Finite Element Modelling of Post-Tensioned Timber Beams under Fire Conditions

Hailey Quiquero, MASc Candidate, Carleton University, Ottawa, Canada
John Gales, Assistant Professor, York University, Toronto, Canada
Anthony Abu, Senior Lecturer, University of Canterbury, Christchurch, NZ
Peter Moss, Adjunct Associate Professor, University of Canterbury, Christchurch, NZ

ABSTRACT

In order to implement innovative timber structures globally, performance-based design using numerical models are often required. A model was developed to simulate post-tensioned (PT) timber beams in FEM software ABAQUS and compared to three furnace beam experiments. A sequentially coupled heat transfer – mechanical analysis was developed and validated. The timber was treated as orthotropic both in elasticity and plasticity, and was modelled with temperature-dependent properties. The simulation results showed good agreement with both the thermal results and the mechanical analysis. The thermal results matched the thermocouple readings well and the char depths were predicted to within 4 mm. The failure modes and times were accurately predicted (within 5% accuracy), and the deflection behaviour was reasonably predicted compared to the experimental data.

INTRODUCTION AND BACKGROUND

Along with the innovation of new building systems, the ways in which we design them must also advance in order for the systems to be implemented by practitioners. This is especially true for the fire engineering design of novel construction methods as their performance under fire is investigated. In order to complete a performance-based design of such systems, a computer model is often required to demonstrate system behaviour. Predictive models are often valuable in the design and optimization of such complex structures providing they can be validated against meaningful data.

Post-tensioned (PT) timber is a relatively nascent building system in which the structure is composed of engineered timber elements which are pre-stressed with high-strength steel tendons. This study has focused on PT timber beam elements. The engineered timber may take the form of Laminated Veneer Lumber (LVL) or Glulam, for example, as a structural element which has a central cavity running along its length. The steel tendons are typically placed eccentrically below the neutral axis of the beams, in order to induce a negative moment and overall compression in the beam as it is pre-stressed. Analogous to the widely-used pre-stressed concrete system, such an approach allows for longer span beams using smaller cross-sections. A typical PT timber beam schematic is shown in Figure 1.

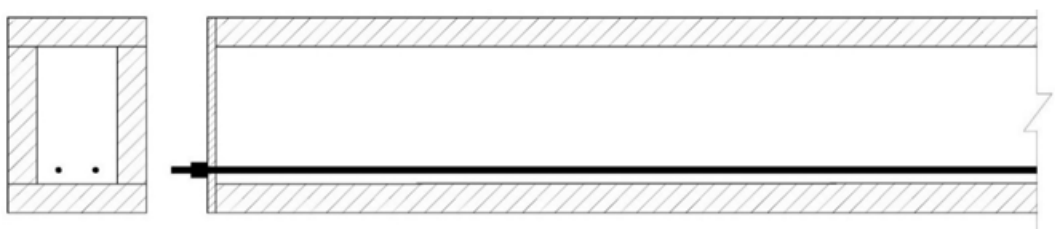

Figure 1: Schematic of typical PT timber beam cross-section showing built up wood webs and flanges and high-strength steel tendons, along with section cut through beam length showing tendon anchorage with a steel plate at the end

The PT timber system was developed as part of a research study at the University of Canterbury, and was originally studied for its enhanced seismic performance [1]. The system has been used in buildings internationally and has been proposed for several planned buildings in North America. There have been a number of large-scale experiments done on the system at ambient temperature, and only two experimental studies done in fire conditions. These studies were done by Spellman in 2012 [2] and Costello in 2013 [3].

Spellman completed three beam furnace tests, of which one was reasoned to be an anomaly because of premature failure due to possible manufacturing imperfections. Later, Costello completed a single furnace test along with several ambient beam tests in order to further investigate the failure mechanisms observed. The two experiments from Spellman's research that were not abnormal (Beams A and C) and Costello's furnace test were all modelled and are discussed herein.

In order to capture the complex behaviour of a PT timber beam, a three-dimensional Finite Element Model (FEM) is required. The commercially available general purpose FEM software package, ABAQUS was chosen to build the model as it has a wide variety of material models available, and has the ability to simulate both heat transfer and mechanical analyses. ABAQUS is a valuable research tool which may be used to demonstrate the ability to model structural systems, after which a refined, simpler and less costly model may be developed for use by practitioners. The following section will detail the creation of the model, while the results section discusses the outcomes from each simulation compared to their respective experimental data.

