

**TOWARDS THE DEVELOPMENT OF A COUPLED FIRE-STRUCTURE MODEL FOR FLAX/PP COMPOSITE BEAMS**

Swagata Dutta¹, Raj Das² and Debes Bhattacharyya³

¹ Centre for Advanced Composite Materials (CACM), Department of Mechanical Engineering, University of Auckland, New Zealand. email: sdut182@aucklanduni.ac.bd, website: https://unidirectory.auckland.ac.nz/profile/sdut182

² Sir Lawrence Wackett Aerospace Research Centre, School of Engineering, RMIT University, Melbourne, Australia. email: raj.das@rmit.edu.au, website: https://www.rmit.edu.au/contact/staff-contacts/academic-staff/d/das-dr-raj

³ Centre for Advanced Composite Materials (CACM), Department of Mechanical Engineering, University of Auckland, New Zealand. email: d.bhattacharyya@auckland.ac.nz, website: https://unidirectory.auckland.ac.nz/profile/d-bhattacharyya

Keywords: Polymer composite, Natural fibre composites, Fire properties, Thermo-mechanical analysis, Finite elements

**ABSTRACT**

Fibre reinforced polymer composites, when burned are highly flammable in nature and therefore, pose a serious fire hazard. The situation becomes even worse in case of natural fibres due to the inherent combustibility of the fibres. Modelling fire-structural behaviour of composites poses various challenges due to the complex multi-physics nature of the problem. To date, there have been very few studies to numerically model the fire-structural behaviour of composites; in particular, natural fibre based composites. In the current work, a coupled fire-structural model combining the finite volume (FV) and finite element (FE) methods that captures the essential physics of the problem was developed. The model is based on a multi-physics framework, whereby the essential physics pertaining to the combustion process of the fire and the resultant thermo-mechanical response of the structure, in particular, of natural fibre reinforced composites, has been incorporated. The model predicts the temperature, stress distribution and deformation behaviour of composite beams under combined thermal and mechanical loads. The model is validated against other numerical models.

**1 INTRODUCTION**

Composite materials are susceptible to combustion. High flammability and poor fire resistance are two important characteristics that are associated with polymer matrix composites (PMCs) [1, 2]. This is essentially due to the release of heat, smoke, and toxic fumes [3, 4]. In addition, while an external load is present, softening due to thermal irradiation and pyrolysis of the matrix occurs. Softening of the fibre reinforcement causes distortion, weakening and eventually lead to failure. When these materials are subjected to high temperatures complex physical and chemical processes occur. Processes like glass transition and decomposition take place which may lead to extensive degradation of mechanical properties, such as stiffness and strength. This phenomenon makes the fire behaviour of composite materials an important design criterion. In addition, the changes in the polymer molecular structures are also responsible for the degradation of stiffness and strength [3, 5]. Nonetheless, composite materials can be designed to provide the ideal properties for many structural applications. As mentioned before, one prime barrier to their implementation is the inherent combustibility and the associated risk from fire, specifically for natural fibre composites. The number of tests required to qualify a set of new composite materials for a given application are significant. Therefore, developing robust fire models will not only contribute to predict the thermal response of composite materials, but also to minimize standardized tests. These models can be validated by standard fire tests. Additionally, once the fire behaviour under the heat transfer conditions of a standard fire test is well comprehended, then the same model may be augmented to foretell thermal response under more realistic fire situations. Realistic thermal modelling of fire behaviour provides the designer with the capability to
bespeak innovative, new designs to a designated fire rating, with the minimum effort to expensive standard fire testing [6].

