ACTA UNIVERSITATIS AGRICULTURAE ET SILVICULTURAE MENDELIANAE BRUNENSIS

Volume LVIII 37 Number 5, 2010

FINITE ELEMENT ANALYSIS OF DOVETAIL JOINT MADE WITH THE USE OF CNC TECHNOLOGY

V. Sebera, M. Šimek

Received: March 29, 2010

Abstract

SEBERA, V. ŠIMEK, M.: *Finite element analysis of dovetail joint made with the use of CNC technology.* Acta univ. agric. et silvic. Mendel. Brun., 2010, LVIII, No. 5, pp. 321–328

The objective of the paper is the parametrization and the finite element analysis of mechanical properties of a through dovetail joint made with the use of a specific procedure by a 3-axis CNC machine. This corner joint was used for the simulation of the bending load of the joint in the angle plane – by compression, i.e. by pressing the joint together. The deformation fields, the stress distribution, the stiffness and the bending moment of the joints were evaluated. The finite element system ANSYS was used to create two parametric numerical models of the joint. The first one represents an ideally stiff joint – both joint parts have been glued together. The second model includes the contact between the joined parts. This numerical model was used to monitor the response of the joint stiffness to the change of the static friction coefficient. The results of both models were compared both with each other and with similar analyses conducted within the research into ready-to-assemble furniture joints. The results can be employed in the designing of more complex furniture products with higher demands concerning stiffness characteristics, such as furniture for sitting. However, this assumption depends on the correction of the created parametric models by experimental testing.

furniture, dovetail joint, numerical simulation, mechanical properties, CNC technology

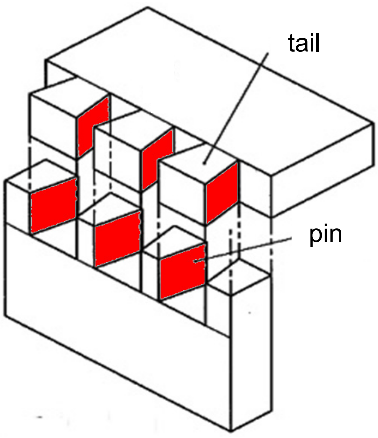
The through dovetail joint is a classical furniture joint with outstanding strength properties. The joint consists of tails and pins, see fig. 1. The specific construction of the joint, which ensures its self-locking character and consequently considerable strength, requires a demanding technology. This joint is usually made manually (individual production) or by special machines and devices (batch production). The technology of current 3-axis CNC machines makes the manufacture of dovetail joints fast and accurate. However, it also bears some specifics. One of them is the inability of the technology to make sharp inner edges of the milled profile from angle 0° to 180° because of the rotating movement of the cutting tools. This limitation can be removed by boring a hole in place of the demanded edge which results in what is called "the Mickey Mouse ears", see fig. 2c. CNC manufacturing technologies have a big potential in the fields of furniture production and wood processing, for example in competition with cheap labour. A new manufacturing technology of CNC machines and new composite materials are prerequisites for successful usage of this joint in the current environment of wood and furniture industry (Susnjara, 2006).

Testing of furniture joints has been dealt with many times, both by means of experiments and numerical simulations. Mihailescu (2003) analysed various forms of parametric models of joints using the finite element method (FEM). He simulated loading of a glued joint on the tenon and the mortise made from beech and oak wood by a bending moment in the angle plane in tension. The result of his research was the analysis of the influence of individual joint parameters on the deformation and stress.

Smardzewski and Prekrad (2002) examined the stress distribution in ready-to-assemble furniture corner joints using experimental testing and numerical simulation. The authors tested three types of ready-to-assemble joinery in combination with beech pins and particleboard by bending in the angle plane in compression. The results show that besides mechanical connectors the pins play an important role in the joint as they support its strength. The numerical simulation of the examined joints clarified how specific connecting elements transmit stress in the joint.

Joščák (1999) used experimental testing to explore mechanical properties of several typical furniture corner joints used for joining composite wood materials. By loading selected joints he found the maximum values of the bending moment and stiffness. Furthermore, Černok, Joščák and Lang (2004) analysed the stress distribution, the stiffness and the deformation of a glued beech pin joint using the finite element method.

