22nd International Conference
ENGINEERING MECHANICS 2016
Svratka, Czech Republic, 9 – 12 May 2016

ANALYSIS OF TRADITIONAL CARPENTRY BUTT JOINT FINITE-ELEMENT MESH

M. Hataj[*], P. Kuklík[**]

Abstract: *The aim of this article is to compare a quality of butt joint 3D finite-element meshes. FEM solution accuracy and calculation convergence speed are the main factors for a mesh quality evaluation. Numerical model of the subjected butt joint includes material nonlinearities. Material model of wood presume elasto-plastic behaviour and has orthotropic – transversal isotropic property. Contact elements are modeled among the individual structural components of the joint. The finite-element meshes introduced herein differ one from other by element type and their quantity applied, type of mapping and local density of mesh. Number of nodes and elements, calculation convergence speed, FEM solution exactness, symmetry and mapping of elements are observed.*

Keywords: Finite element method, Mesh, Butt joint, Carpentry, Timber structures.

1. Introduction

Wood is one of the first structural material applied in the civil engineering practice. It disposes of beneficial structural properties which make it, together with its renewability, convenient for primary load-bearing element application. Timber elements often constitute bearing part of roof structures. A connection is usually the weakest point of a timber frame structure. Traditional carpentry joints are still frequently performed despite the great technological progress in timber joining. However, these types of connections are not supported by the applicable standards much and therefore their design normally considers only simple and empirical relationships based on a carpenter's experience. The aim of the carpentry joints research is to derive analytical relationships supported by modern numerical calculations and experiments and so enable their effective application in a structural practice. For instance, a lapped scarf joint with inclined faces and wooden dowels starts was applied within a historical structure reconstruction process, Arciszewska-Kedzior et al. (2015).

This article is focused on a perpendicular butt joint examination. The connection is composed of two structural elements – one is longitudinal and one is transversal, see Fig. 1. A traditional butt joint is usually fixed by a carpentry iron dog. The transversal timber element is exposed to compression parallel to the grains. Longitudinal element is subjected to compression perpendicular to the grains.

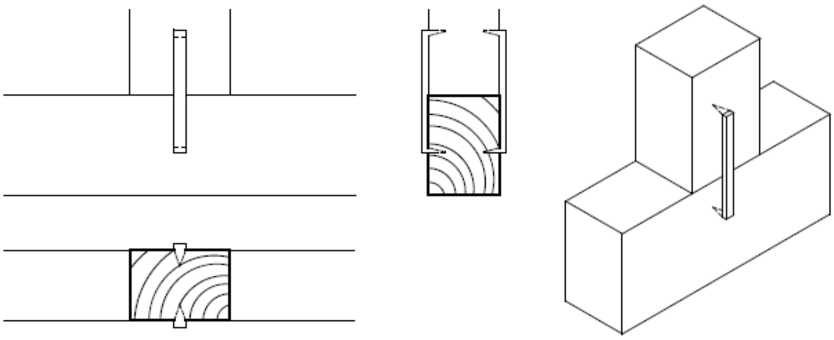


Fig. 1: Traditional carpentry butt joint.

[*] Ing. Martin Hataj: Department of Steel and Timber Structures, Czech Technical University in Prague, Thákurova 7; 166 29, Prague; CZ, martin.hataj@fsv.cvut.cz

[**] Assoc. Prof. Ing. Petr Kuklík, Ph.D.: University Centre for Energy Efficient Buildings CTU, Czech Technical University in Prague, Třinecká 1024; 273 43, Buštěhrad; CZ, kuklik@uceeb.cz

Compression strength perpendicular to the grain flow is noticeably lower comparing to compression strength parallel to the grain flow. Characteristic values of compression strength for a commonly applied timber strength class in the Czech Republic - C24 - are listed in standard ČSN EN 338 (2003):

$$f_{\mathrm{c,0,k}} = 21\,\mathrm{MPa} \tag{1}$$

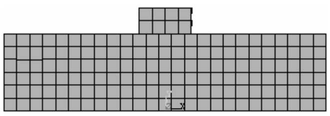
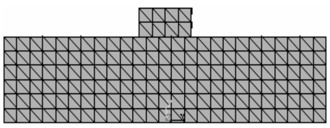
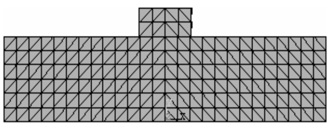
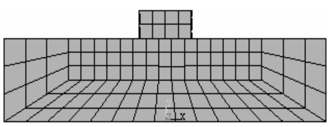
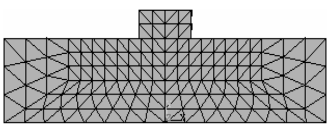
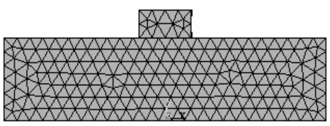
$$f_{\mathrm{c,90,k}} = 2{,}5\,\mathrm{MPa} \tag{2}$$

where $f_{\mathrm{c,0,k}}$ is characteristic value of compression strength parallel to the grains and $f_{\mathrm{c,90,k}}$ is characteristic value of compression strength perpendicular to the grains. Regarding the fact, that $f_{\mathrm{c,0,k}}$ is approx. ten times higher that $f_{\mathrm{c,0,d}}$, the strength perpendicular to grains is often exceeded in linear elements and undesired local deformations occur.

2. Methods

This research is focused on 21 numerical models of a traditional carpentry butt joint. All calculations are conducted on a mutual desktop computer. Hardware parameters are consisted from CPU Intel Xeon E5-1650, 6 cores 3,2 GHz, RAM 16 GB. Numerical models are created in software ANSYS 16.0, Academic. All the inputs, except for an appertaining finite-element mesh, are identical in all the numerical models examined within the research.

Tab. 1: Finite-element mesh types.

Mesh type	Model No.	El. size [mm]	Element type	Elem. number	Node number	Time [h:m:s]
	1_1_1	20	SOLID45 hex	873	1115	0:00:15
	1_1_2	10	SOLID45 hex	5940	6741	0:01:46
	1_1_3	5	SOLID45 hex	43728	46631	0:22:01
	1_1_4	20	SOLID95 hex	873	3897	0:00:48
	1_1_5	10	SOLID95 hex	5940	25063	0:08:11
	1_1_6	5	SOLID95 hex	43728	179531	5:58:24
	1_2_1	20	SOLID95 pent	1921	4737	0:00:49
	1_2_2	10	SOLID95 pent	12596	30887	0:09:03
	1_2_3	5	SOLID95 pent	90320	222795	5:03:21
	1_3_1	20	SOLID95 pent	1994	4911	0:00:48
	1_3_2	10	SOLID95 pent	12596	30887	0:08:13
	1_3_3	5	SOLID95 pent	90320	222795	5:18:57
	1_4_1	20	SOLID95 hex	767	2903	0:00:29
	1_4_2	10	SOLID95 hex	4382	16413	0:04:25
	1_4_3	5	SOLID95 hex	30536	117255	2:14:44
	1_5_1	20	SOLID95 pent	1456	3693	0:00:37
	1_5_2	10	SOLID95 pent	8176	20207	0:04:57
	1_5_3	5	SOLID95 pent	58720	145439	1:53:31
	1_6_1	20	SOLID92 tetr	5371	8434	0:00:21 [1]
	1_6_2	10	SOLID92 tetr	40960	59123	0:06:20 [2]
	1_6_3	5	SOLID92 tetr	326167	451800	18:39:58

Numerical models of the traditional carpentry butt joint include orthotropic elasto-plactic wood definition. This material model expects a bilinear stress-strain relation, published by Moses & Prion (2002). Different types of finite-element meshes applied in the particular simulations can be found in Tab.1. Individual meshes differ one from other by types of element and their quantity, by type of mapping and local density of the mesh. Computing time for individual models is listed in the right column of Tab.1. Models 1_6_1 and 1_6_2 are not convergent and so time for 30 % [1] a 42 % [2] of the final computing time is listed in the table. Elements SOLID45 and SOLID95 are used for mapped finite-element mesh whereas element SOLID95 forms free mesh. Applied elements are demonstrated in Fig. 2.