Fire is a very complex phenomenon that can develop in stages of increasing temperature and size before decaying. When FRP composites are exposed to fire, the fire environment can become severely complicated since the inclusion of polymer composite can eventually control the temperature, size and spread of the flame [6]. Cellulosic content also has an important contribution to flammability. A high cellulose content results in high flammability. Lignin and ash, if present in the fibres, lead to greater char formation [7, 8]. The formation of char is a crucial factor since it protects the underline material from thermal exposure and thereby increases the thermal resistance and structural integrity of the composites. Modelling structures under fire conditions requires three generally separate and independent processes to be considered. Firstly, modelling the evolution of fire which signifies the start, propagation and end of fire needs to be performed. Thereafter, the temperature generated at the vicinity of the structure needs to be recorded. Secondly, the heat transfer from the hot gases to the structure needs to be conducted. Thirdly, the investigation on the structural deformation due to the combined effect of mechanical and thermal loading requires significant attention [9]. Since a fire can be considered an extreme situation where a structural element will be dealing with responses nearly at its state of collapse, development of an accurate model capturing pertinent effects of thermal and mechanical loads considering the change in the material properties will not only require a robust, multi-physics modelling framework, but also needs to be validated against large-scale tests in real fire scenarios [6, 9].

Although a considerable amount of research has been directed to understand and characterize the fire properties and flammability of FRP composite materials [1-4, 10-15]. Little is known about the structural behaviour of FRP composites in fire conditions. In most cases, the structural integrity of the composites in fire is assessed by conducting fire tests representative of the structural applications. However, the tests are expensive and limited not only by the experimental conditions, but also by the limited possibility to extrapolate the information obtained from these tests to structural behaviour of composites. In addition, the use of natural fibre reinforced composites (NFRCs) has been predominantly restricted to non-structural parts. So the study investigates the performance of NFRCs in load carrying applications under fire conditions. The prime consideration is to investigate the suitability of natural fibres as potential reinforcement in structural parts under thermal loads. The present paper embarks on developing a coupled fire-structural model combining the finite volume (FV) and finite element (FE) methods that captures the essential physics of the fire-structural problem.

2 SIMULATION OF COMPARTMENT FIRE

Fire dynamics is simulated using “PyroSim” developed by Thunderhead Engineering [16]. PyroSim is a graphical user interface (GUI) to the computational fluid dynamics (CFD) solver “Fire Dynamics Simulator (FDS)” developed by National Institute of Science and Technology (NIST) [17]. FDS can be used as a solver in both large-eddy simulation (LES) and direct numerical simulation (DNS) methods. In this work, the default method ‘large-eddy simulation (LES)’ was used due to its improved handling of turbulence and reasonable accuracy with moderate grid size. A compartment configuration was built as shown in Figure 1 comprising of three windows and a large opening representing passive contact with the outside. The opening was modelled using a user-specified air volume flow rate of 12 l/s. The size of the compartment is 16×8×4 m. The I-beam was placed at the top of the compartment as shown in Figure 1. Table 1 summarises the number of mesh cells and cell parameters used for the mesh sensitivity study. Details of the procedure can be found in our earlier work in [18]. The fire simulation is conducted following the earlier work of Banerjee et al. [19]. In order to conduct heat transfer analysis in the subsequent thermal analysis step in the FE program (Abaqus), time-dependent temperature and/or heat flux data need to be computed at different locations of the I-beam. In this study, both time-dependent adiabatic surface temperature (AST) and incident heat flux (HFI) were computed by placing gas phase devices and solid phase devices, respectively, at eleven different locations and 4 m above the ‘burner surface’ in the FDS simulation domain (Figure 1).
### Table 1: Characteristics of the samples for mesh sensitivity analysis.

<table>
<thead>
<tr>
<th>Grid name</th>
<th>Number of X, Y and Z cells</th>
<th>Total number of cells</th>
<th>Cell size (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>FireGridCon-1</td>
<td>X 60, Y 30, Z 15</td>
<td>27,000</td>
<td>X 0.27, Y 0.27, Z 0.27</td>
</tr>
<tr>
<td>FireGridCon-2</td>
<td>X 80, Y 40, Z 20</td>
<td>64,000</td>
<td>X 0.20, Y 0.20, Z 0.20</td>
</tr>
<tr>
<td>FireGridCon-3</td>
<td>X 120, Y 60, Z 30</td>
<td>2,16,000</td>
<td>X 0.13, Y 0.13, Z 0.13</td>
</tr>
<tr>
<td>FireGridCon-4</td>
<td>X 140, Y 70, Z 35</td>
<td>3,43,000</td>
<td>X 0.11, Y 0.11, Z 0.11</td>
</tr>
</tbody>
</table>