Zhang and Eckelman (1993) tested the resistance to the bending moment of single-dowel and multi-dowel (glued) furniture corner joints in particleboard. These authors studied the dependence of load capacity in various dimensional modifications of specimens, pins and pin distribution in the joined material. Similarly, Veselovský (1996) and Šimek *et al.* (2010) dealt with the capacity of corner joints within the research into the methods of testing and designing joints.



1: *A through dovetail joint manufactured using classical technology, friction surfaces highlighted in red (Nutsch, 2003)*

MATERIAL AND METHODS

Before the manufacture itself a parametric joint is analysed using numerical simulation of mechanical loading, which makes the subsequent geometry and material optimization possible. Thus we obtain the first strength characteristics of the joint and the distribution of stress which allows us to propose modifications. The manufactured joints are analysed by experimental mechanical testing. The objective factors for evaluation will be the strength (capacity), the stiffness (the ratio of deformation to the bending moment), and the deflection of the joint when loaded by bending (with this type of load furniture joints achieve the lowest values of capacity).

The aim of the paper is the parametrization of a numerical model and a finite element analysis of mechanical properties of dovetail joints manufactured from 12 mm thick birch plywood boards. Two parametric numerical models were created, both loaded by a bending moment. These are intended to serve as tools for designing furniture with the joints in question. The models were solved in the form of a structural analysis; in the first model, both parts (tail and pin) were designed as a glued joint with all degrees of freedom constrained; in the second model a contact was defined between both parts (see below). Both models are supposed to be compared with, or corrected by, experimental testing.

The contact analysis of the dovetail joint includes another derived material characteristic – the static friction coefficient μ, as follows from this equation:

$$F_t = \mu \cdot F_N. \qquad [1]$$

The friction coefficient represents the ratio of the friction force F_t to the perpendicular compressive force F_N affecting the object. In common material tables (Kotlík *et al.*, 2003) or textbooks on mechanics (Meriam, 1978) the wood-wood friction coefficient ranges within 0.15–0.6, or 0.15–0.4 for the dynamic friction coefficient. We based our research on the study conducted by Bejo, Lang and Fodor (2000), who measured the coefficients of friction (static and dynamic) for wood composites LVL (Laminated Veneer Lumber) and LSL (Laminated Strand Lumber). As these wood composites are structurally very similar to plywood, their resulting data are applicable to our numerical model. Their results show that the static friction coefficient in LVL ranges between 0.33 and 0.7 and it is mainly dependent on the contact compression and the direction of fibres both in the specimen and the contact surface. The dynamic coefficient is not taken into account in this paper, see discussion.

The methodology for the dovetail joint testing by a bending moment is based on literature – Joščák (1999). Within the framework of this research, the mechanical load was limited to bending in the angle plane – by compression, see fig. 2 d), as that is the most frequent and the most critical way of loading furniture corner joints. Boundary and geometrical conditions for joint testing following from fig. 2 d) served for the creation of the numerical model and its evaluation.

NUMERICAL MODEL

For the purpose of our research two parametric numerical models were created. The first model represents a dovetail joint where the contact between the tail and the pin is not defined. It is an ideally stiff joint (as if glued). The second model is a joint where the contact between the tail and the pin parts is defined, i.e. this is a contact analysis. Because there is the plane of symmetry and to save computing time, the models are solved as symmetrical, see fig. 2 a). Boundary conditions, i.e. fixing and loading, are defined so that they correspond to the real conditions of a testing machine while measuring joint stiffness and strength, see fig. 2 d). The degrees of freedom UX, UY, UZ are constrained. A 2 mm displacement *d* is applied to the upper edge and at the same time all the nodes of this edge are constrained with the degrees of freedom UX and UY. A boundary condition of symmetry is applied to the side in the XZ plane, which intersects the dovetail joint in the middle. Both models are created using the Ansys Parametric Design Language (APDL). They can be easily modified and thus the parameters can be changed according to the setting of the boundary conditions or the material model.

The finite element mesh (FE mesh) of parametric models consists of quadratic tetrahedral elements SOLID92. To ensure efficient approximation of geometry and a true picture of higher stress gradients, the FE mesh has been refined in the places around the bored holes and pins, see fig. 2 b) and 2 c). The FE mesh of the contact elements consists of TARGE170 and CONTA174 elements, with the contact pair being solved as asymmetric. As a part of the evaluation of results and with respect to their further comparison with experiments, the universal macro was written in the APDL language; the macro will evaluate the reaction forces at the bottom edge of the joint. The reaction forces are then multiplied by the joint arm so that we could calculate the bending moment of the joint in the angle plane according to:

$$R_{sum} = \sum_{i=1}^{m}\sum_{k=1}^{n}(R_{FX}^{n} + R_{FY}^{n} + R_{FZ}^{n})m \text{ and } M = R_{sum} \cdot v_c, \qquad [2]$$

where

R_{FX}, R_{FY}, R_{FZ} are the reaction forces in the individual directions at the constrained nodes [N]

M .. the bending moment affecting the joint [Nm]

n .... the number of constrained nodes of the bottom edge

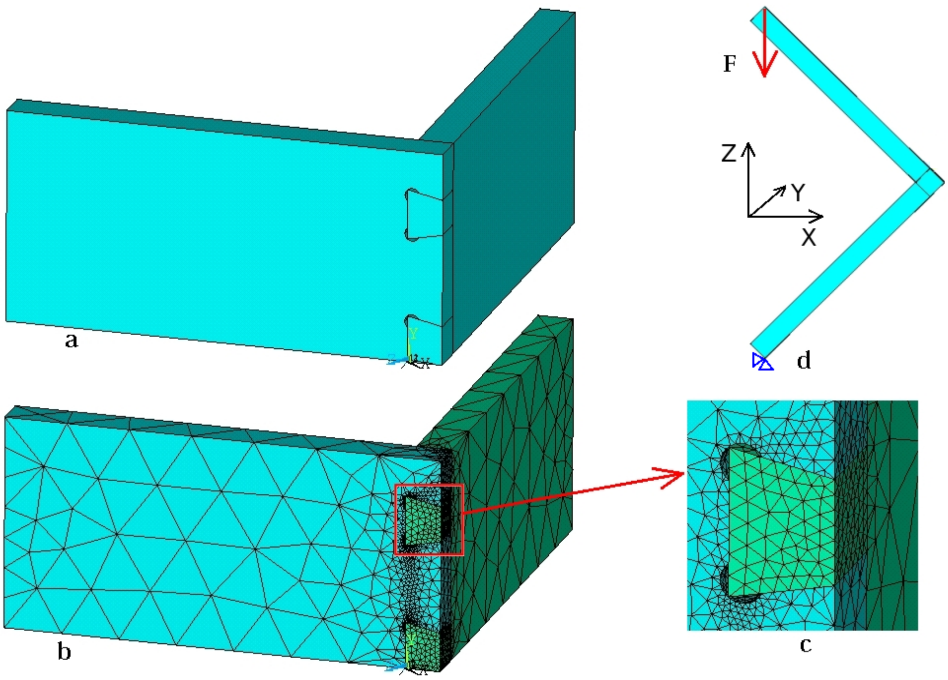
m ... the number of steps of the analysis

v_c.... the moment arm [m].

The sum of the reaction forces is calculated using the individual nodes within each step of the analysis, which gives us the total reaction forces to the loading. The geometrical conditions are updated after each partial step until the final displacement of 2 mm is achieved. The bending moment of the conducted analyses with various friction coefficients was then transported into a graph for comparison. Experimental verification of numerical models and the results of analyses are important aspects of each numerical analysis. In our case, the numerical models precede the experiments which will be made subsequently using the above mentioned methodology. The used material properties for the numerical model were found in literature – Lang (2002), Highett (1987) and Wood Handbook, see Table I.

SOLUTION AND RESULTS

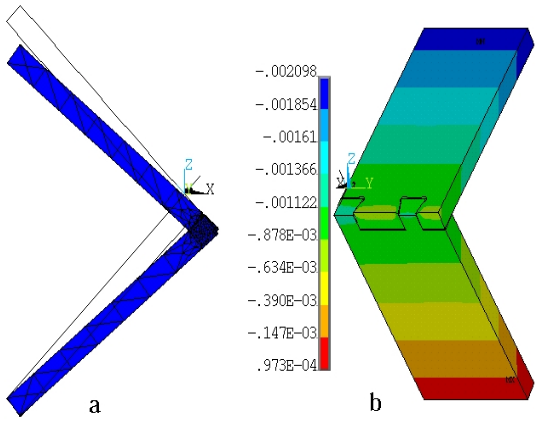
The total deformation and displacement of the joint in the direction of the Z axis (i.e. in the direction of the force) is presented in fig. 3. These pictures enable us to evaluate whether the model had been fixed in the right way and whether the demanded deformation which would occur in the testing machine during experimental testing occurred here.



2: *a) The geometrical model of the joint, b) the FE model, c) the detail of the joint in the FE model, d) loading of the joint in the angle plane by compression*

I: *The used material properties [spruce (Picea sp.), poplar (Populus sp.), ash (Fraxinus sp.), oak (Quercus sp.), plywood]*

	Material properties								
Material	E_x [GPa]	E_y [GPa]	E_z [GPa]	G_{xy} [GPa]	G_{yz} [GPa]	G_{xz} [GPa]	ν_{xy}[-]	ν_{yz} [-]	ν_{xz} [-]
plywood	6.97	7.25	0.65	0.545	0.397	0.283	0.4	0.32	0.31
spruce	0.68	14.3	0.47	1.23	0.8	0.055	0.03	0.41	0.38
oak	2.046	11.778	1.029	0.484	0.285	0.055	0.014	0.452	0.328
ash	1.875	15.7	0.47	1.23	0.8	0.056	0.056	0.566	0.467
poplar	0.885	13.9	0.35	0.84	0.385	0.11	0.037	0.59	0.356



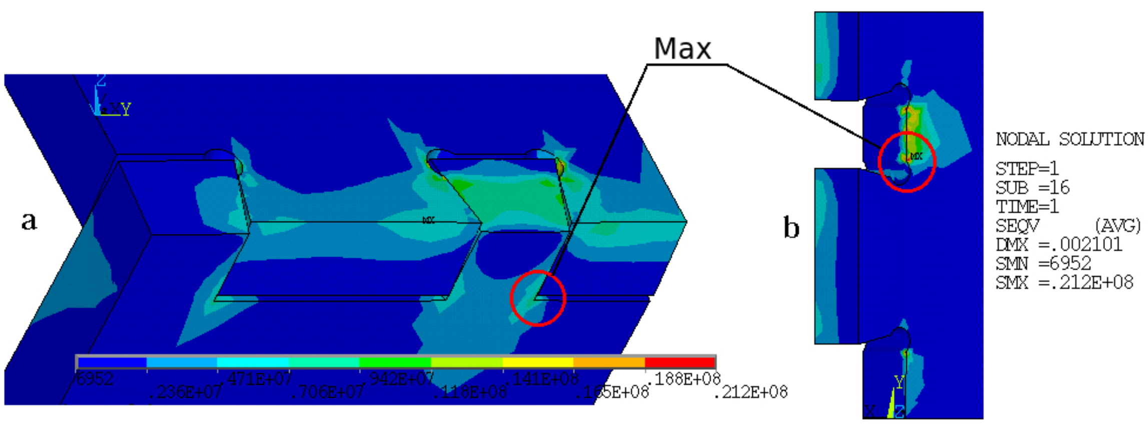
3: *a) The total deformation of the joint (multiplied by 10), b) the displacement in the direction of the force – Z axis*

Fig. 3 b) shows that the maximum displacement of the joint in the direction of the Z axis is approximately 2 mm (the blue strip at the top). This value had been set as a boundary condition for the joint loading. Fig. 4 presents the distribution of von Mises stress in the contact model. Considering the geometrical conditions, the stress distribution and other indicators (contact penetration, contact status during analysis, etc.), we can assume that the contact pair corresponds to loading in a real environment.

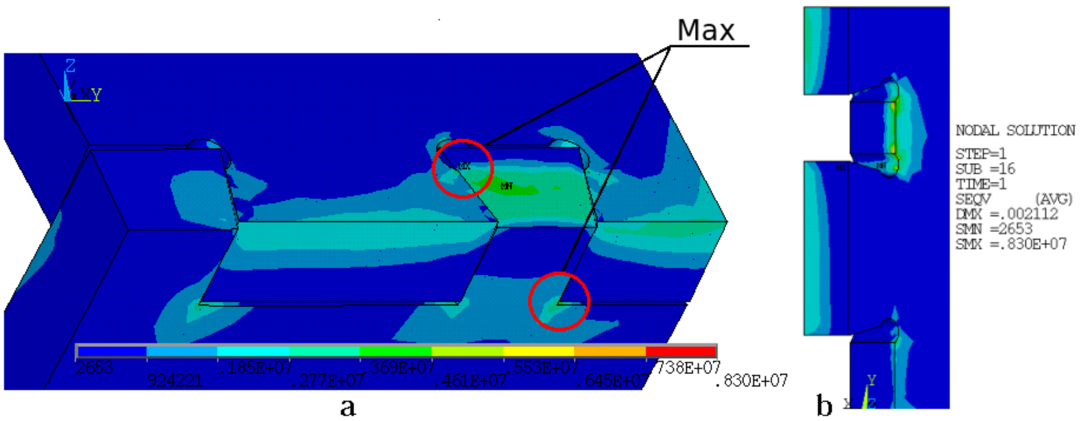
The maximum and the high values of stress expectedly appear in the corner on the base of the tail and in the area of the bored hole in the part with the pin (these values are circled in the picture).

Fig. 4 also shows what distribution of stress the joint model produces when loaded. Thanks to the considered contact, it is obvious that there is a higher stress also in the places of friction of both parts – on the sloping areas of the joint (cannot be seen in the pictures). The stress in these places will be the higher, the higher is the joint deformation in the angle plane and the higher is the friction coefficient. Sloping areas of the joint are what determines its stiffness and resistance to deformation. This confirms the well-known fact about the joint that with the increasing angle of the sloping area the stiffness of the dovetail joint increases, especially in the initial stage of loading. However, stiffness can be raised by increasing the angle only to a certain point which is dependent upon the total strength of the joint because by increasing the angle, the geometrical characteristics change considerably and its manufacturability is limited. A bigger angle increases the force necessary for the friction in the pin but at the same time the joint gets weaker and its strength is lowered with respect to the loading in question. Empirical experience and our previous results (Nutsch, 2003; Sebera, Šimek, 2008) show that the optimal angle of pin ranges from 10 to 14 degrees.

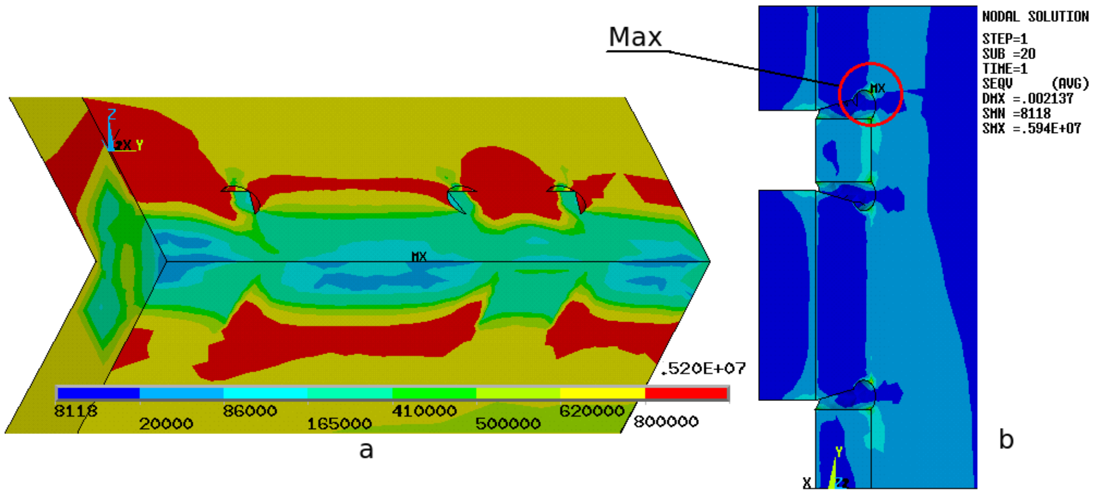
Fig. 5 illustrates a situation similar to fig. 4, the only difference being in the material model; the model has the properties of spruce (Picea species). It is obvious that the stress distribution only differs in



4: *a) von Mises stress (MPa) in the contact model of the joint made from plywood, b) detail of the distribution of stress (MPa) in the tail*



5: *a) von Mises stress (MPa) in the contact model of the joint made from spruce, b) detail of the distribution of stress (MPa) in the tail*



6: *von Mises stress (MPa) in the non-contact model of the joint made from plywood, a) the entire joint (non-uniform scale), b) detail of the tail (uniform scale)*

the magnitude of stress. We do not consider it necessary to present the distribution of stress in the joint with other material models as these are highly similar. The values of reaction forces of analyses in other material models are summarized in Table II.

The non-contact model has a different stress distribution, see fig. 6. The maximum stress appears on the edge of the bored hole on the basis of the tail, which is the same as in the contact model. However, we can notice that an increased stress also appears on the inner side of the joint, see fig. 6 b). For better illustration of von Mises stress distribution contours, the entire non-contact model uses a non-uniform scale instead of a uniform scale of contours, see fig. 6 a). The stress distribution shows that this is a different stress field than in the contact model, in which the joint does not behave as if made from one piece. Therefore, the stress can be divided into the compression stress, inside the joint, and the tension stress on the external sides of the joint. Further, we can see that although the joint is defined as glued (stiff), the bored holes are still the places where stress concentrates.

It has to be mentioned that in the model there are 'singular' places where the stress due to sharp edges gains higher values than it should. Taking this into consideration, the maximum stress values obtained in our model can contain numerically conditioned inaccuracies. In spite of this, the models are able to provide us with highly usable results for the establishment of the deformation, the evaluation of the stress distribution, the finding of weak spots, or the evaluation of stiffness. Again, similar situation appeared in the other material models.

Furthermore, we can state that the bored holes on the basis of the tail positively distribute stress and thus reduce the probability of a crack initiation. This is the same effect as when we want to reduce the probability of a crack initiation in polymers by boring a hole or rounding in place of a sharp edge. The fact that there is a bored hole in the tail part has a negative impact on the joint in the form of a partial weakening of its geometry but, on the other hand, in this place a higher stress is necessary for a crack to appear than if there was a sharp corner.

The evaluation of the joint stiffness according to equation [2] was carried out as the next stage using the above mentioned macro. A graph was drawn expressing the dependence of the bending moment of the joint on the angular deflection, where the slope of the regression equation expresses the actual joint stiffness. Fig. 7 shows the results of contact analyses of the joint with various friction coefficients and also of the glued joint – the non-contact analysis. Regression equations in fig. 7 are ordered in the same way as the regression lines; the regression equation of the non-contact analysis is marked blue.

It follows from fig. 7 that the resulting bending moment, or the stiffness, of the joint in the contact analysis depends on the defined static friction coefficient μ. The higher coefficient μ, the bigger force we need to deform the joint in its angle plane. Considering the self-locking character of a dovetail joint and our results we can assume that a contact model with friction coefficient of 0.4–0.45 reaches the strength characteristics of a glued joint (non-contact analysis). However, this assumption has to be verified by experimental testing.

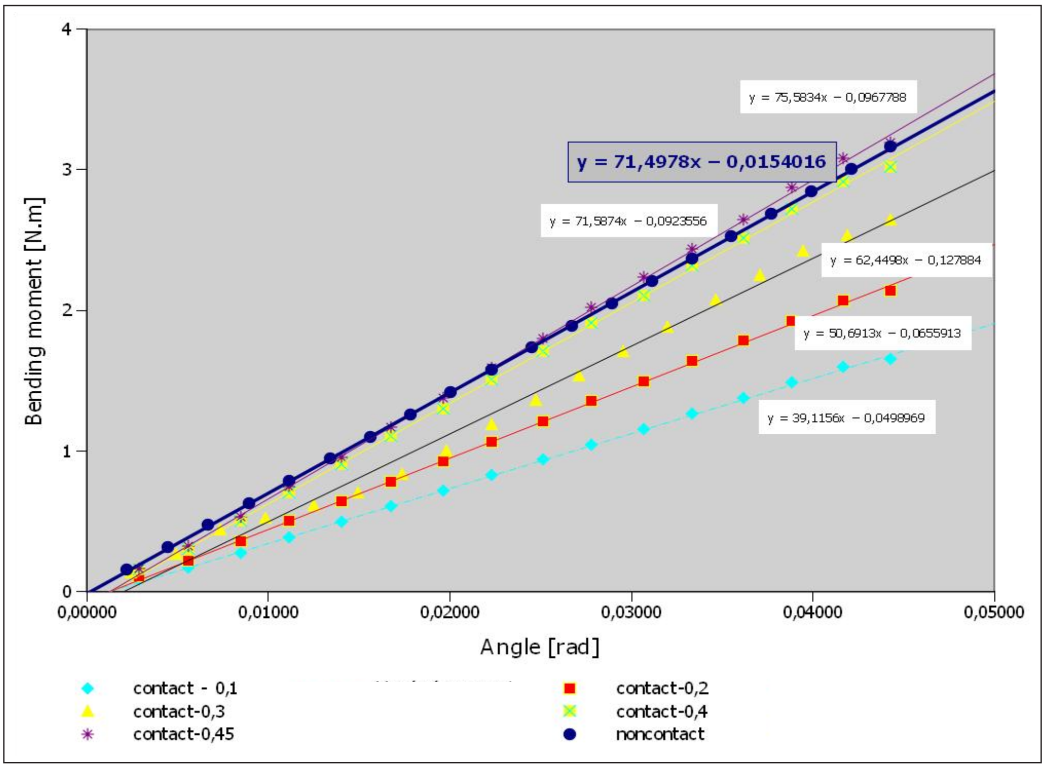
The values of stiffness for individual partial solutions of the joint are in the range of 39–76 $N.m.rad^{-1}$. In comparison with e.g. Šimek (2008), these values are about 5–180 $N.m.rad^{-1}$ lower, i.e. the results are quite different. However, it is necessary to note that it was research into different types of joints made from different materials (metal cam connectors). Still, such a comparison is of value as it provides us with an approximate idea of the stiffness of the selected furniture joints. As has been mentioned before, the verification of our numerical models of dovetail joints requires experimental measuring.

The dependence of the bending moment at 2 mm deformation on the static friction coefficient is presented in fig. 8. It shows us that although the contact analysis is a highly non-linear problem, the dependence of the bending moment on the coefficient μ is directly proportional (within the declaration of the numerical model).

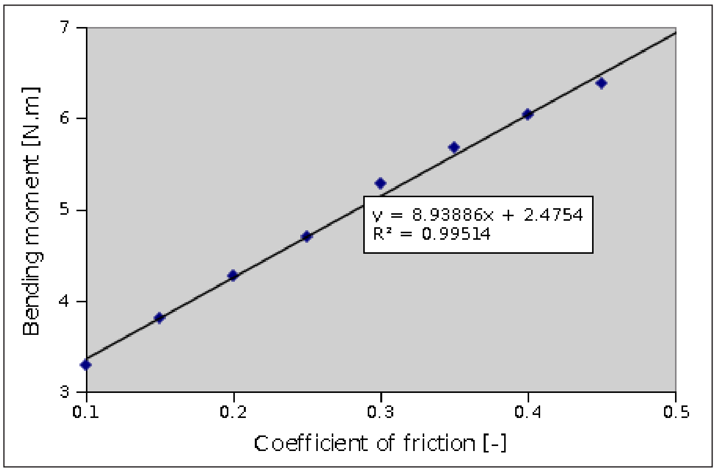
The issue of friction is much more complicated in the real environment as it includes the friction occurring during the movement of an object; for this purpose we would have to include the dynamic friction coefficient. This coefficient, which is usually lower than the static coefficient, has not be considered in our simulations because we are dealing with quasistatic loading in which none of the parts of the joint is put to motion before the analysis, i.e. this situation does not require the dynamic friction coefficient.

To get a notion to what extent the reaction forces of the joint can vary in different material models, we conducted further analyses, see Table II.

Table II reveals that the reaction force (or the joint stiffness) is highly dependent on the material properties. Because of the high variability of properties of wood, table II serves for the purpose of information valid ad hoc. The highest reaction forces are to be found in the joints from plywood, followed by ash and oak.



7: *Graph of dependence of the bending moment and stiffness on the angular deflection (plywood joint)*



8: *The dependence of the bending moment on the friction coefficient*

II: *Maximum reaction forces with various material models*

	Contact – μ = 0.2					Non-contact	
Material	**spruce**	**poplar**	**ash**	**oak**	**plywood**	**plywood**	**spruce**
F [N]	7.3	9.1	12.9	11.8	23.7	35.1	22.3

CONCLUSION

The paper deals with the parametrization and finite element analysis of the furniture dovetail joint. Two numerical models were created for this purpose – a contact model and a non-contact model. Both models are fully parametric and have been written in the APDL language. Moreover, the impact of the static friction coefficient on the joint stiffness was examined within the framework of the contact model. The non-contact model served as a reference model for the contact model and also determined its limits of usability.

The results of the contact analysis clearly show that the bending moment, or the stiffness of the joint, is directly proportional to the friction coefficient, as the regression equation in fig. 8 proves. When the friction coefficient is increased from 0.1 to 0.45, the joint stiffness almost doubles. Literary sources conclude that the friction coefficient in the case of wood and wood based products can be between 0.1 and 0.45. It depends on the orientation of the friction surfaces in relation to the orientation of fibres, as well as on the technological operations the material surface was treated by. Moreover, we can assume on the basis of the results that the stiffness of the contact model will approach the upper limit of the empirical interval of the friction coefficient (0.4–0.45). The results of the finite element analyses can be considered correct at this stage, however, really accurate comparisons have to wait until after subsequent experimental verification. Only then can we say to what extent our numerical models are accurate. The partial analysis of the impact of wood species with their material properties on the stiffness of the joint showed that the highest stiffness is achieved when plywood is used, followed by ash, oak, poplar and spruce. The parametric models can be used for the designing of more complex furniture products with higher demands concerning stiffness characteristics (such as furniture for sitting). However, this assumption is dependent on the correction of the created parametric models by experimental testing.

The development and usage of the FEM tools in the field of design as such, but especially in the industrial design, proves that there are many opportunities for using the FEM analyses both in the design and the construction of interiors and furniture, as we are also trying to show in this paper. This is valid for both stability or structural analyses and optimization analyses (geometrical and topological). The FEM analyses have been used in purely technical fields, e.g. in electrical technology, engineering or civil engineering, for some time now. However, considering the contribution of the furniture industry for example to the gross domestic product, it seems that FEM has a big potential in this field as well.

SOUHRN

Numerická analýza ozubového spoje vyrobeného CNC technologií

Cílem práce je parametrizace a numerická analýza mechanických vlastností otevřeného ozubového nábytkového spoje, vyrobeného specifickým postupem pomocí tříosého CNC obráběcího stroje.

Otevřený ozubový spoj je klasický nábytkový spoj, který se vyznačuje vynikajícími pevnostními vlastnostmi. Specifičnost konstrukce spoje, zajišťující samosvornost a následně výraznou pevnost, spočívá v náročnosti technologie výroby. Na tomto rohovém spoji byly provedeny simulace ohybového namáhání spoje v úhlové rovině – tlakem. Byly vyhodnoceny deformační pole, pole napětí, tuhost a ohybový moment spoje. V konečně-prvkovém systému ANSYS byly pro tento účel vytvořeny dva parametrické numerické modely daného spoje. První představuje dokonale tuhý spoj – oba díly spoje jsou slepeny dohromady. Druhý model zahrnuje navíc kontakt mezi díly spoje. Na tomto numerickém modelu byla sledována odezva tuhosti spoje na změnu statického součinitele smykového tření. Výsledky obou modelů byly porovnány jak mezi sebou navzájem, tak i s obdobnými analýzami z výzkumu demontovatelných nábytkových spojů. Uplatnění výsledků spočívá v možnosti komplexního navrhování složitějších nábytkových výrobků s vyššími požadavky na pevnostní charakteristiky, jako je tomu například u sedacího nábytku. Tento předpoklad je podmíněn korekcí sestavených parametrických modelů experimentálními zkouškami.

nábytek, ozubový spoj, numerická simulace, mechanické vlastnosti, CNC technologie

Acknowledgment

The research was supported by the Research Plan of Faculty of Forestry and Wood Technology (FFWT), Mendel University in Brno – ZA4501516 and the Internal Grant Agency of FFWT within project IG 490411.

REFERENCES

BEJO, L. et al, 2000: Friction coefficients of wood-based structural composites. Forest Products Journal. Vol. 50, No. 3. pp. 39–43.

ČERNOK, A., JOŠČÁK P., LANG M., 2004: Výpočtový model kolíkového spoja. In: Nábytok 2004, Drevárska Fakulta, Technická Univerzita vo Zvolene, 12 pp.

NUTSCH, W., 2003: Konstrukce nábytku, nábytek a zabudované skříně. Grada publishing Praha, 400 pp. ISBN 80-247-0220-7.

HIGHETT, C. G., 1987: The equivalent orthotropic elastic properties of plywood. Wood Science and Technology 21: 335–348, