Proceedings of the 8th meeting of the
NORTHERN EUROPEAN NETWORK FOR WOOD SCIENCE AND ENGINEERING (WSE)

September 13-14, 2012
Kaunas, Lithuania

Edited by Antanas Baltrušaitis and Kristina Ukvalbergienė

Wood Science and Engineering

TECHNOLOGIJA
KAUNAS • 2012

Title
Proceedings of the 8th meeting of the
Northern European Network for Wood Science and Engineering (WSE)
September 13-14, 2012
Kaunas, Lithuania

Editors
Antanas Baltrušaitis
Kristina Ukvalbergienė

Publisher
Technologija
Kaunas University of Technology
Studentų str. 54
LT-51424 Kaunas

Cover photo
Gintaras Keturakis
Valley of Junipers (age 90), Kauno Marios Regional Park

Cover Design
Evelina Garliauskienė

Published with support from SNS – The Nordic Forest Research Cooperation Committee

ISBN 978-609-02-0581-5

NUMERICAL MODELLING AND EXPERIMENTAL VALIDATION OF DENDROLIGHT® CELLULAR WOOD MATERIAL

Labans, E.[1] & Kalnins, K.[2]

ABSTRACT

The aim of a current research is to estimate most appropriate numerical modelling approach in order to predict mechanical behaviour of small scale DendroLight® specimens under compression/bending loading conditions. For this task several finite element (FE) modelling techniques has been utilised assessing the supremacy of shell our solid finite element models in ANSYS and ABAQUS commercial software. Due to complicated DendroLight® structure topology a special attention has been devoted to experimental validation of numerical models. Several series of DendroLight® cellular specimens have been tested in longitudinal and transverse compression along with four point bending sandwich samples with HDF (High Density Fibreboard) skins. It has been confirmed that both finite element model types predict specimen mechanical behaviour sufficiently well compared to results obtained experimentally. Obtained differences usually not exceed 20% margin. Moreover it has been concluded that solid elements are more appropriate as they delivers more extensive information about the stress/strain distribution compared to the shell element model.

Key words: DendroLight®, cellular wood structure, finite element modelling, mechanical testing, ANSYS, ABAQUS

INTRODUCTION

DendroLight® is a unique wood material specifically developed as core material for furniture industry. Manufacturing started at the year 2010 in Ventspils, Latvia. It is made from profiled/perfored wood boards stacked in perpendicular layers and then sliced once more in plates perpendicularly to the board's layers. The main advantage of such a solution is significant reducetion of structural weight (up to 40 %) comparing to

[1] Phd student, Institute of Materials and Structures, Riga Technical University, Azenes 16/20, Riga, LV-1048, Latvia Tel: +371 28319475, Fax: +371 67089254, E-mail: edgars.labans@rtu.lv

[2] Dr.sc.ing. Leading researcher, Institute of Materials and Structures, Riga Technical University, Azenes 16/20, Riga, LV-1048, Latvia Tel: +371 28319475, Fax: +371 67089254, E-mail: kaspars.kalnins@sigmanet.lv

conventional timber. Thus such a cellular wood material has a potential to be utilised in near future in load bearing structures, like walls and floors as core of sandwich panels. For this task a reliable structural analysis methodology is required, able to assess detailed geometry component (like profiled board web thickness) influence over the stiffness for large scale structure.

Computer software based on finite element method has invaluable influence over modern engineering design practice in various steps from design of a wood furniture pieces up to building structures. Computer analysis allows achieving higher accuracy and reducing the calculation time for complicated models comparing with the analytical solution. The finite element analysis becoming more exploited for conducting research related to development of innovative wood products. For example recently Persson (2008) devoted his doctoral thesis for elaborating of mechanical behaviour of the wood cell walls using an ABAQUS code. Moreover the finite element analysis has been acknowledged by number of researchers as convenient method for analysing stress/strain distribution in wood connections with the steel fasteners (Nishiyama and Ando 2003, Resch and Kaliske 2010). Moreover Zhou et al (2010) analysed seismic behaviour of the wood structures employing ANSYS code. A most comprehensive summary is given by Mackerle (2005) in review article regarding the wood product development and researches with emphasis to FEM application.

In case of DendroLight® the ANSYS (2009) and the ABAQUS (2009) software were utilised for simulation of mechanical behaviour of cellular wood structure under compression and bending loading conditions. DendroLight® has not jet been widely investigated. A basic set of material properties are described by manufacturer (2012) and Iejavs et al (2011) has conducted experimental investigation on a large scale wood sandwich panels and concluded that the cellular wood material could be successfully applied as the core material.

In this particular paper a comparison of different FE techniques for modelling of DendroLight® structure has been given. It has been evaluated the time consumption for model creation, meshing efficiency and stress/strain information amount acquired from the model. To assess the model accuracy numerical results were compared with the values obtained in experimental tests.

MATERIALS AND METHODS

FEM modelling

Mechanical properties employed for modelling of wood material are as follows: The modulus of elasticity in longitudinal direction - 11 GPa; the modulus of elasticity in transverse and radial direction $E_T= E_R$=0.6 GPa, the shear modulus: $G_{LR}=G_{LT}$=0.34 GPa, G_{RT}=0.2 GPa and aPoisson`s ratios $\mu_{LR}= \mu_{LT}$=0.34, $\mu_{RT}$=0.03. The Young`s modulus is taken according to the timber strength class C24 in European standard EN338 (2003). Other properties evaluated using the Wood handbook (1999). Such properties are more characteristic for pine, however accounting influence of the wood species was out-of-scope in current research, because manufacturing technology currently is set to utilise both pine and spruce raw material.

Corresponding isotropic mechanical properties have been assigned also to High Density Fibreboard (HDF) skins: E_{HDF}=3.2 GPa, μ_{HDF}=0.3.

Numerical model in the ANSYS software has been made employing SHELL181 elements with transversal isotropic wood material properties (Figure 1.a). Elements were connected using node-to-node connections at coincident points. The cellular wood structure created in sequence of real production process, starting with profiled board modelling, forming layers and cutting blocks into DendroLight® layers. Boundary and loading conditions have been set according to experimental test set up. The ABAQUS software has been used for modelling DendroLight® structure from solid type elements. Due of prismatic component forms mainly tetrahedral element shapes have been developed in meshing process. Profiled boards and cellular wood core surfaces have been connected together with surface-to-surface bounding. Mesh size step were set to magnitude of 8 mm. Largest of modelled specimens *B2* (see details in Table 1) has about 12000 shell elements in ANSYS and 15700 solid elements in ABAQUS.

Structural loads were assigned to sets of nodes with jointed deflections along vertical direction. It allows simulating uniform pressure on top plate in case of compression specimens and line loads with rollers in case of bending specimens. Boundary conditions for compression specimens have been assigned by selecting all lower nodes and restricting their translations along all axes. In case of bending specimens boundary conditions have been applied only at the ends of the sandwich beam. Bounding conditions allows the rotation of all nodes and translation of nodes along longitudinal direction for one end of the beam type specimen.

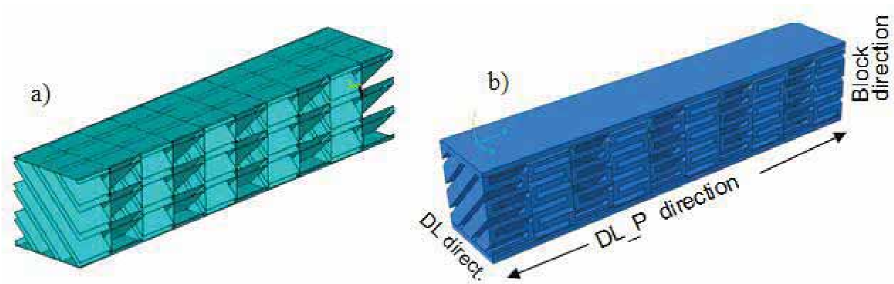


Fig.1. Finite element models for sandwich beam specimens a- ANSYS shell 281 model, b-ABAQUS – solid 3D model

In order to reduce the calculation effort only the linear analysis has been performed. Such an approach is in line with good design practice, where the serviceability limit state is reached much faster than the ultimate limit state.

Experimental investigation

To assure created numerical model accuracy a several test series for the small scale specimens have been performed. Specimens have been tested in compression and bending in ZWICK Z100 testing equipment (Figure 2). All specimens for bending set-up have 4 mm thick HDF skins. More detailed specification of tested specimens is given in Table 1. It has been assumed that profiled board thickness for all specimens is kept constant 25 mm.

Specimens have been loaded until failure in quasi static compression with the test speed of 1 mm/min. Displacements have been measured by the machine crosshead travel. Mechanical properties of DendroLight® largely depend on wood cell direction; therefore specimens with different orientations have been evaluated (Directions are shown in Figure 1.b). For compression specimens mechanical properties are different in two directions namely: 'DL' and 'Block'. 'Block' direction for compressed specimen and 'DL' direction for bending sandwich specimen is displayed in Figure 2. For bending specimens are possible three types of core orientation affecting mechanical properties. Those similar to compression specimens and additionally 'DL_P' which is 'DL' direction turned perpendicularly to bending the specimen longitudinal direction.

Table 1 Specification of tested specimens

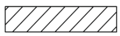
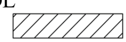
Notation	Dimensions			Number of specimens	Test mode	Structure orientation
	Length, *L* [mm]	Width, *B* [mm]	Height, *H* [mm]			
C1	100	40	100	4	Compression	DL 
C2	100	40	100	3	Compression	Block 
B1	200	50	40	2	4-point bending	DL 
B2	300	50	60	3	4-point bending	DL_P 
B3	350	50	30	3	3-point bending	DL 

Fig.2. 3- point bending and compression (on the right) test set-up on ZWICK Z100

RESULTS AND DISCUSSIONS

In order to ease the comparison of experimental and numerical results a load deformation curve plot has been elaborated. One may see that the experimental results for compression specimens are shown in Figure 3. An average Modulus of elasticity for *C1* series specimens is near 50 MPa, for *C2* series specimens 110 MPa (calculated using EN789 (1995) methodology for reduced specimen size). Specimens have clear elastic mechanical behaviour zone until 80 % of critical load. It makes reasonable use of linear numerical model. Numerical results are in good agreement with *C1* series specimens, however for *C2* type specimens numerical results shows more than 40 % less stiffness. It leads to assumption that wood mechanical properties in radial direction are weaker than applied in numerical model. Long plasticity region for *C2* specimens appears due to slow buckling and crushing of profiled board walls. Comparing to *C1* series specimens there is no rapid losing of load carrying capacity.

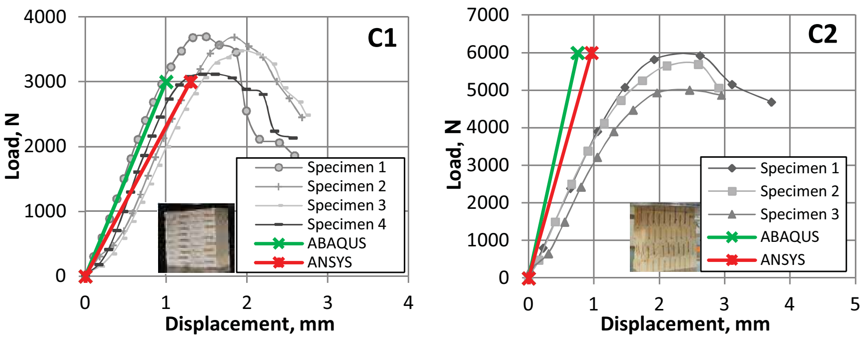


Fig.3. Experimentally obtained displacement values compared with numerical results for compression specimens *C1* and *C2*

Mechanical behaviour of bending specimens mainly depends on properties of outer skins. As in previous plots all specimens have clearly visible elastic behaviour region. Except for *B2* specimen's elastic region is only half of the critical load due of appearance of shear deformations when bond between profiled boards was lost near the loading point. Numerical results in linear mechanical behaviour regions are close to experimental deflection values. In general ABAQUS numerical model has higher stiffness than model from shell elements.

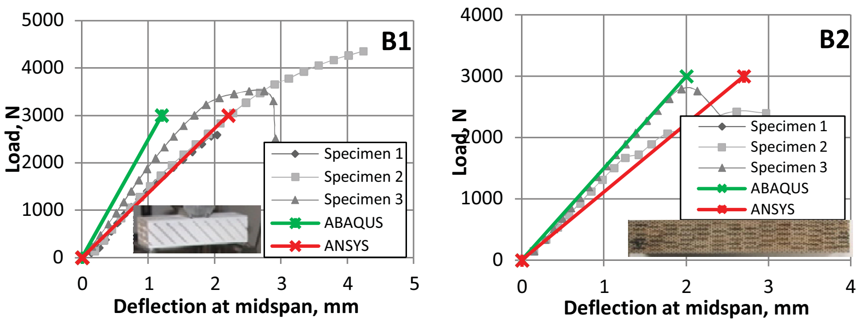


Fig.4. Experimentally obtained deflection values compared with numerical results for bending specimens *B1* and *B2*