**3 FINITE ELEMENT MODELLING (FEM) OF THE THERMAL AND THERMO-MECHANICAL BEHAVIOUR OF COMPOSITE BEAMS IN FIRE**

In this study, a commercial finite element (FE) software, ABAQUS, was used for thermal and subsequent thermal-stress analysis. In all of the simulations, it was assumed that fire has a strong contribution to the mechanical behaviour; however, neglecting the influence of the mechanical behaviour on the thermal response, thus a one-way coupling was used. Two sets of materials were used in this study, namely steel (ASTM A992)-concrete composite and unidirectional flax/polypropylene-concrete composite. In both cases, the thermal response of the composite structure was modelled. Afterwards, a sequential thermo-mechanical analysis was conducted. The results from the thermal analysis were taken as the input for the subsequent thermo-mechanical analysis. The temperature distribution was made output at a user-chosen time interval. Temperature dependent material properties were considered. Eight-node standard heat transfer brick elements with linear geometric order (DC3D8) were used to discretise the composite beam sections. The length of the composite beam was 8 m. The cross-section of the reinforcing I-beam is shown in Figure 2.
Table 2 shows the characteristics of samples used for mesh sensitivity study. A typical FE model of the I-beam and concrete is shown in Figure 3, where X and Y axes define the cross-sectional plane, and Z axis represents the longitudinal beam axis (out of X-Y plane).

<table>
<thead>
<tr>
<th>Grid name</th>
<th>Total number of elements</th>
<th>Total number of nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>ThermoGrid-1</td>
<td>4,460</td>
<td>6,888</td>
</tr>
<tr>
<td>ThermoGrid-2</td>
<td>8,480</td>
<td>11,886</td>
</tr>
<tr>
<td>ThermoGrid-3</td>
<td>14,780</td>
<td>19,656</td>
</tr>
</tbody>
</table>

Table 2: Characteristics of the samples for mesh sensitivity analysis.

In the subsequent thermo-mechanical analysis, the temperature distribution obtained during the thermal analysis was applied as the thermal boundary condition over the entire structure. The mesh and the number of temperature points were carefully made identical in both of the analysis. This aids in the transfer of results from one analysis to another. The contact between the I-beam material and concrete was modelled using contact interaction with user-specified stiffness coefficients. Two static steps were created. A uniform pressure load of 10 kPa was applied on the top surface of the composite at the first step for a period of 1s in order to integrate the effects of dead (weight) and live loads. In the next step, the combined effect of mechanical and thermal loads on the composite beams was simulated for a time period of 9000 s. The composite beams were discretised using eight-node standard hexahedral linear brick elements with reduced integration and hourglass control (C3D8R element type in Abaqus). The composite beam was modelled as a restrained beam, whereby one end was pinned and the other end was in roller support. The structural analysis was conducted for the entire duration of the fire.
4 RESULTS AND DISCUSSION

