. In the numerical models, the steel verticals are modelled as steel beam elements with the same load carrying capacity as the 45° inclined screws in [2].

Table 2. Structural system and connection characteristics in each considered model in the structural analysis.

<table>
<thead>
<tr>
<th>Numerical model</th>
<th>Connection characteristics</th>
<th>Structural system</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model I</td>
<td>Rigid connection (no slip between the subcomponents occurs)</td>
<td>![Model I Diagram]</td>
</tr>
<tr>
<td>Model II</td>
<td>Flexible connection (slip between the subcomponents occurs)</td>
<td>$h_{verticals} = 180$ mm</td>
</tr>
<tr>
<td>Model III</td>
<td>Flexible connection (slip between the subcomponents occurs)</td>
<td>$h_{verticals} = 70.71$ mm</td>
</tr>
</tbody>
</table>
BEAM element were used for modelling the analysed TCC system. In each time step during the structural analysis, 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 [1].

**Material models**

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

For concrete - SILCON ETC [5] material model is used. This material model is based on the EN 1992-1-2 [7] model except that 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 [6], 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 [7].

For describing the temperature-dependent mechanical behaviour of timber the uniaxial material model of Annex B of EN 1995-1-2 [8] is used. The strength and stiffness start to decrease as soon as the temperature exceeds 20°C and 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.

The input data describing the mechanical and thermal characteristics of the materials used in the numerical analysis are presented in Table 3 and Table 4.

**Table 3. 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/mm$^2$] 37000</td>
<td>37600 (Silicon ETC)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$f_c$ [N/mm$^2$] 47</td>
<td>47</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$f_t$ [N/mm$^2$] /</td>
<td>4.7</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\rho$ [kg/m$^3$] /</td>
<td>2400</td>
<td></td>
</tr>
<tr>
<td>GL24h</td>
<td>$E$ [N/mm$^2$] 10620</td>
<td>same as in experiment</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$f_c$ [N/mm$^2$] 24</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$f_t$ [N/mm$^2$] 16.5</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$\rho$ [kg/m$^3$] 431-445 (438)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Numerical modelling of TCC structures in fire
Guidance document

### Table 4. Thermal characteristics of materials

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

**Results**

This newly developed method in SAFIR for analysis of a TCC slab in fire was able to calculate the slip between the subcomponents of the composite (concrete and timber). The main results from the analysis are presented in Table 5. Comparison of deflection curves and slip curves from the numerical models in SAFIR, the experiment and the simplified calculation method developed by Frangi et al. [3] are presented in Fig. 14 and Fig. 15, respectively.

As expected, Model I gave lower deflection in comparison to the fire test.

The time-deflection curve calculated with Model II matches the results from the fire test up to 50 minutes, but afterwards the deflection is slightly underestimated. In terms of the slip this model overestimates the relative slip between the concrete slab and the timber beam.

Model III gave best results in term of both deflection and slip. The results in this model are highly influenced by the diameter of the steel vertical. Model III with diameter of the steel verticals of 16

<table>
<thead>
<tr>
<th>Material</th>
<th>Thermal characteristics</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Reinforcement steel</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\omega$ [%]</td>
<td>13</td>
<td></td>
</tr>
<tr>
<td>$E$ [N/mm$^2$]</td>
<td>/</td>
<td>200000</td>
</tr>
<tr>
<td>$f_y$ [N/mm$^2$]</td>
<td>/</td>
<td>500</td>
</tr>
<tr>
<td>$d$ [mm]</td>
<td>6</td>
<td>6 /100/100mm</td>
</tr>
<tr>
<td><strong>Connection</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$E$ [N/mm$^2$]</td>
<td>/</td>
<td>210000</td>
</tr>
<tr>
<td>$f_y$ [N/mm$^2$]</td>
<td>/</td>
<td>400</td>
</tr>
<tr>
<td>$d_{\text{inner}}$ [mm]</td>
<td>4.3</td>
<td></td>
</tr>
<tr>
<td>$d_{\text{outer}}$ [mm]</td>
<td>7.3</td>
<td></td>
</tr>
</tbody>
</table>
mm gave time-deflection curve which is somewhere between the curves from the fire test and the simplified model up to 45 minutes, but afterwards this curve matches the one from the fire test. The slip values in this model are highly overestimated. Increasing the diameter in model III to 16.5 mm and 17 mm leads to slightly decreased deflections and significantly decreased slips. In terms of slip, Model III with 17 mm gave results that are in very good agreement with the results from the fire test. Here, the apparent “peak” in the time-slip curve that occurs at $t = 30$ minutes (zero slip) is due to the slight change of rotation of the outer steel verticals during the deformation process of the structure.

