

Computer simulations of timber stressed-skin panels using finite element

Christophe Gerber
Research Associate
Faculty of Engineering, University of Technology
Sydney, Australia

Keith Crews
Professor
Faculty of Engineering, University of Technology
Sydney, Australia

Christophe Sigrist
Professor
Berner Fachhochschule, School of Architecture, Civil and Wood Engineering
Biel, Switzerland

Summary

A Finite Element Model (FEM) can provide a helpful and accurate tool for acquiring a better understanding of complex structures such as stressed skin panel (SSP) systems and can contribute towards saving costly experiments. The FEM discussed in this paper is an outcome of a comprehensive research conducted on SSP structures at the University of Technology, Sydney between 2002 and 2007 (Gerber 2007). Albeit some conceptual recommendations, which characterise the boundaries of FEM, it is capable of predicting/simulating the behaviours of SSP systems in many loading configurations. A good agreement between the FEM simulations and experimental data (structures in the “healthy” state) has been demonstrated. It has also been established that FEM can successfully simulate the effects of discontinuities in SSP sheathing (structures in “damage” state).

1. Introduction

Stressed-skin panel (SSP) systems are highly complex, orthotropic, statically indeterminate and multi-layer assemblies. Furthermore, each member of the SSP system – joist and panel – exhibits orthotropic, viscoelastic properties, and non-linear behaviour under certain conditions. In addition to this, structural timber is randomly affected by natural growth characteristics which can be described as “defects” in terms of uniform structural properties. Several researchers, for example, Polensek et al. (1972) and Vanderbilt et al. (1974), in describing wood joist systems, acknowledged that such constructions are emphatically complex because of the material properties on the one hand, and the intricate interactions between the floor members on the other hand. Therefore, accommodating SSP systems into a finite element model (FEM) is an arduous task and inevitably represents an idealisation of the physical structures.

This paper briefly introduces and discusses the concept of an FEM, that has been developed with the ANSYS software package (ANSYS Inc. 2005), imposing a series of restrictions on the choice, definition and control of the element attributes and the solution mode. For example, the interlayers between the joists and the sheathing are modelled with contact element technology, to which restricted characterisation is possible. The capability of the FEM to simulate the linear-elastic behaviour of the SSP structures is also presented in this paper. Furthermore, the modelling of gaps in the SSP sheathing(s) is also discussed hereafter.

2. Finite element model (FEM)

2.1 Fundamentals and concepts

A computer model inherently represents an idealisation as well as a compromise of the “real” structures. In concept, the FEM needs to comply with ANSYS rules and principles (ANSYS Inc.

2005). This imposes restrictions and compatibility requirements. The construction of the FEM required the characterisation of the type, the geometric parameters, and the material properties of the element. These main features of these aspects are concisely summarised hereafter:

Element type: fundamental characteristics of the finite element, such as the number of nodes, the degree of freedom of the nodes and the field of analysis.

Real constant: characterisation of the elements, for example, it can define geometric aspect and/or stiffness properties of some elements.

Material properties: mechanical properties of the element material.

Furthermore, mapped meshing has been chosen, as opposed to free meshing, because this enables greater control over the dimensions and shape of the mesh (size of the element), the number of elements and the transition and connectivity between the elements.

An FEM, which matches the high level of sophistication of SSP systems, has been successfully developed. The FEM succeeds in accounting for the complex composite assembly of I-joists and engineered wood product sheathing using elements from the ANSYS library (ANSYS Inc. 2005) and characterised accordingly. The interlayers are modelled with contact element technology. The boundary conditions – loads and support conditions – are directly imposed on selected nodes.

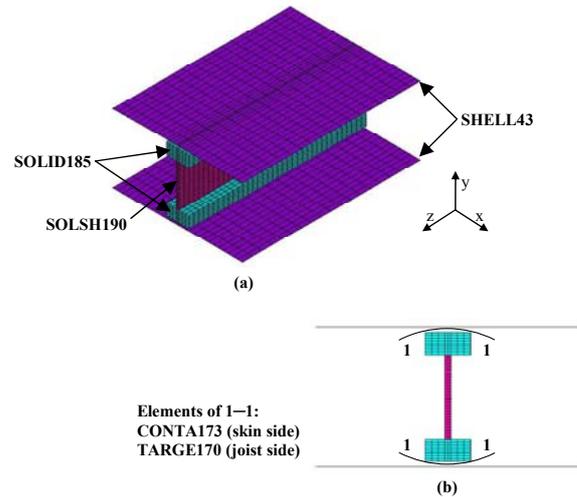


Fig. 1: Element attributes of the FEM.

The FEM associates five different element types: SOLID185, SOLSH190, SHELL43, CONTA173 and TARGE170 (Fig. 1). Further details about these elements are available in ANSYS handbook (ANSYS Inc. 2005).

2.2 Material properties of the members of the finite element model

Sets of orthotropic matrices have been assembled to characterise the mechanical properties of the FEM members (Table 1).

Table 1: Material properties of FEM members.*

Description		MGP15	9-mm plywood	12-mm plywood	15-mm F11 plywood	19-mm particleboard	22-mm oriented strand board
Modulus of Elasticity in Pa	E_x	955E6	955E6	955E6	3150E6	3350E6	4719E6
	E_y	955E6	5775E6	9135E6	8500E6	2054E6	4000E6
	E_z	15200E6	1260E6	5040E6	10500E6	3900E6	8136E6
Poisson's ratio	ν_{xy}	0.569	0.05	0.05	0.1	0.3	0.1
	ν_{yz}	0.029	0.02	0.02	0.3	0.35	0.2
	ν_{xz}	0.029	0.30	0.30	0.5	0.35	0.5
Shear modulus in Pa	G_{xy}	100E6	525E6	525E6	52.5E6	85E6	100E6
	G_{yz}	940E6	52.5E6	52.5E6	525E6	850E6	1000E6
	G_{xz}	940E6	52.5E6	52.5E6	52.5E6	85E6	100E6

*Source: Australian standards and specialised literature.

2.3 Parameterisation

The “parameterisation” of the FEM can be viewed as the operation with which the “imperfections” of the physical structure are introduced into the model. It aims at identifying – in justified manners

– the best calibration coefficients of some aspects/parameters of the model in order to improve and/or refine the predictive/simulative performance of the FEM.

a) *Longitudinal stiffness of the model*

The FEM exhibits a tendency to underestimate the deflection in comparison to the experimental measurements. Presumably, the FEM fails to simulate the shear deformation, in particular that of the I-joist web. Such distortion, which has been identified during the laboratory investigation, can be significant (Gerber 2007).

It is thus suggested that the behaviour of the I-joists requires some measure of parameterisation (Gerber 2007). This is carried out by applying a calibration coefficient upon the mechanical properties of the I-joist web. Table 2 summarises the coefficients that have been identified for 200-mm and 356-mm I-joists. The outcomes of this calibration indicate that the characteristic of the SSP systems may also influence the FEM performance. This could be related to the composite interaction/features of the SSP systems (members and construction).

Table 2 : Longitudinal calibration coefficients.

I-joist type	SSP system	
	Open section	Box section
200 mm	0.220	0.255
356 mm	–	0.339

b) *Orthogonal stiffness of the model*

It has been identified that ANSYS lacks the capability to simulate the behaviour of SSP systems in the orthogonal direction in an accurate manner. Such response may be related to ANSYS idealisation of the tongue-and-groove connection of the I-joist web into the flanges. Presumably, ANSYS ignores any torsional distortion, whereas this connection is weak and deformable. Some measure of distortion may also occur in the interlayers between the joists and the sheathing(s). Thus, it is anticipated that some rotational displacement takes place, particularly under eccentric loading configurations.

In order to improve FEM responses, it has been chosen to impose a calibration factor on the orthogonal mechanical properties of the sheathing, each panel material requiring a coefficient (Table 3). The values of the “correction” is severe, ie., the mechanical properties of the sheathing is reduced to a fraction of their actual values. A series of aspects may explain the significance of these corrections, eg., ANSYS ideal environment, ANSYS element behaviour and interaction, SSP construction features, and SSP torsional distortion. There is however no certainty about the magnitude of the influence of each parameters

Table 3 : Orthogonal calibration coefficients.

F11 plywood	0.0157
Particleboard	0.0880
Oriented strand board	0.0700

Both calibration operations indicate that the extension of the FEM to SSP structures with different construction parameters is curtailed by the need to determine the calibration coefficients first. To some extent, this reduces the FEM power for researching and developing new SSP constructions.