4.1 Grid convergence study

The grid sensitivity study was conducted in order to investigate the effect of grid cell size on the adiabatic surface temperature (AST) and incident heat flux (HFI). The variation of the adiabatic surface temperature (AST\textsubscript{max}) with time for different grid cell size was recorded. The description of the samples for the grid convergence study is presented in Table 1. It was observed that as the grid cell number increased, the maximum temperature (AST\textsubscript{max}) obtained gradually increased. However, in almost all the cases, the AST\textsubscript{max} values obtained for successive grid sizes showed very little differences. The maximum temperature (AST\textsubscript{max}) obtained for the grid FireGridCon-3 was 857\textdegree C. The difference in AST\textsubscript{max} values of FireGridCon-3 with FireGridCon-2 and FireGridCon-4 was 0.35\% and 0.16\%, respectively. Therefore, in order to ensure a balance between the accuracy and computational time, the grid FireGridCon-3 was chosen for this study. Similarly, the time to attain the maximum heat flux (HFI\textsubscript{max}) was noted. For lower grid sizes (FireGridCon-3 and FireGridCon-4) the time to attain the HFI\textsubscript{max} was nearly ~13 seconds. The HFI\textsubscript{max} obtained for grid FireGridCon-4 was the highest (104 KW/m\textsuperscript{2}). The difference in HFI\textsubscript{max} between FireGridCon-3 and FireGridCon-4 was 0.2 \%. So from HFI\textsubscript{max} value consideration, grid FireGridCon-3 was chosen in order to save computational time without compromising the accuracy. Likewise, the grid cell size for temperature distribution was determined based on the maximum temperature obtained for each of the grids. The difference between the maximum temperature obtained ThermoGrid-2 (312.42 \textdegree C) and ThermoGrid-3 (312.45 \textdegree C) was 0.03 \textdegree C (0.01 \%). Therefore, in the rest of the simulations, the grid ThermoGrid-3 was chosen for thermal and thermo-mechanical analysis.

4.2 Fire simulation

Simulation of fire is the first step in predicting the performance of composites in load bearing applications under thermal exposure. The fire was modelled in such a manner so that the heat release rate (HRR and the mass loss rate (MLR) of the simulated fire remained fairly steady at ~ 1300 kW/m\textsuperscript{2} and 0.58 Kg/s, respectively, until fully developed (t = 0-300s). Afterwards, the HRR dropped gradually (~900 KW/m\textsuperscript{2}) at 400s. This is the stage where the fire starts its final decay. Later on complete enervation occurs at 600s (MLR = 0 Kg/s). These specific HRR and MLR values were chosen to reflect the typical room fire simulation. Figure 4 illustrates the critical stages of growth and decay of the simulated fire. It can be visualized that during an early period (t = 50s), the flame height was reasonably less than the height of the compartment (Figure 4a). This implies that the I-beam was not in direct contact with the fire. Therefore, at this stage, the intensity of the fire was relatively less at different positions (L1- L11). The heat transfer at this stage was mostly governed by convection and radiation. The intensity of thermal conduction became lower due to the thick layer of air in between the fire plume and I-beam locations. It can be envisioned from the flame location as illustrated in Figure 4(a) that the temperatures attained in the mid positions (L6-L8) would be significantly higher than those at the end positions (L-1-L5, L9-L11). The temperatures for the mid positions varied from 525-725\degree C whereas the temperature attained at the end positions was nearly 200\degree C. This suggests the importance of flame height during the period of fire.

During a later period (t = 300s), the fire grew and became fully developed. At this stage, the flame height increased significantly and the flame was in direct contact with the beam (Figure 4b). The temperature distribution confirms that the mid positions were exposed to 850-900\degree C. However, as before, the temperatures at the two ends were significantly lower. This implies the importance of fire location as well as the flame height on the temperature distribution in a specific part of the compartment. Later on (t = 400s), the fire started to decay. As it can be observed from Figure 4 (c), the region of fire started necking, thereby reducing the intensity. Afterwards, the fire enervation occurred at 600s (Figure 4d). The flame height also reduced considerably at this stage.
Figure 4: Evolution of fire at (a) t = 50s, (b) t = 300s, (c) t = 400s, and (d) t = 600s.

Figure 5: Variation in the adiabatic surface temperature (AST) with time ($t_{\text{start}} = 0, t_{\text{end}} = 650s$) at eleven different locations ($X=8$ m, $Z=4$ m, and $Y=0, 0.8, 1.6, 2.4, 3.2, 4.0, 4.8, 5.6, 6.4, 7.2, 8$ m) in the compartment.