![Figure 14. Comparison between the calculated deflection in Safir, the measured deflection in the experiment and the calculated deflection with the simplified method proposed in Frangi et al.[3].](image)

![Figure 15. Comparison between the calculated relative slip in Safir, the measured slip in the experiment and the calculated slip with the simplified method proposed in Frangi et al.[3].](image)
Table 5. Results from the structural analysis: fire resistance, deflection and slip.

<table>
<thead>
<tr>
<th></th>
<th>$t_{fr}$</th>
<th>$t_{fr, model} / t_{fr, test}$</th>
<th>Deflection at failure $\Delta y$ [mm]</th>
<th>$\Delta y_{model} / \Delta y_{test}$</th>
<th>Slip at failure $\Delta x$ [mm]</th>
<th>$\Delta x_{model} / \Delta x_{test}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fire test</td>
<td>67 /</td>
<td>≈39.4</td>
<td>/</td>
<td>≈1.04</td>
<td>/</td>
<td>/</td>
</tr>
<tr>
<td>Simplified model</td>
<td>65 0.97</td>
<td>≈33</td>
<td>0.84</td>
<td>≈0.8</td>
<td>0.77</td>
<td></td>
</tr>
<tr>
<td>Model I (rigid connection)</td>
<td>71.8 1.06</td>
<td>22.5</td>
<td>0.57</td>
<td>/</td>
<td>/</td>
<td>/</td>
</tr>
<tr>
<td>Model II $L_{vertical}=180$ [mm]</td>
<td>72 1.07</td>
<td>38.4</td>
<td>0.97</td>
<td>2.02</td>
<td>1.94</td>
<td></td>
</tr>
<tr>
<td>$d=25$ [mm]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Model III $L_{vertical}=70.71$ [mm]</td>
<td>68 1.01</td>
<td>38.9</td>
<td>0.99</td>
<td>1.57</td>
<td>1.51</td>
<td></td>
</tr>
<tr>
<td>$d=16$ [mm]</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$d=16.5$ mm</td>
<td>68.8 1.03</td>
<td>35.5</td>
<td>0.90</td>
<td>1.18</td>
<td>1.13</td>
<td></td>
</tr>
<tr>
<td>$d=17$ mm</td>
<td>69.5 1.04</td>
<td>32.7</td>
<td>0.83</td>
<td>0.84</td>
<td>0.81</td>
<td></td>
</tr>
</tbody>
</table>

3.3.3 Conclusion, recommendation and guidance

The numerical model successfully predicted the general behaviour of the structure in fire. The calculated fire resistance, deflection and slip between the subcomponents of the TCC structure from the numerical model were in very good agreement with the experimental results.

In the modelling process user should pay attention to the following factors that have highest influence on the response of the TCC structure:

- The mesh size in the thermal analysis:
  - Proper sized finite elements are necessary to avoid skin effect at the surfaces and to correctly calculate the area of all elements in the cross-section (timber beam, concrete slab and reinforcement).
- The diameter of the steel verticals:
  - Different structural systems needed different diameter of the steel verticals to simulate as close as possible the global structural response (deflection and slip) of the tested slab. Change of only 0.5 mm in the diameter of the steel verticals in one structural system made difference in the results.
- The structural system in the structural analysis (see Table 6).
Table 6. Variations in the structural system of Model III.

<table>
<thead>
<tr>
<th>Description</th>
<th>Structural system</th>
</tr>
</thead>
<tbody>
<tr>
<td>a) Virendeel truss without steel verticals at the end</td>
<td></td>
</tr>
<tr>
<td>b) Virendeel truss with steel verticals at the end</td>
<td></td>
</tr>
<tr>
<td>c) Virendeel truss with steel verticals at the end and horizontal constraint at the left end</td>
<td></td>
</tr>
</tbody>
</table>

References


3.4 Modelling of TCC in COMP-WOOD

3.4.1 Introduction of the COMP-WOOD software

Fire resistance and behaviour of TCC beam (hereinafter beam) is determined based on the advanced calculation method, where own developed software is used. The entire analysis is divided in two separated phases, assuming that the temperature of gases (fire curve) in fire is known:

- 1st phase: Computation of temperature field over TCC beam cross-section
- 2nd phase: Calculation of the mechanical behaviour of the beam exposed to fire.