3. Evaluation of the finite element model’s capability

After parameterisation, the simulation capability of the FEM is evaluated by comparing the FEM estimates of the mid-span deflection to the laboratory data of the specimens in healthy state (Gerber 2007) – deflection is scaled to a unit load. The evaluation corresponds to a quantitative and qualitative evaluation: (1) the approximation of the deflection of the girder onto which a load is applied, and (2) the prediction of the deformed shape, which is characterised by the mid-span deflection of the girders. The acceptability range is set at $\pm 10\%$.

Looking at uniformly distributed line-loadings (third- and centre-point loadings), the comparison indicates that the FEM simulations are generally acceptable. Of 60 simulated results, 54 (90%) are within $\pm 10\%$ of the test data, another 4 (7%) are within $\pm 15\%$, while the last 2 (3%) are within $\pm 20\%$. The quality of the agreement also suggests that the deformed shape is simulated accurately.

Examining the point load configurations, the FEM simulated deflections of the girder directly under load are generally acceptable. Of 30 computed results, 25 (83%) are within $\pm 10\%$ of the test data, another 4 (13%) are within $\pm 20\%$, while the last one (4%) is within $\pm 25\%$. Most of the unloaded girders generally exhibit acceptable variations, indicating that the deformed shape is well simulated. Some of the unloaded girders may however show a significant deviation – these variations may not be relevant because these girders experience very small deflection. Furthermore, this may also confirm that the FEM ideal character and environment may prevent simulating some physical aspects of the specimens such as the torsional rigidity and the support conditions.

In Fig. 2, a histogram is used to depict the evaluation of the FEM. It illustrates that the FEM tends to overestimate the deflection slightly, this being epitomised by the location of the median in the positive zone of the x-axis – marginal variation.

Overall, it can be concluded that the calibrated FEM is capable of providing accurate estimates of the deflection magnitude and deformed shape of the specimens.

In Fig. 3, graphical depictions (perpendicular profiles of deflection at mid-span – load unit scale) of representative FEM simulations are presented. These graphs illustrate the acceptability of the FEM.

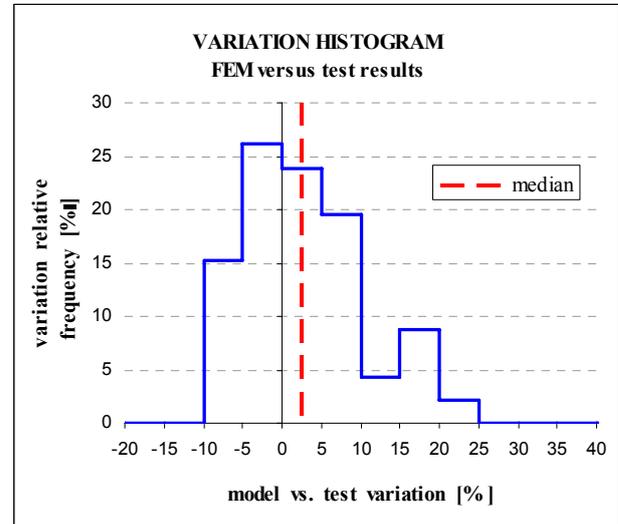
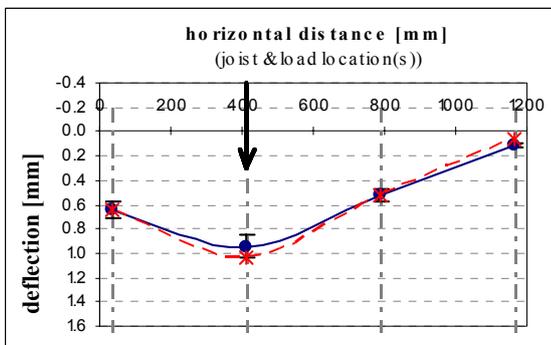
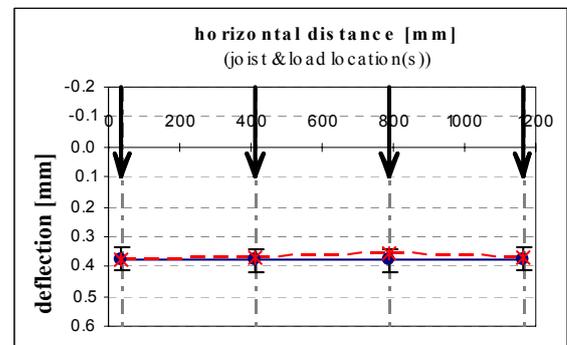


Fig. 2: Histogram of the FEM evaluation.