Figure 5 shows the AST values obtained from the simulation during the entire period of fire. It is noted that the middle locations of the I-beam suffered higher heat flux at all stages of fire. This is reasonable since the fire is located directly below these points. The temperatures (AST1) at one end (L1) of the I-beam location were lower than those of the other end (L11) during the entire period. This is due to the shift of fire region to the left end during its evolution. The reason can be attributed to the windows created at the right end signifying passive opening to the outside. The air channelling effect may be responsible for the shift of fire to the left end. This observation implies that depending on the extent of ventilation in a closed space, the fire can cause differences in the temperature distribution at
a particular region in a compartment. It can also be seen that during the evolution of fire, the temperature posed on the I-beam was in a range of 250-450°C.

The results of the incident heat fluxes (HFI) at different locations of the I-beam in the compartment shows that the incident heat fluxes were higher in the middle locations (L5-L7). The heat fluxes are typically in the range from 15-40 kW/m². The maximum heat flux (~90 kW/m²) attained was obtained at L7 and at a very early stage of the fire. This could be viewed as counter-intuitive since at this earlier time, the fire is in the birth stage and far apart from being fully developed. However, the maximum heat flux persisted for a short period of time and was due to the sudden increase of flame height associated with the ignition phase of fire. The average heat flux imposed by the fire was ~30 kW/m². This is significantly lower than the heat flux (~1300 kW/m²) attained by the fire. This suggests that heat fluxes calculated by FDS are possibly low to be used for thermal and subsequent thermo-mechanical analysis. Therefore, in the rest of the study only AST was taken into consideration.

4.3 Thermal analysis of the composite beams

Figure 6 shows the maximum temperature obtained at different locations of the steel-concrete and flax/PP-concrete composite beams using thermal analysis. A difference between the maximum temperature obtained in the lower and upper flange of the I-beam is to be noted. In all of the cases, the maximum temperature at the bottom point was considerably higher than those of the middle and top points, because the bottom points are directly exposed to the heat flux due to fire from below. However, the difference in maximum temperature for the flax/PP system is considerably larger compared to the steel-concrete system.

![Graph showing temperature over time and location](image-url)
It can be seen that the differences in the maximum temperature are significantly higher for the top and middle locations. For the flax/PP-concrete beams, the temperature difference between the bottom (heated) and the top location is \(\sim 260^\circ\text{C}\), which is far greater than that obtained for the steel-concrete beams (\(\sim 160^\circ\text{C}\)), which is primarily due to the difference in thermal conductivity of the beam materials. Steel has a higher thermal conductivity compared to flax/PP, which leads to a lower temperature difference between the top and the bottom locations. This also implies that greater thermal stresses will develop in case of flax/PP composite-concrete beams.

4.4 Thermo-mechanical analysis of the composite beams

To understand the combined effect of thermal and mechanical loads on the composite beams, a thermo-mechanical analysis was performed. Figure 7 compares the Von Mises stress distributions on beam sections (on the X-Y plane of the beams) for the steel-concrete (left column) and flax/PP-concrete (right column) composite beams. For each case, the stresses are symmetric about the web midplane (in the longitudinal Y-Z plane), as expected. It can be noticed that in the case of the steel-concrete composite beam, the thermal and mechanical loads are mostly carried by the steel I-beam during the entire period (\(t = 1 - 9000\) s), which is exhibited by high stresses in the steel I-beam section.