Below the two mentioned steps are presented to some detail.
**Thermal analysis**

The first step, i.e. heat transfer analysis, comprises the determination of temperature fields in composite member subjected to a given fire temperature time regime. Heat transfer is modelled based on the well-known Fourier partial differential equation which describes heat transfer through solid body. Heat exchange at the contact between beam cross-section and the surroundings is prescribed in terms of specific surface heat flux consisting of convective and radiative part. The system of basic equations is solved numerically by the self-developed finite element software Heatko [5], which operates in Matlab environment. Important input parameters of the thermal analysis are the thermal properties of the materials. The temperature dependence of the specific heat, thermal conductivity and density of concrete and timber are taken from the Eurocode standards EN 1992-1-2 [2] and EN 1995-1-2 [3] respectively. Following the recommendations of EN 1992-1-2 (2004) the lower limit of the thermal conductivity of concrete gives more realistic temperatures than the upper limit. With the increase of the specific heat of timber in the temperature range between 100 – 120°C the indirect impact of evaporation of water on the development of the temperature in timber is modelled. In addition, thermal conductivity suggested by the EN 1995-1-2 [3] takes increased heat transfer due to shrinkage cracks at temperatures above 500°C into consideration.

When determining the temperature field of the beam, the assumption that the temperature of the entire fire area, or at least part of the area along the beam, is uniform, which significantly simplifies thermal analysis, won’t lead to a significant mistake. In this case, only the temperature field in the characteristic cross-section of the beam need to be determined. In the case of a normal rate of reinforcement, according to EN 1992-1-2 [2], influence of the reinforcement on the time and space distribution of the temperature is minor and the temperature of the reinforcement bars can be determined based on the temperature of the concrete at the position of the reinforcing bars.

**Mechanical analysis**

For the mechanical analysis, software COMP-WOOD is used. This program uses a strain based finite element formulation to determine the mechanical response of the planar beam subjected to time-varying mechanical and temperature loadings [10]. The formulation is based on the kinematically exact planar beam theory of Reissner [11]. The remaining unknown functions, (the displacements, rotations and internal forces and moments) appear in the functional only through their boundary values. The finite element formulation yields a system of discrete generalised equilibrium equations of the structure, which are solved by the Newton incremental iterative method. For a full description of the mechanical model reader is referred to [4],[6]. In the model an iterative method is used, and the whole time domain is divided into time increments $\Delta t = t_i - t_{i-1}$. Based on the given stress and strain state at the time $t_{i-1}$ and temperature $T$ at $t_i$, the geometrical strains $D$ of any point of the beam at time $t$ can be determined. Considering the principle of additivity of strains and the material models of concrete and timber at elevated temperatures, the strain increment, $\Delta D_{ui}$ consists of the sum of the individual strain increments due to temperature $\Delta D_{th}^{ij}$ and stress $\Delta D_{st}^{ij}$. Details can be found in [4],[6]. The temperature strain increment is calculated from EN 1992-1-2 [2] for concrete. Linear relationship for temperature
strain increment of timber is considered with the coefficient of thermal expansion given as $\alpha_t = 5 \times 10^{-6} \text{ K}^{-1}$.

The cross section of a composite beam is divided into subsections. Subsections are then divided into small segments, as shown in Fig. 16. Each subsection can then have its own material, thermal and mechanical properties. The integral over the cross-section is written as a sum of integrals over the individual segments. The integral over each individual segment is evaluated by the Gaussian 3x3 point rule.

![Figure 16. The mesh of the Gaussian integration nodes over the cross-sections of the composite beam element](image)

**Material models**

The stress-strain relationship and reduction factor for the mechanical properties of concrete at the elevated temperatures are taken from EN 1992-1-2 [2] (Figs. 17 a and b).

(a) EC2 constitutive model of concrete:

![Figure 17. (a) Stress-strain relationships of concrete at elevated temperature. (b) temperature dependent reduction factors for siliceous concrete according to [2].](image)

Bi-linear stress-strain relationship is considered for timber (given in [8],[9]; see Fig. 18).
In Fig. 18, \( D_{ui} \), \( E_{ui} \) and \( f_{ij} \) (\( i = c, t; j = T, T_0 \)) are the limit elastic strains, the Young’s modulus and the strengths of timber in compression (c) and tension (t) at room \((T_0)\) and elevated temperatures \((T)\), respectively. Similarly, \( E_{ipj} \) and \( f_{ipj} \) (\( i = c, t; j = T, T_0 \)) denote the plastic hardening modulus and limit plastic stress, respectively. Temperature dependency of all described parameters is determined by the following rule:

\[
X_T = k_T X_{T_0},
\]

where \( X_T \) represents the parameter value at elevated temperature, \( X_{T_0} \) is the value at room temperature and \( k_T \) is temperature dependent reduction factor. The reduction factors for strength and stiffness of timber are considered according to EN 1995-1-2 [3] and presented in Fig. 19.

Connection between timber beam and concrete slab is usually flexible, where longitudinal as well as transversal slips between the layers during the deformation occurs. It is a common practice in
engineering design that the layers are rather rigidly connected in the transverse direction, which lead us to assume that the transverse separation can be neglected.

The constitutive law of the contact is usually determined with the standard pull-out shear test. Results of such a test present the relationship between the resultant shear force $P_{\text{pull}}$ and the interlayer slip in the direction of the ‘pull-out’ force. Furthermore, with rising temperature the stiffness of the contact connection decreases and this has to be considered in the constitutive law of the contact. In the analysis, linear or nonlinear relationship between the contact tangent traction $p_t$ vs. slip $\Delta$ on the timber-concrete contact can be accounted for. Linear contact law is of the form:

$$p_{t,T} = K_T \Delta,$$

where $K_T$ is temperature dependent slip modulus determined as $K_T = k_{E,T} K_{20}$, further on, $K_{20}$ is the slip modulus at room temperature and $k_{E,T}$ is temperature dependent reduction factor of young modulus (EN 1995-1-2 [3]). Nonlinear contact law is adopted from Huang et al. [7] and given as:

$$p_{t,T} = A \cdot k_{f,T} \cdot p_{t,\text{max}} \left(1 - e^{-B k_{E,T} \Delta}\right),$$

where $A$ and $B$ are empirical coefficients, $k_{f,T}$ is temperature dependent reduction factor of timber strength (EN 1995-1-2 [3]) and $p_{t,\text{max}}$ is the bearing capacity of screws per unit length.

**Interpretation of results**

The software COMP-WOOD enables to display results important to understand the behaviour of beam in fire such as: the development of vertical and horizontal displacement, the development of slip at the contact between timber and concrete, stress distribution over beam cross-section, the development of different strain increments and many others. In addition, important feature is that failure time can be estimated. In the software the failure of the structure (fire resistance) is determined when the tangent stiffness matrix of the structure (beam) becomes singular during the iterative procedure. This can occur due to the global instability of the structure or due to material failure. The latter occurs when the determinant of the tangent constitutive matrix of the beam cross-section becomes zero, which indicates that the ultimate bearing capacity of the cross-section has been reached [1].

3.4.2 Numerical analysis of the reference TCC slab in COMP-WOOD

For the thermal analysis, the cross section of reference case give in Fig. 1 is discretized into 2192 quadrilateral finite elements. Thermal properties of glulam timber and concrete are taken according to EN 1995-1-2 [3] and EN 1992-1-2 [2], respectively. The initial density of timber is $\rho_0 = 500$ kg/m$^3$.

Temperatures at the bottom of the screw and at the screw position at the contact between timber and concrete reach 200°C and 60°C, respectively. This means that the load-carrying capacity of the connection is reduced according to the temperature dependent reduction of strength and stiffness of timber. In the concrete part of the beam, just above the timber formwork system, small temperature increase is observed, since the maximum temperature remains under 200°C.
Consequently, almost entire load bearing capacity of concrete is retained, concrete strength is reduced by only 3% at this temperature according to EN 1992-1-2.

In the mechanical analysis, the beam-type slab is modelled as simply supported beam discretized on 10 one dimensional finite elements. The cross section is divided on 80 rectangular fields for timber part of the section and 104 fields for concrete part. Inside each field, 3x3 Gaussian integrations scheme is used to determine normal stresses over the cross section. According to Frangi et al. [4], strength classes of used materials are: timber 24f, concrete C30/37, net reinforcing steel B500. Bi-linear constitutive relationship for wood is used [8], while for concrete and reinforcing steel, stress-strain relationships given in EN 1992-1-2 [2] are adopted. Linear connection model is used in the study with the slip modulus given as \( K_T = 40 \text{ N/mm} \).

Fig. 20a represent the development of midspan displacement. Almost perfect agreement between measured and calculated midspan displacement is observed. The response in the model is mainly governed by the reduction of mechanical properties of timber part of the beam due to the temperature increase. Good agreement is also obtained by comparing experimentally and numerically determined failure times. In the model material failure was identified, which occur when the tangent constitutive matrix of the beam cross-section becomes zero. This indicates that the load bearing capacity of the cross-section has been reached, namely, the load bearing capacity of timber part of the beam. Fig. 20b demonstrates slip development with time determined at the beginning of the slab (at the supports). In general, measured and calculated slip development agree well, although some small discrepancies are observed. A possible reason for this could be, that measured and calculated temperature at the screw location is not the same or that different temperature dependent reduction factors for the load-carrying capacity of the connection should be considered.