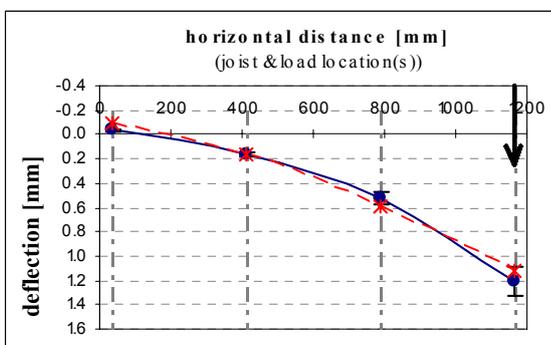
C03&04-family (point load applied on joist J2)



C13-series (line loads (third-point loading))



C13-series (point load applied on joist J4)



C12-series (point load applied on joist J3)

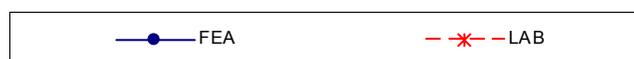
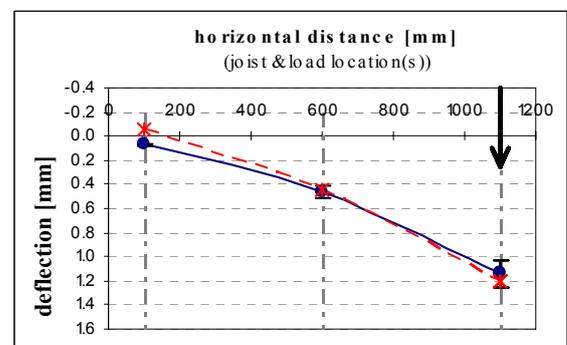


Fig. 3: FEM evaluation – orthogonal profiles of deflection at mid-span – healthy state.

4. Introducing a discontinuity in the sheathing

Discontinuing the sheathing of floor systems – damaged state – decreases their global stiffness and has been as such observed in the subject research (Gerber 2007). For laboratory investigation, a discontinuity is created by inflicting a cut to the sheathing 150 mm away from mid-span. In the FEM, the discontinuity is introduced by leaving nodes of the areas, which model the sheathing, unmerged. No further characterisation is required because, in regular solution modus, ANSYS permits the “interpenetration” of the elements (ANSYS Inc. 2005).

The evaluation of the FEM capability in accounting for sheathing discontinuities is carried out by comparing the mid-span deflections of the FEM and the laboratory measurements, the data being characterised to a unit load. The performance of the FEM is not as accurate for the damaged state as for the healthy state. Nevertheless, in view of the scale of the deviations, the overall performance of the FEM remains acceptable. Of 90 computed deflections, 38 (42%) are within $\pm 10\%$ of the measured deflections, another 24 (27%) are within $\pm 15\%$, another 18 (20%) are within $\pm 20\%$, while the last 10 (11%) are within $\pm 25\%$. These deviations also suggest that the FEM provide reasonable simulations of the deformed shape of the specimens in damaged state.

The deviation between the model simulations and the test results is further depicted in Fig. 4. The median, located notably on the positive side of the x-axis, indicates that the FEM tends to overestimate the mid-span deflection.

Furthermore, the outcomes of the FEM evaluation also suggest that the practice used for the introduction of the sheathing discontinuities in the FEM is suitable.

Representative FEM simulations are depicted in Fig. 5 (mid-span deflection per load unit). These graphs illustrate the satisfactory performance of the FEM.

5. Concluding summary

This paper has presented an FEM developed in ANSYS environment. It has been identified that some calibration coefficients are required in order to incorporate the “imperfections” of the real structure into the FEM. On the one hand, the calibration permits to enhance the simulative performance of the FEM. On the other hand, it may curb the range of applicability/transposability of the FEM.