Initially (at 1 s), high stresses developed at the web sections for both types of beams. As stated before, at this stage, a uniform pressure load of 10 kPa was applied on the top surface of the composite beam to integrate the effects of dead and live loads (Figure 7a-b). At 2700 s, severe stress concentration with a maximum value of 196 MPa appeared in the top flange of the steel I-beam. The corresponding maximum stress at the same location for the flax/PP beam was considerably lower (137 MPa) (Figure 7c-d). At 4500 s, the maximum Von Mises stress value were 225 MPa and 176 MPa for the steel and flax/PP composite beams, respectively. The maximum stress was found on the upper flange region of the steel I-beam, and the stresses at certain locations exceeded the yield strength of steel. Hence, permanent plastic deformation or yielding occurred at these specific regions of high stress. This implies that the junction where the upper part of the web and the lower part of the upper flange meets is a critical section in the design of these structures (Figure 7e-f).
At 9000 s, the lower flange of the steel I-beam and the lower part of the concrete were subjected to greater stresses. The maximum stress value attained for the steel-concrete beams was considerably lower than the ultimate strength of structural steel (~ 400-600 MPa). This suggests that the steel I-beam plastically deformed in certain regions, however, did not undergo failure due to exceeding the ultimate strength at the given thermal and mechanical loads (Figure 7g-h). On the other hand, the stresses produced on most parts of the concrete were fairly less (0.0015 – 3.54 MPa) until 2700 s. With time, the lower part of the concrete block had greater stresses (77 MPa). This is a significantly higher stress value for concrete. The differences in the coefficient of thermal expansion between steel and concrete create high compressive stresses in the interface region. Steel having a higher coefficient of thermal expansion induces compressive stresses on the concrete interface.

For the case of the flax/PP-concrete beams, the behaviour was significantly different. It appears that high Von Mises stresses were mostly associated with the lower flange and the lower part of the concrete block. The maximum value of stress at 4500 s was 175 MPa and was observed on the bottom face of the lower flange. The stress generated on the concrete for the flax/PP beam is higher (~75-89 MPa). The resultant stress is contributed by the difference in the coefficient of thermal expansion between flax/PP composite and concrete, in addition to the thermal and mechanical loads. However, as observed in the thermal analysis, the flax/PP composite is likely to decompose beyond 70 minutes (4200 seconds). Since no decomposition model was introduced in these simulations, the prediction of the fire behaviour of the flax/PP-concrete composite beam was limited beyond the decomposition temperature, based on the assumption of no decomposition. However, degradation of material properties was incorporated in the model by temperature dependent properties.
Figure 7 Comparison of the Von Mises stress distribution between the steel-concrete (left column) and flax/PP-concrete (right column) composite beams.

Figure 8 summarises the displacement of the composite beams, via showing the variation of displacements in three different directions, namely transverse (x), loading (y) and axial (z) directions. The displacements of the three monitoring points in the transverse (d_x) direction for steel-concrete and flax/PP-concrete composite beams are shown in Figure 8a-b, respectively. The displacements show similar trends. During the early period (0-2000 s), the displacement values were in the negative x-direction for all of the points. This is due to the relative expansion and shrinkage of the beams on the opposite sides of the Y-Z plane (symmetric web plane) due to the presence of the passive opening on the right side (or positive x-axis) of the beam which promotes cooling. The stiffness of the interface between the reinforcements (steel or flax/PP composite) and the concrete also contributed to this behaviour. Afterwards (2000 s onwards), all the points show positive displacements (extension) in the transverse (x) direction. The maximum transverse (x) displacement value (~ 53 μm) for the flax/PP-concrete composite beam was significantly higher (about four times) than that of the steel-concrete beam (~ 13 μm). The combined thermo-mechanical loading caused the beam to bend in the web plane (Y-Z plane) with deflections in the loading direction (y), which also produces extension (positive displacement) in the transverse direction (bulging). An important difference also lies between the two material systems in the displacement values for the top point (point 3) in x-direction (d_x). While for the steel-concrete composite the displacement was mostly positive, for the flax/PP-concrete composite, it was mostly negative. For the displacements of the three points in the loading (y) and axial (z) direction, it is found that for both the steel and flax/PP composite materials, the trend in displacement in the loading direction (d_y) and beam axial direction (d_z) is also similar. The displacement trend for both the directions and for both the beam types can be characterised as an initial contraction period where the displacements are negative followed by an expansion period with all the displacements are positive. The maximum displacement for the flax/PP-concrete composite in the loading direction was 2.4 mm and occurred at the middle point of the beam section (point 2). In contrast, the maximum
displacement for the steel-concrete beam was nearly half, around 1.2 mm, and occurred at the top point (point 3). This is due to the larger bending stiffness of the steel-concrete beam that resulted in lower displacement. The elastic properties (i.e., strength and stiffness) of the materials together with the properties of the interface between the reinforcements (steel, flax/PP) and the binder (concrete) had a dominant role in determining the deformation behaviour of the composite beams. In all of the cases the displacement values reduced sharply after attaining the maximum value. This phenomenon can be attributed to the elastic recovery of the beams. However, the recovery was mostly observed in the loading (y) and axial (z) directions.

Figure 8: Comparison of the displacements in transverse ($d_x$), loading ($d_y$) and axial ($d_z$) directions between the steel-concrete (left column) (a, c, e) and flax/PP-concrete (right column) (b, d, f) composite beams.
4.5 Model Validation

There have been few studies related to modelling the fire-structural response of composite beams coupling the finite volume and finite element methods. Nonetheless, the current methodology of analysis to predict the fire responses of composite beams was validated by conducting a simple through-thickness heat transfer analysis on a steel I-beam according to the earlier work of Banerjee et al. [19]. Figure 9 shows the nodal temperature (NT) distribution in ASTM A992 steel at a time of 1666 s (~28 mins). The time dependent ASTs obtained at specific beam locations from fire simulation in FDS was applied as a thermal boundary condition at the lower flange of the I-beam. It can be seen that the temperature gradually decreased from the lower flange to the upper flange region. The maximum temperature of $T_{\text{max}} \approx 425^\circ \text{C}$ was obtained at the lower flange. The temperature at the web region of the I-beam was in the range of 200-300°C. The nodal temperature distribution obtained in the present study is similar (within ~ 4.8%) to the earlier work of Banerjee et al. [19]. The maximum temperature ($T_{\text{max}} \approx 404.5^\circ \text{C}$) obtained in that work is also fairly close to the value (~ 425°C) obtained in this study. The comparison reveals that the concept of using AST is a reasonable approximation to simulation of fire conditions.

![Figure 9: Nodal temperature (NT) distribution in an ASTM A992 steel beam with ASTs applied at the lower flange as thermal boundary conditions at 1666 s.](image)

Thereafter, a thermo-mechanical analysis was conducted on an axially restrained steel beam subjected to heating according to reference [20]. The normal force versus temperature (Figure 10) curves generated at the support and midspan of the beam show similar trends to those obtained in [20]. The normal force at the midspan matched well (within ~ 2%) with that reported in reference [20] until 200°C. However, after 200°C, there was a slight deviation in the normal force value. Figure 11 presents the deflection of the axially restraint steel I-beam in the midspan. It can be noticed that as the normal force increased in the midspan (Figure 10), the deflection of the beam at the midspan location continued to increase (Figure 11). The deflection variation with temperature shows similar trend to that reported in [20]. However, the deflection values significantly differ at lower temperatures (0-250 °C) from that reported in [20]. At 250 °C, the two deflection curves intercept show good agreement. Beyond 250 °C the deflection curves also show slight difference. The deviation of the deflection curves after 250 °C can be explained by the differences in the normal forces attained in the midspan. However, it is noteworthy that the deviations of the deflection curves at the initial period (0-250 °C) existed, since the force-temperature curves showed similar results. Therefore, from the above it is established that the present model is capable of predicting the thermal and thermos-mechanical behaviour of the composite beams with reasonable accuracy.
5 CONCLUSIONS

The paper develops a coupled fire-structural model combining the finite volume (FV) and finite element (FE) methods that captures the essential physics of the combustion process and the resultant thermo-mechanical response of the structure subjected to fire conditions. The model predicts the temperature, stress distribution and deformation behaviour of composite beams under combined thermal and mechanical loads with reasonable accuracy. The model is validated against other numerical models.

ACKNOWLEDGEMENTS

The authors would like to thank the Ministry of Business, Innovation and Employment (MBIE) for the financial support. The authors wish to acknowledge the support of NeSI high-performance computing facilities at the University of Auckland.
REFERENCES