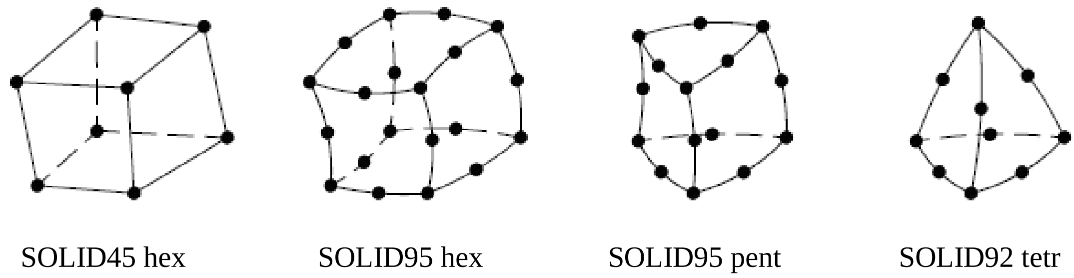


Fig. 2: Used element types.

A solution quality and calculation convergence speed with finite-element meshes listed above are compared with each other. Load-displacement behaviour of joint is depicted in Fig. 3 respectively. Loading is applied on the horizontal surface of the transversal timber element. Elasto-plastic behaviour of numerical model sets 1 – 6 is marked with a grey line. The most exact numerical solution that is reached by the procedure is marked with the black dashed line.

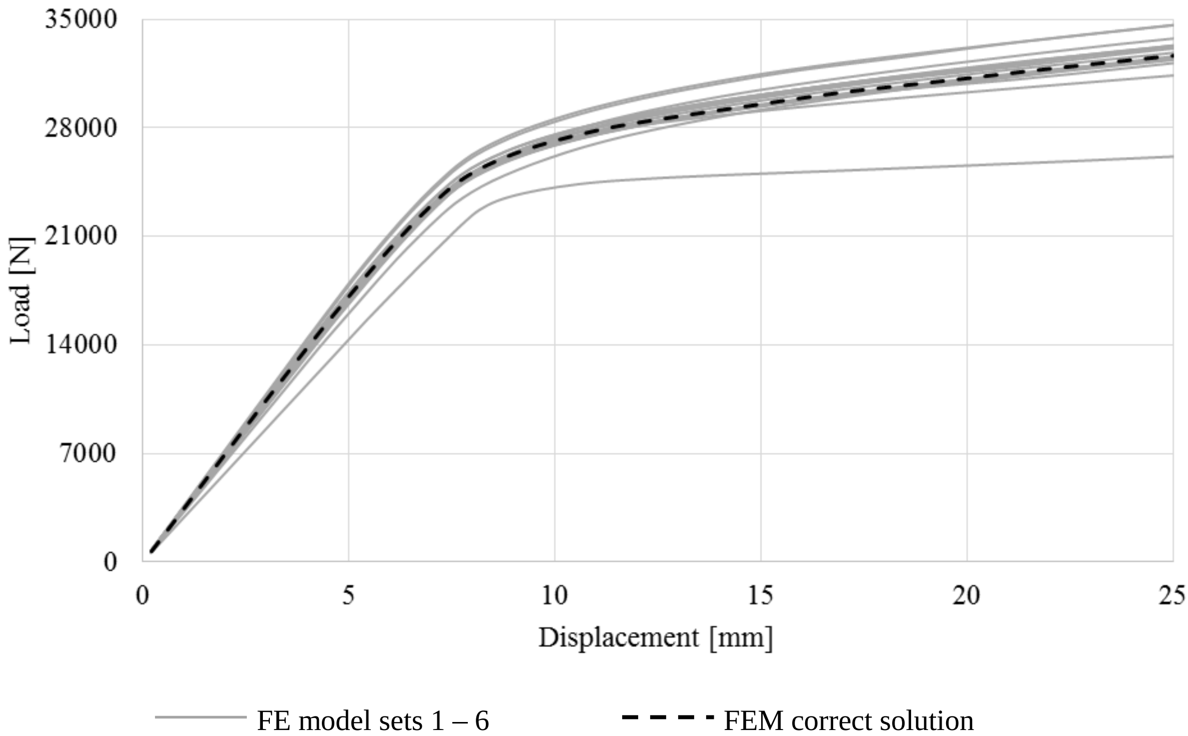


Fig. 3: Load-displacement diagram of FE model sets 1-6 and correct solution.