The FEM evaluations have demonstrated that the FEM can simulate most load situations within clear boundaries. The FEM simulations are acceptable both quantitatively and qualitatively. It is also capable to accommodate the effect of discontinuities – “damage” – in the sheathing successfully.

Albeit some conceptual constraints and calibrations – characterising the boundaries of the FEM, it has thus been demonstrated that the FEM provides a helpful and accurate assistance for acquiring a better understanding of SSP structures. Such modelling tool can also contribute towards saving costly experiments. In these circumstances, the FEM corresponds to the investigation and development tool while the laboratory work becomes the validating aspect of the research.

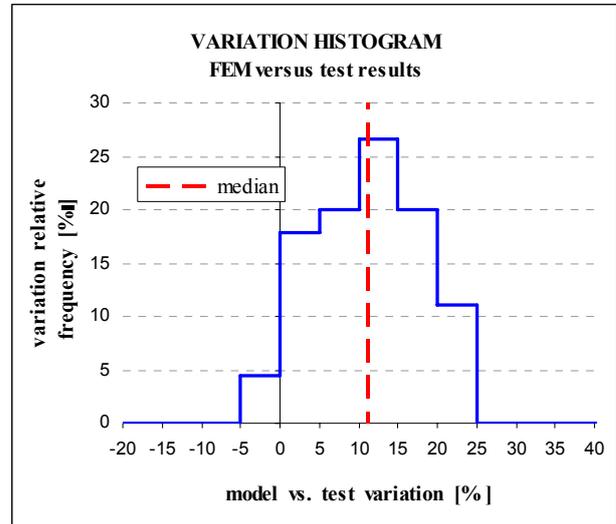
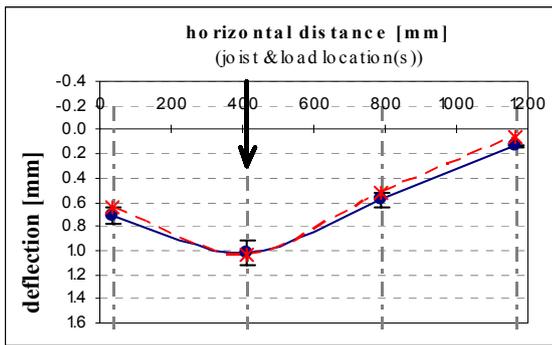
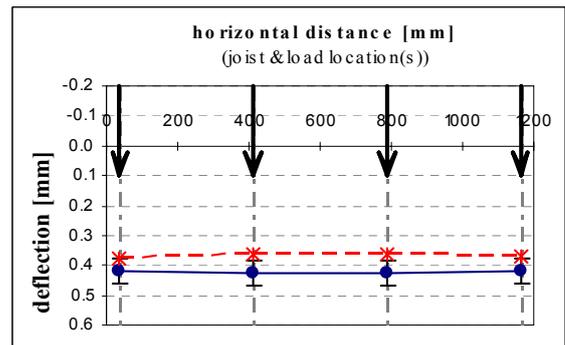


Fig. 4: Histogram of the FEM evaluation (2).

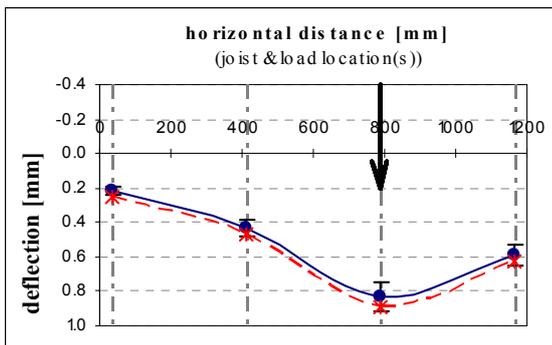
C03&04-family (point load applied on joist J2)



C13-series (line loads (third-point loading))



C13-series (point load applied on joist J3)



C12-series (point load applied on joist J2)