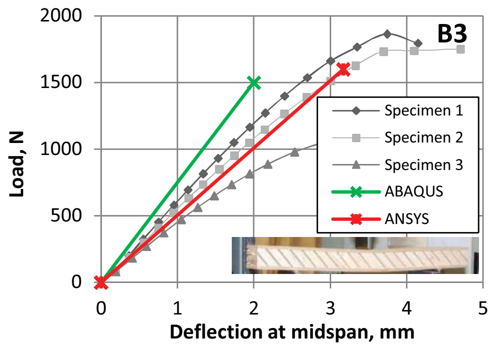


Fig.5. Experimentally obtained deflection values compared with numerical results for bending specimens *B3*

Comparing calculation times on a single PC with two core processor it is obvious that model from shell elements (made in ANSYS code) has shorter analysis time comparing to ABAQUS model made with solids. For *B2* specimen in case of shell model – 20 seconds; for solids model - 140 seconds. Calculation time could be especially important for further optimisation tasks of sandwich panels topology, where several hundreds of experimental runs are required on the full scale structure. Model in ANSYS is made fully parametrical therefore it is more computational time efficient for creating of model variations. In case of B3 specimen it takes nearly 280 seconds for analysis task. Solid geometry for ABAQUS was made employing SOLIDWORKS CAD software and then imported for analysis. Both codes have a good compatibility because are made by the same developer company. However to make parametrical ABAQUS model, programmable features in Python language should be elaborated instead of SOLIDWORKS 3D drawings. Another challenge for numerical modelling is appearance of the small mesh elements during geometry forming operations. They could dramatically decrease calculation speed or in worst case scenario to cause crush of the calculation process entirely.

CONCLUSIONS

It has been demonstrated that by employing numerical modelling it is possible to predict mechanical properties of DendroLight® sandwich panels with sufficient accuracy in elastic deformation range. For experimentally tested specimens is the margin is up to 80% of critical load. The obtained differences between numerical and experimental results usually do not exceed 20 %. In most validation cases numerical models made by solid elements in ABAQUS code demonstrated higher stiffness than model of shell elements made by ANSYS code. Nevertheless a shorter calculation time is possible to reach by creating the numerical model from shell type elements.

Taking into the account relatively long calculation time for creating and analysis of detailed DendroLight® structure, such an analysis is suggested only for small scale structural elements with dimensions not exceeding one meter of length. For large scale structures mixed approach should be utilised instead when mechanical properties are extracted from small scale numerical model and assigned to continuum layer of large structure as equivalent stiffness properties.

Further investigation is required to improve numerical model`s accuracy also including adhesive layer and structural imperfections as well to decrease the calculation time and to improve the model calculation stability.

REFERENCES

Perrson, K. 2008. Micromechanical modelling of wood and fibre properties. Doctoral Thesis, Lund University, Sweden

Nishiyama, N & Ando, N. 2003. Analysis of load-slip characteristics of nailed wood joints: application of a two-dimensional geometric nonlinear analysis. Journal of Wood Sciience. 49(6):505–512

Resch, E & Kaliske, M. 2010. Three-dimensional numerical analyses of load-bearing behavior and failure of multiple double-shear dowel-type connections in timber engineering. Computers and Structures. 88(3-4):165-77.

Zhou, Q., Yan, W., Zhou, X. Ji J. 2010. Vibration characters and seismic response of chinese ancient buildings. Beijing Gongye Daxue Xuebao / Journal of Beijing University of Technology. 36(1):13-7.

Mackerle, J. 2005. Finite element analyses in wood research: a bibliography. Wood Science and Technology. 39: 579–600

ANSYS Version 11. 2009. User Manual, Papenburg, USA

ABAQUS/Standard User's Manual. 2009. Version 6.9.1 ABAQUS. Inc. 166 Valley Street Providence. RI 02909. USA

http://www.dendrolightlatvija.lv [30.07.2012]

Iejavs, J., Spulle, J., Jakovļevs, V. 2011. The comparison of properties of three-layer cellular material and wood-based panels", Drewno, 139-48

EN 338:2003. (2003) Structural timber. Strength classes". European Committee for Standardization

1999. Wood handbook Wood as engineering material, United States Department of Agriculture

EN 789 (1995) Timber structures – Test methods - Determination of mechanical properties of wood based panel. European Committee for Standard.