In Tab. 2, percentage equality of the individual finite-element types' solution with the most exact numerical result is presented. The value of compression stress linked to the vertical displacement equal to 25 mm is the dominant result assessed within the numerical simulation analyses. Finite-element models 1_6_1 and 1_6_2 are not evaluated due to non-convergence of numerical calculation.

Tab. 2: Finite-element mesh quality.

Mesh number	1_1_1	1_1_4	1_2_1	1_3_1	1_4_1	1_5_1	1_6_1
Quality [%]	75	106	102	103	106	101	-
Mesh number	1_1_2	1_1_5	1_2_2	1_3_2	1_4_2	1_5_2	1_6_2
Quality [%]	93	101	101	101	101	100	-
Mesh number	1_1_3	1_1_6	1_2_3	1_3_3	1_4_3	1_5_3	1_6_3
Quality [%]	98	100	100	100	100	100	100

3. Conclusions

In contrast with application of SOLID95 hex elements, an application of SOLID45 hex elements leads to an accurate result with increasing number of elements. Symmetric element configuration does not significantly affect a solution quality and calculation convergence speed in comparison with asymmetrical configuration of the same elements (SOLID95 pent). Application of SOLID92 elements resulted into a poor calculation convergence. Calculation converges in the case the number of elements is high however computation is rather time-consuming. Application of SOLID95 pent elements proves to be more advantageous than SOLID45 hex elements. Numerical model with SOLID95 pent elements delivers a better result accuracy comparing to the model with twice smaller elements SOLID45 hex. Furthermore, calculation is approximately twice faster in favour of finite-element mesh with SOLID95 pent. SOLID95 pent element application is also useful in comparison with elements SOLID 95 hex. SOLID 95 pent elements collect results in two other nodes. Both these simulations are almost the same time-consuming, although SOLID95 pent elements are twice longer than SOLID95 hex elements in compared finite-element meshes and calculation results are more precise. Parts of volume that are discrete distributed by the described elements are mutually compared and depicted in Fig. 4. Numerical models with a mesh locally densified in the interface (connecting area), where the timber elements are connected and where prismatic elements SOLID95 pent are applied, appear to be the most favourable in the aspects of results quality and calculation speed.

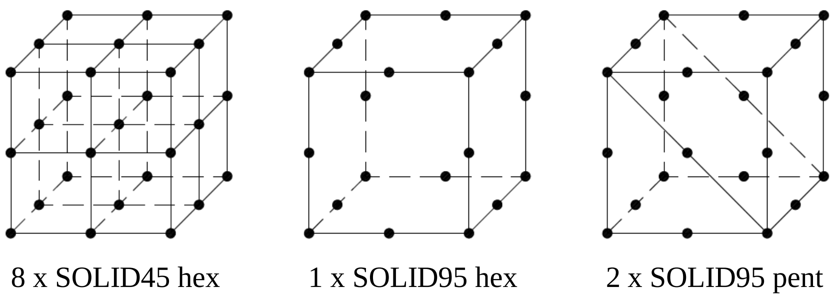


Fig. 4: Applied element types for comparison.

Acknowledgement

This work has been supported by project of University Centre for Energy Efficient Buildigs and by Czech Technical University in Prague project no. SGS14/178/OHK1/3T/11.

References

Arciszewska-Kedzior, A., Kunecký, J., Hasníková, H. & Sebera, V. (2015) Lapped scarf joint with inclined faces and wooden dowels: experimental and numerical analysis. Engineering Structures, 94:2015, pp.1-8.

ČSN EN 338: Konstrukční dřevo – Třídy pevnosti. (2003) Český normalizační institut, Prague.

Moses, D. M. & Prion, G. L. (2002) Anisotropic Plasticity and Failure Prediction in Wood Composites. Research Report, University Of British Columbia, Vancouver.