MODEL OBJECTIVES AND SETUP

The primary objective of the model is to provide a setup within a widely-used FEM software that may be used to predict the behaviour of a PT timber beam in fire conditions reasonably accurately. As many experiments were also done under ambient conditions, and beams will often have to be designed for both ambient and fire conditions, it was desirable for the model to provide accurate results in both types of analyses. Additionally, it was desired that the model should be applicable for structures using both LVL and Glulam timber, as the preference for the type of engineered timber varies internationally and both products have been proposed for use with the system. Thus, the material model was simplified in such a way as to ensure this suitability.

Material Models

The timber material's mechanical properties were treated as orthotropic with transverse isotropy in order to be suitable for modelling LVL or Glulam timber. The stiffness reductions suggested in Eurocode 5 Part 1-2 Annex B [4] were used. They are given as functions of temperature and of stress state, varying from tension to compression, as shown in Figure 2.

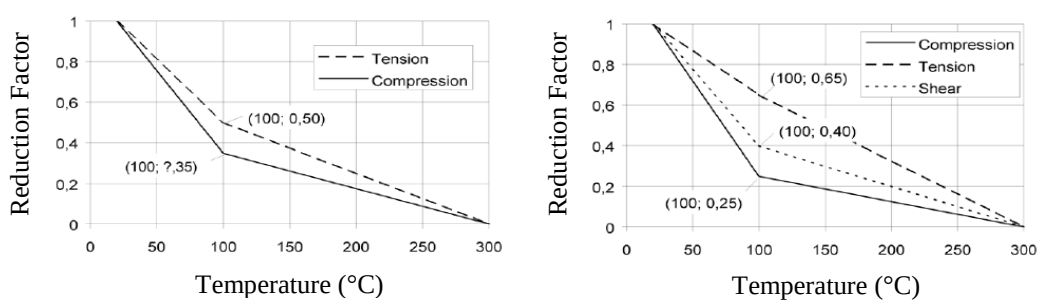

Figure 2: Eurocode 5 Part 1-2 Annex B suggested reduction factors for elastic modulus (left) and strengths (right) [4]

For the *Elastic* material properties, an *Engineering Constants* model was used. This allows the definition of elastic moduli in each orthogonal direction as a function of temperature, a summary of which is given in Table 1. The manufacturer specified an elastic modulus of 13200 MPa and a shear modulus of 660 MPa for the specimens of both Spellman [2] and Costello [3]. The values used for the elastic moduli in the transverse directions, Poisson's ratios, and the rolling shear modulus were determined based on extensive material testing done by Van Beerschoten [5].

The *Engineering Constants* material elasticity definition does not distinguish between moduli for tension and compression, so only one value could be used to represent both. This only affected the definition of the

moduli at 100°C, where the reduction factor differs for tension and compression. The reduction factor of 0.5 (for tension) [4] was used as the compression zone of the beam is largely unaffected by temperature, due to the top flange being insulated from the heat exposure. Additionally, the modification factors are given only for the principal elastic modulus. Thus, it was assumed that the reduction factors could also be used for the other two material directions and all the shear moduli. ABAQUS specifies that the minimum parameter reduction allowed is 1/100th of the initial magnitude, so this requirement was used instead of the Eurocode specification of zero [4].

Table 1: Input parameters for engineering constants orthotropic elasticity as a function of temperature

E_1 (MPa)	E_2 (MPa)	E_3 (MPa)	ν_{12}	ν_{13}	ν_{23}	G_{12} (MPa)	G_{13} (MPa)	G_{23} (MPa)	Temperature (°C)
13200	400	400	0.55	0.55	0.2	660	660	100	0
13200	400	400	0.55	0.55	0.2	660	660	100	20
6600	200	200	0.55	0.55	0.2	330	330	50	100
132	4	4	0.55	0.55	0.2	6.6	6.6	1	300

In order to model the complex plasticity and failure behaviour of wood, several material models were considered within ABAQUS. Ideally, the material model should define failure strengths in each material direction which may be defined as functions of both temperature and stress state. As no such material model exists within ABAQUS, and the desire was to not use a user subroutine to maintain convenience for practitioners, it was determined that the most suitable model available was the *Hill's Potential Function* within the *Plastic* material definition. Similarly to the *Engineering Constants* model, this definition does not distinguish between tensile and compressive strengths.

The manufacturer of the LVL for Spellman's tests gave a compressive strength parallel to grain of 45 MPa and for Costello's test of 38 MPa, while both manufacturers specify a tensile strength parallel to grain of 33 MPa [2, 3]. A purely elastic analysis showed that the material stresses parallel to grain always remained below 33 MPa, therefore 33 MPa was used as the yield strength. This was also used in conjunction with the tensile strength reduction factor of 0.65 [4], as the compressive zone remained largely at ambient temperature (due to three-sided exposure). The tensile stresses perpendicular to grain in the analyses were negligible, therefore the strength for compression perpendicular to grain was used in the transverse directions, along with the corresponding reduction factor of 0.25 [4]. The manufacturer shear strength of 5.3 MPa was used in all shear directions with a reduction factor of 0.4 for longitudinal shear and 0.25 for rolling shear [2,3,4].

The *Plastic* material properties in ABAQUS was defined simply as a baseline yield stress, σ_0, as a function of temperature, as is shown in Table 2. The *Hill's Potential Function* then calculates a corresponding baseline shear strength, $\tau_0 = \sigma_0/\sqrt{3}$. The material strengths in each direction are then defined as discussed above (summarized in Table 2), using a ratio between that stress and the baseline yield or shear strength, respectively. For example, the ratio of the longitudinal strength, $R_{11} = f_{11}/\sigma_0$ and the ratio of the first longitudinal shear strength, $R_{12} = f_{12}/\tau_0$. The ratios are summarized in Table 3 as they are input into ABAQUS.

Table 2: Strength values used for calculation of the Hill's Potential Function ratios

σ_0 (MPa)	f_{11} (MPa)	f_{22} (MPa)	f_{33} (MPa)		τ_0 (MPa)	f_{12} (MPa)	f_{13} (MPa)	f_{23} (MPa)	Temperature (°C)
33	33	10	10		19.1	5.3	5.3	5.3	0
33	33	10	10		19.1	5.3	5.3	5.3	20
21.45	21.45	2.5	2.5		12.4	2.12	2.12	1.325	100
0.33	0.33	0.1	0.1		0.191	0.053	0.053	0.053	300

Table 3 : Input parameters for strength ratios used in Hill's Potential Function for plasticity

R_{11}	R_{22}	R_{33}	R_{12}	R_{13}	R_{23}	**Temperature (°C)**
1	0.303	0.303	0.278	0.278	0.278	0
1	0.303	0.303	0.278	0.278	0.278	20
1	0.117	0.117	0.171	0.171	0.107	100
1	0.303	0.303	0.278	0.278	0.278	300

The thermal properties were treated as isotropic corresponding to heat transfer perpendicular to grain, as very little heat transfer along the grain of the wood was expected due to uniform heating. A wet density of 530 kg/m^3 was assumed for the Radiata pine used in the experiments, and the moisture content (MC) was assumed to vary from 6% to 12%, the former having been measured in wood acclimatized to a lab setting and the latter being a typical value for wood structures. This variance was included as the experimentalists did not report the moisture content of the wood in their theses, and the former matched the thermal results of Spellman well while the latter provided more accurate results for Costello's tests. The latent heat of evaporation of water was included in the thermal model as 271.2 $\mathrm{kJ/kg}_{wood}$ for 12% MC and 135.6 $\mathrm{kJ/kg}_{wood}$ for 6% MC, based on a heat of evaporation for water of 2260 $\mathrm{kJ/kg}_{water}$. The density, conductivity, and specific heat of wood were all defined following the recommendations from Eurocode 5 Part 1-2 Annex B. The jump in the effective specific heat definition was excluded as the evaporation of water was accounted for using the latent heat method, as this provided more stable results.

There were two types of steel used in the experiments. The first was regular strength steel for the loading, support and anchorage plates, and the second was high-strength steel used for the PT tendons. In the experiments and in the simulations, all steel parts remained well within their elastic range and well below 100°C, while degradation in steel tendons is not expected below 200°C. Therefore, a simple isotropic elasticity model was defined with an elastic modulus of 200 GPa and a Poisson's ratio of 0.3.

Analysis Setup

The full simulation was done as a sequentially coupled heat transfer – mechanical analysis. First, the heat transfer analysis was done on just the three-dimensional beam part with heat transfer brick elements. The beam was exposed to the fire on three sides with a convection interaction (using a film coefficient of 25 W/m^{2}K) and with a radiation interaction (using an emissivity of 0.8). The furnace temperature was defined as an amplitude in ABAQUS and used as the sink/ambient temperature for both interactions. The internal cavity surfaces were defined with a cavity radiation approximation where the average surface temperature at each step was used as the ambient temperature for heat transfer. The thermal results were then compared with the available experimental data and used subsequently in the mechanical analysis.

In the succeeding mechanical analysis, the timber beams were modelled in ABAQUS using three-dimensional linear, reduced-integration stress brick elements. The beams in Spellman's two tests were simply supported as was indicated in his thesis [2]. In Costello's furnace test, the beam rested on the concrete frame, creating a more fixed support as it did not allow for much rotation [3]. The boundary conditions were applied through a hard contact interaction between the beam and steel support plates, which had the corresponding degrees of freedom restrained at a coupled reference point.

The PT tendons were modelled as wire elements tied to anchorage plates at each end of the beams, which were then tied to the end faces of the beams. In the first step (after initial), the PT load was applied as a bolt load on a central partition of the tendon which uses a change in length to apply the prescribed load. The change in length was then fixed for all subsequent steps. The vertical load plates were tied to the top of the beams in a four-point loading scheme with the total load applied as a pressure over the plate top surfaces in the second step, and was held constant over the subsequent step. Following the PT and vertical loading steps, the fire exposure step was created. This was input as a predefined field using the output database file created from the heat transfer analysis using incompatible meshes. This allowed the beam mesh to be

optimized for each type of analysis. The numerical results were compared to the available time-deflection data from the experimental results.

RESULTS

There are two sets of results to be discussed. The first are the thermal results compared to the experimental thermocouple readings, and the second are the mechanical results compared to the experimental recorded deflections. Select results are shown herein to demonstrate the model outcomes.

Heat Transfer Analysis Results

The thermal results from the numerical model of Spellman's tests were measured at the same locations as in the experiments, as shown in Figure 3, with 0 mm representing the exposed surface.

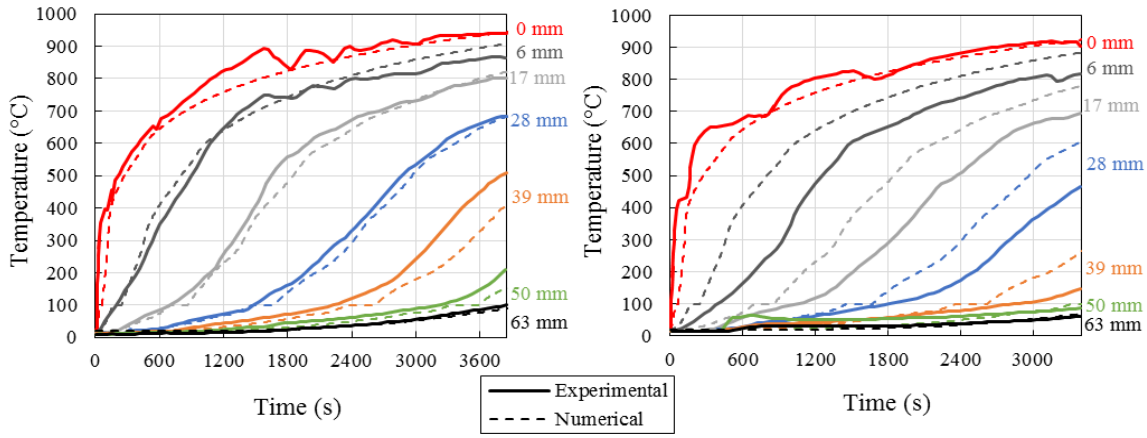

Figure 3: Comparison between experimental [2] and numerical thermal results through the web thickness for Spellman's Beam A (left) and Beam C (right)

The thermal results from the simulation of Spellman's Beam A matched very well with the experimental data, as can be seen in the left plot in Figure 3. The char depth predicted was 44 mm compared to the experiment reported char depth of 47.5 mm. The temperature results for the numerical model of Beam C were notably higher than the thermocouple readings, but did seem to follow a similar trend. Spellman noted in his thesis that the thermocouple readings for Beam C were substantially lower than those measured in the other furnace tests. Upon inspection of the final char depth in the model for Beam C (taken as the 300°C isotherm), the model predicts a depth of 39 mm, while the char depth reported [2] was 40 mm, showing good agreement with the thermal results in this respect.

Mechanical Analysis Results

The numerical deflections were extracted at the mid-span for Costello's test and were the average of the load plate deflections in Spellman's tests to match the data acquisition method of each [2,3]. A summary of the failure times is given in Table 4, and a comparison between the numerical and experimental results for all three tests discussed is shown in Figure 4.

Table 4: Comparison of beam experiment [2,3] and model failure times

Beam Test	Experiment Failure Time (min)	Model Failure Time (min)	Difference (min)
Spellman A	64	66	+2
Spellman C	56	53	-3
Costello	29	29	0

In general, the simulation results showed good agreement with both the failure time and modes predicted, and reasonably predicted the deflection behaviour. All simulations stopped due to excessive deformations at the failure point, where runaway deflections are seen in the results. Costello's beam test was the most

accurately predicted including the failure mode, images of which are shown in Figure 5. In Spellman's tests, the predicted behaviours slightly differed from the experiments, particularly the failure in Beam A which was much more abrupt than the experiment. The differences could be due to the lack of cracking in the numerical model. In the experiment, large cracks propagated through the beam which would have resulted in a more gradual failure, while the model beam remained intact for the entire analysis causing it to have a higher stiffness than observed in the real deflection behaviour.

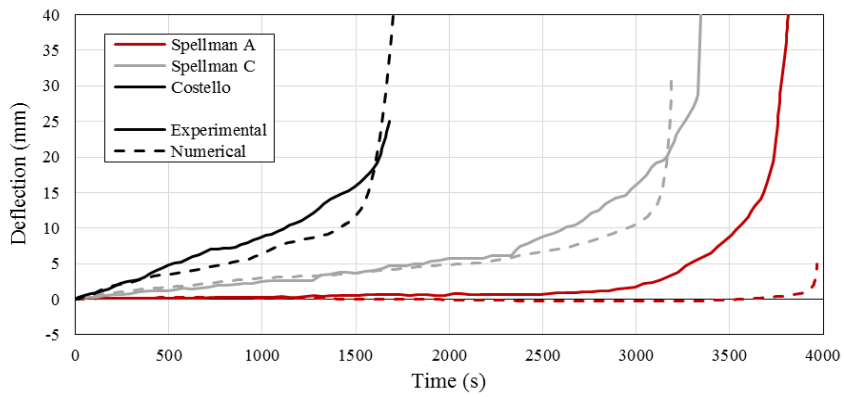

Figure 4: Comparison between experimental [2,3] and numerical deflection results

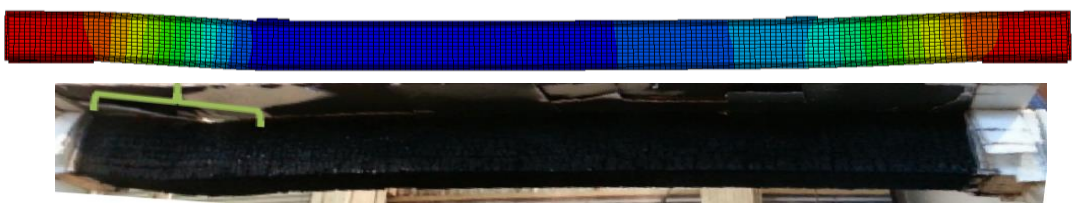

Figure 5: Failure region in left side shear zone in simulation results and experiment (modified from [3])

RECOMMENDATIONS AND CONCLUSIONS

Overall, the approximation of the PT timber beam behaviour in fire conditions using the Hill's Potential Function for plasticity in ABAQUS was reasonably accurate, predicting failure times within 5% accuracy. The model could be improved with a material model which allows for both material directionality and stress state dependence. Additionally, with the implementation of a crack propagation model, the exact failure mechanisms observed in the experiments could be captured. In conclusion, the model presented promising results and could be refined in a designated model to be made available for use by practitioners for performance-based design.

REFERENCES

1. Palermo, Alessandro, et al., "Seismic design of multi-storey buildings using Laminated Veneer Lumber (LVL)", *Proceedings of the New Zealand Society for Earthquake Engineering Conference*, Paper 14, 11-13 March, 2005.
2. Spellman, Phillip, "The Fire Performance of Post-Tensioned Timber Beams", Master's Thesis, University of Canterbury, 2012.
3. Costello, Reuben, "The Fire Performance of Post-Tensioned Timber Buildings", Master's Thesis, University of Canterbury, 2013.
4. "Part 1-2: General – Structural Fire Design", *Eurocode 5: Design of Timber Structures*, European Committee for Standardization, 2004, pp.48-51.
5. Van Beerschoten, Wouter, "Structural Performance of Post-tensioned Timber Frames under Gravity Loading", PhD Thesis, University of Canterbury, 2013.