3.4.3 Conclusion, recommendation and guidance

Own developed software COMP-WOOD is an advanced computation method for the analysis of TCC beam exposed to fire. Some modelling guidelines when using COMP-WOOD as well as other own developed software are given in the subsequent:
• The software has to be based on a detailed physical description of the problem, where all the main phenomena influencing the behaviour of TCC element exposed to fire, has to be considered.
• The influence of reinforcing steel and connectors on temperature distribution over the TCC system can be neglected.
• Thermal properties of timber and concrete can be taken according to standards EN 1995-1-2 [3] and EN 1992-1-2 [2], respectively.
• Material models for concrete and reinforcing steel can be considered according EN 1992-1-2 (2004).
• Bi-linear material model for timber in tension and compression is recommended to use ([8],[9]).
• Slip at the contact between timber and concrete part of the TCC system has to be modelled. Different slip models (linear or non-linear) can be used, depending on the type of connection.

References
3.5 Modelling of TCC in ABAQUS

3.5.1 Introduction of the ABAQUS software

ABAQUS is a general-purpose finite element computer software, which can be used to simulate various engineering problems [1]. Here preliminary overview for the key steps to assess in the finite element numerical modelling of fire exposed TCC assemblies with steel screwed connections (for example shown in Fig. 21), giving evidence of major issues and influencing parameters.

![Figure 21. Example of TCC assembly, according to [2].](image)

The fire resistance of Timber-Concrete Composite elements is mainly influenced by timber members and by the connectors [2],[3]. Major effects of the assigned fire on the timber components, such as heating and reducing overall section size act to weaken both the timber section and the connection between timber and concrete. The geometrical and mechanical features of connections in TCC assemblies, in this regard, have also a key role, as the integrity of the connection during the fire is governed by its weakest element, which can sometimes be difficult to predict.

As a general numerical approach for ABAQUS, thermal and structural analyses on a given TCC system can be carried out as uncoupled simulations, where the effects of thermal and mechanical loads are properly combined. As an alternative, fully coupled thermo-mechanical analyses can be also set in ABAQUS, so to perform a single run. Major issues, in this latter case, are however associated to lack of convergence and numerical issues.

In any case, both the numerical methods can only roughly account for the complex phenomena that are typically involved in the fire performance of timber and TCC structural systems [4],[5]. In this regard, special care should be spent to assess the reliability of all the FE modelling assumptions, including material properties, interactions between the TCC components, etc.

According to the uncoupled procedure, the FE thermal and mechanical models are conventionally assembled and calibrated independently. The reference numerical analysis for a TCC system in fire conditions hence consists of two uncoupled sub-steps:
• thermal analysis and
• mechanical analysis.

The first sub-step deals with the determination of the temperature distribution throughout the cross section of the TCC components, while the second one deals with the estimation of thermo-mechanical effects due to additional (if present) mechanical loads.

**Thermal analysis**

The reference thermal analysis must be carried out in the form of a heat transfer step, so as to describe the thermal state of the examined TCC assembly when exposed to the assigned fire scenario (i.e. ISO fire curve or other input data).

3D solid elements (heat transfer, DC3D8 type from ABAQUS library) must be used for thermal assessment purposes, spending careful consideration for mesh (size and pattern), so as to ensure the accuracy of thermal estimations (especially with respect to local thermal effects) but preserving the computational cost of modelling and simulation phases.

A set of boundary conditions for radiation and convection need then to be defined, according to the fire exposure of the surfaces of interest for the TCC assembly, as well as on the thermo-physical properties of the involved materials and on the ambient conditions. Emissivity and convection coefficients for steel, timber and concrete can be derived from design standards Eurocode provisions ([6],[7],[8]).

**Mechanical analysis**

The thermal, heat transfer of the given TCC system is then followed by an uncoupled, nonlinear mechanical simulation (static general step) carried out on a FE model able to capture - for the TCC assembly - its stress-strain response as a function of the assigned mechanical loads (to define) and temperature field (to derive from the thermal sub-step “A”).

Compared to the sub-step “A”, key variations in the FE model include the element type (C3D8R type, linear brick elements with reduced integration solid elements, in place of DC3D8 heat transfer elements), as well as boundaries and surface-to-surface contact interactions. This way the desired mechanical performance of the TCC system is enabled, as well as to account for possible reciprocal displacements and rotations of the TCC components (surface-to-surface mechanical interactions, allowing for the progressive separation of the TCC components in tension, etc).

**Key material properties**

To perform the transient thermal analysis, the theory behind the heat transfer analysis shows that only three material properties are needed, namely represented by conductivity, density and specific heat, including their variation with temperature.

In case of timber-concrete-composite (TCC) slabs, thermal properties for timber, concrete and steel can be adopted according to relevant Eurocode standards.

For the structural sub-step, key input mechanical properties are represented by material density, coefficient of thermal expansion, modulus of elasticity and Poisson’s ratio. Spruce anisotropy
should be accounted via appropriate engineering constants, representative of the longitudinal and shear moduli along the principal directions of the timber resisting members. Careful consideration should be spent for the material orientation. Several sub-models and constitutive laws can then be accounted to simulate the progressive propagation and evolution of damage in single TCC components, including ductile phenomena, concrete brittle cracking, etc.

**Model definition and assembly**

The typical FE model in ABAQUS consists of several parts and instances that should be able to reproduce the actual geometrical and thermo-mechanical features of each TCC component. Given the separate FE model components, however, a key role is assigned to their mechanical interaction, being part of an assembled system. In this case, the assembly and interaction modules allow the user the possibility to merge and assemble the TCC system under investigation, including a wide possibility of surface-to-surface mechanical interactions (see for example [9]) and / or cohesive interactions that could be particularly suitable when specific failure mechanisms should be accounted. Recent applications for timber-concrete push-out specimens and timber-timber composite beams with inclined self-tapping screws can be found (for cold conditions) in [10],[11].

Full 3D solid assemblies should be generally preferred, to ensure the accuracy of thermo-mechanical predictions, especially when advanced damaged models and interactions are used to investigate the post-damage performance of the same systems. In this regard, special care is required for the definition of the mesh size an pattern, to preserve the computational efficiency of FE models.

**Loads and boundaries**

The load and BC modules are intended - for both the thermal and mechanical sub-steps - for the definition of input loads and boundary conditions. Careful consideration should be paid - in the thermal step - for the temperature exposure of the TCC parts. Similar effects to account for thermal conditions can be implemented, as alternative, in the form of initial, predefined fields.

In the mechanical sub-step, the structural self-weight can be calculated automatically by the software (gravity load), based on the material density provided in the material module. Additional external loading can be defined acting on single nodes, or faces of the elements (pressure).

**Job & Results / Visualization**

The visualization of results an their post-processing is strictly related to the type of analysis being performed. In case of thermal-structural analyses on TCC systems under fire, the temperatures at each mesh node should be saved as a major output of the first sub-step, so to be used - in the following mechanical simulation - as a key nodal input for the nonlinear static estimations.
3.6 ABAQUS

3.6.1 Numerical analysis of the tensile test in ABAQUS

Here short summary of the results tensile tests shown in Fig. 22 and experimentally investigated in [12] are presented.

The mean value of the load-carrying capacity of the axially loaded screws experimentally measured in [12] at normal temperature was $F_{\text{max}} = 12.4 \pm 0.6$ kN, and most of the samples failed due to withdrawal of the threaded part of the screw. From a numerical point of view, this means that major issues are related to thermal effects in the samples components, but also to damage propagation due to the withdrawal of screws. During the fire tests the screws were axially loaded with a constant tensile force of 3.6 kN (about 30% of their load-carrying capacity at normal temperature). In experiment two series of tests were performed. Here results from the test “type II”, where specimens were exposed to and EN/ISO fire, until failure of the screws, are presented.

Numerical model

For the assembled FE models discussed herein, the sensitivity of basic materials to temperature variations were properly taken into account. A key role was assigned to the numerical description of the connection detailing, including a refined FE description of the involved components but also a set of surface-to-surface contact interactions and cohesive damage laws, so to allow a reliable estimation of the actual interaction between the involved components (i.e. relative slip and possible separation of the sample components). Due to symmetry, $1/4$th the nominal geometry of Figure 1(a) was numerically reproduced, with appropriate restraints and boundaries (Fig. 23).

The load-bearing performance in fire conditions was numerically investigated in the form of uncoupled, thermo-mechanical analyses. Accordingly, two separate FE models were described in ABAQUS/Standard [1]. DC3D8 and C3D8R type elements were used respectively, to describe the 3D solid components of each model, to perform the thermal and mechanical steps.

For the thermal sub-step, specific thermo-physical input properties were assigned to the timber surfaces exposed to the EN/ISO fire curve. An ideal, uniform fire exposure accounting for the experimental insulation layers of Figure 1(a) was numerically reproduced. Emissivity and convection coefficients for timber were set to 0.8 and 25 W/m²K. Otherwise, ambient temperature was accounted on the unexposed timber side. The variation of timber mechanical properties with temperature was finally considered, in accordance with the Eurocode provisions for standard fire exposures [6], whereby the modulus of elasticity and strength reduce to zero at 300 °C.
Numerical modelling of TCC structures in fire
Guidance document

Figure 23. Reference FE model for the thermo-mechanical analysis of tensile specimens in fire conditions (ABAQUS).

Based on [11], major simplifications in the joints detailing were made to the geometrical description of steel screws and their mechanical interaction with the surrounding timber parts (see the detail view in Figure 2). In particular, see also [11], each self-tapping screw was reproduced in the form of:

i. an equivalent, 100 mm long, circular cross-section (with uniform diameter) equal to the net thread size of screws;

ii. an upper circular cross-section, 50 mm the nominal length, with 6 mm its diameter, and

iii. a ‘soft layer’, being representative of screws threads and timber fibres (6 mm the outer diameter), being interposed between each screw and the surrounding timber members. According to [11], such a fictitious soft layer and the related cohesive contact/damage interaction - given the typically high withdrawal strength of self-drilling screws - aims to account for possible brittle failure mechanisms at the screw-to-timber interface, specifically for possible damage occurring for shear or tension perpendicular to the grain.

Connection interface

Given the reference FE assembly displayed in Fig. 21, a tangential ‘penalty’ and normal ‘hard’ surface-to-surface behavior was first defined for the timber-to-timber surfaces in contact ($\mu = 0.5$ the static friction coefficient). The steel screw and the surrounding ‘soft layer’ were then rigidly connected via a ‘tie’ mechanical constraint, hence enabling relative rotations and displacements among the interested nodes.

The external surface of the ‘soft layer’ and the adjacent timber elements were then interconnected via a ‘cohesive contact’ interaction. The Cohesive Zone Modelling technique is largely used for several structural typologies (i.e. [13],[14], etc.), but rather limited applications are still available for timber composites (i.e. [15],[16]), and no literature efforts can be found for timber composite systems in fire.

As a result, special care was taken to model both the elastic stiffness and the damage input data, see [11]. For the radial, longitudinal and normal stiffnesses (i.e., being representative of the interface stiffnesses prior to damage onset), the ‘default contact enforcement method’ was used. In terms of damage at the ‘soft layer’-to-timber interface, being expected to initiate together with
the failure propagation in timber, the maximum nominal stress (MAXS) criterion was used. The ‘damage initiation criterion’ was detected as a combination of stresses, i.e.

\[
\max \left( \frac{f_n}{f_{n0}}, \frac{f_s}{f_{s0}}, \frac{f_t}{f_{t0}} \right) = 1,
\]

where the variables \(f_n, f_s\) and \(f_t\) present the maximum allowable values of nominal stresses when the deformation is purely normal (n) to the bonding interface or in the first (s) or second (t) shear directions. Following [11] and presented criteria, the reference stress values for damage initiation were defined in this study by accounting for the mean timber mechanical properties, as separately detected for the normal, tangential and radial directions. Finally, a linear damage evolution law was set for the degradation of mechanical contact properties, with full residual stiffness for the cohesive contact interactions at the first attainment of 4 mm deformation. The so defined cohesive damage interaction was combined with a tangential penalty / normal hard behaviour, as previously described. In this manner, once attained the failure condition for the cohesive contact region, a reliable performance of the FE samples was ensured, avoiding the interpenetration of screws in the adjacent timber components.

**Materials**

Both timber and steel constitutive laws were described in accordance with product standards, including variations with temperature (for timber) and appropriate damage models. Spruce was defined as an orthotropic media with brittle elastic behaviour. As also in accordance with [11], given the grain direction and loading condition (Fig. 21), the Hill plastic criterion was used to specify appropriate resistance values along the principal directions of interest for the timber components. In the case of the equivalent ‘soft layer’, being characterised by an indefinitely linear behaviour, the elastic mechanical properties of timber were again considered. The variation was represented by the radial MOE, where a fictitious value of 50MPa was taken into account (see Table 2 and [11]).

In this regard, it is important to point out that the given input features should be properly calibrated, as far as different structural systems and/or wooden resistance classes are considered, compared to the selected samples.

**Table 7. Variations in the structural system of Model III.**

| ‘Soft layer’ moduli (mean values, in MPa) | Longitudinal (i.e., cylinder axis) and tangential | |  |  |  |
| | Shear | Radial | Max. shear (MPa) | Damage evolution / displacement (mm) | Linear / 4 |
| ‘Soft layer’ failure | | | Longitudinal | Transverse | Shear | 37.55 | 3.85 | 3.85 | 3.5 |
| Cohesive damage contact resistance (mean values, in MPa) | | | Longitudinal | Transverse | Shear | Rolling shear |

FE results presented in next paragraph are presented for the tensile sample with 80x120 mm timber beam cross-section, 40/40 mm the side/bottom cover of the steel screws. In the experimental investigation this case is noted as “SBZ 4.1” [12].

**Results**

Figure 24 illustrates the experimentally measured deformation, as a function of time (the beginning of the fire tests coincides with time 0). The screws with 60 mm side cover (SBZ 4.2 type), for
example, experimentally failed after about 78 minutes (with 0.8 mm/min the measured charring rate). Accordingly, failure of screws with 40-mm side cover (like the selected SBZ 4.1 samples) occurred in the experiments after about 54 minutes (i.e., 24 minutes of fire resistance scatter, when reducing the covering depth to 20 mm). A mostly stable behaviour, in addition, was observed from the two test repetitions. The corresponding FE model, see Fig. 24b), proved to offer a rather good correlation with the past experimental estimations. Major issues were represented by weak convergence of the analyses, in the final stage of the simulations, due to severe damage in the structural components (i.e. material side) as well as at the cohesive interfaces.

![Figure 24. Tensile specimen results under "type II" tests, (a) experimental [12] and (b) numerical time-deformation history output (ABAQUS).](image)


In terms of temperature and damage evolution in the sample components, the FE analysis highlighted an abrupt propagation of the charred timber section and cohesive damage for the screw in tension, after about $\approx 30$-35 minutes of fire testing. Such a finding, in close correlation with time-displacement measurements, is displayed in Fig. 25 in the form of selected contour plots for the screw and the timber beam (‘CSMAXCRT’ and temperature contour plots, respectively). As shown in Figure 25b), in particular, after $\approx 45$ minutes of fire exposure the residual section of timber is mostly vanished. In the following time instants - up to the experimental failure time of 54 minutes - both fire and cohesive damage effects further propagate, leading to the final withdrawal of the screw.

![Figure 25. Selected contour plots for the screw and the timber beam.](image)
Numerical modelling of TCC structures in fire
Guidance document

3.6.2 Conclusion, recommendation and guidance

ABAQUS software package can offer a wide range of possibilities for the numerical simulation of the thermo-mechanical response of structural components and assemblies exposed to fire loading. The modelling approach presented here provides some preliminary, general guidelines for modelling TCC slabs subjected to fire. During the modelling user should pay special attention to:

- account for the thermo-mechanical boundaries and loads, in order to properly estimate the expected load-bearing performance of a give TCC system.
- A key role of the presented FE models is represented by the use of cohesive contacts and damage laws, being able to reproduce possible withdrawal mechanisms for the selected samples.

As shown the presented FE modelling approach proved to offer rather good correlation with past test results, hence confirming - in accordance with past numerical efforts of literature - to represent a robust tool for fire performance assessments.

References

[1] Simulia, ABAQUS v.6.14 computer software, Dassault Systèmes, Providence, RI, US


4 Conclusions

The presented guideline gives recommendations and tips in numerical modelling of TCC systems with the commercial software: ANSYS, SAFIR, ABAQUS and self-developed software COMP-WOOD. Analysis showed that all the software’s are quite a robust tool for fire performance assessments of TCC systems. Study of the reference TCC beam revealed that all software’s, without ABAQUS, were able to predict the time-displacement response and related fire resistance quite well. More detailed results such as slip-force curve is somehow difficult to capture in ANSYS, and SAFIR while more simple yet special tool COMP-WOOD described this relation relatively good in comparison to experimental results. In ABAQUS a comprehensive study of tensile tests was performed in order to present more advanced and detailed modelling of TCC components in fire. Numerical modelling is more demanding in this case compared to the reference study of TCC beam since the results are more sensitive to material parameters, meshing, solving algorithm, etc. and thus users are advised to perform several convergence and test studies before application use. In general, following can be concluded:

- All software’s provide robust solvers which need to be optimised, in terms of computational efficiency, depending on the type of problem analysed.
- In majority results will depend on the input and thus user should take special attention when performing this task.
- Regardless of the computer hardware developments in the past decades, the nature of the physical problem demands large quantities of computational recourses, still preventing numerical simulations of such systems from entering the everyday engineering practice.
Guidance on Numerical Modelling of Timber Concrete Composite Structures in Fire

COST Action FP 1404
“Fire Safe Use of Bio-Based Building Products”

Document N224-07

Chair of COST Action FP1404: Joachim Schmid
Vice Chair of the Action: Massimo Fragiacomo