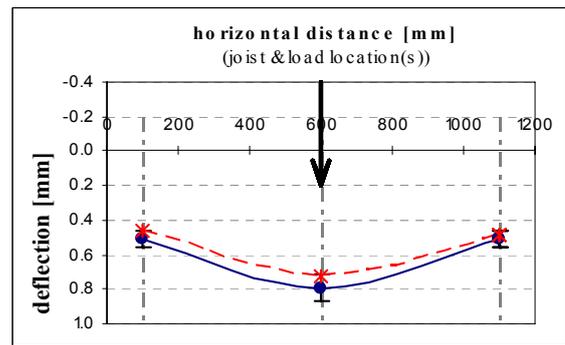


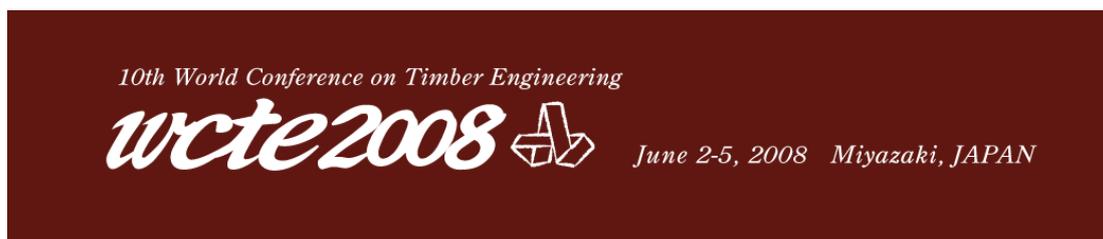
Fig. 5: FEM evaluation – orthogonal profiles of deflection at mid-span – damaged state.

6. Acknowledgement

The authors gratefully acknowledge the support of the Académie suisse des sciences techniques, Switzerland, and the University of Technology, Sydney, Australia; and the contributions, such as technical advice and materials, of Trus Joist™ a Weyerhaeuser Business, Purbond AG, Kronoply GmbH & Co. KG and Paslode Australia Pty.

7. References

- [1] ANSYS Inc. 2005, ANSYS®, Release 10.0 edn, Canonsburg (PA), USA, < <http://www.ansys.com/> >.
- [2] Gerber, C. 2007, 'Wood Stressed-Skin Panels: An Investigation into their Behaviour, Load Distribution and Composite Properties', Ph.D. Thesis, Faculty of Engineering, University of Technology, Sydney (NSW), Australia.
- [3] Polensek, A., Atherton, G. H., Corder, S. E. & Jenkins, J. L. 1972, 'Response of Nailed Wood-Joist Floors to Static Load', *Forest Products Journal*, vol. 22, no. 9, pp. 52–61.
- [4] Vanderbilt, M. D., Goodman, J. R. & Criswell, M. E. 1974, 'Service and Overload Behavior of Wood Joist Floor Systems', *Journal of the Structural Division*, vol. 100, no. ST1, pp. 11–29.



Welcome to WCTE2008 in Miyazaki,JAPAN

Dear WCTE2008 Delegates,

The World Conference on Timber Engineering showcase the latest technology, innovative design and research relating to the timber engineering field. WCTE is held every two years providing the ultimate forum to exchange information and learn cutting edge technology from leaders in the field of timber engineering worldwide. The participants of the convention include leading architects, engineers, educators, and timber engineering researchers. On the administrative side, the leaders of the convention are respected architects, engineers, educators, timber engineering researchers as well as business leaders in the field of timber engineering and related fields.

There has been a growing movement to utilize biomass, in the face of global warming, a serious shortage and depletion of fossil resources, and the consequent rise in prices. A typical biomass resource is wood. It is a resource converted from carbon dioxide in the atmosphere through photosynthesis of solar energy. This circulating resource returns to carbon dioxide through combustion or biodegradation. Wood species and its uses are remarkably diverse. The most important aspect of wood—renewable or sustainable resources using solar energy—is that human beings commit themselves to their production. The use of wood for timber engineering has direct influence on human life, affecting people involved and producing wider ripple effects on the community and various fields. In other words, its role is driving force and efficiency is not the only measure. We need to take account of forests, which are the place for production, and of the ecological system, in which living creatures co-exist. Deeply concerned with issues of climate and environment, we must be always aware of the need for cooperation in terms of “space” (in same generation) and “time.” (beyond generations).

The 10th WCTE Conference 2008 in Miyazaki, Japan received many abstracts and proceedings for presentations with topics of interest spanning the spectrum of the timber engineering field.

We do hope these reports are effective and instructive for mutual understanding between these sectors and will also connect into “the next ones”.

WCTE 2008 Chair
Dr. Takanori Arima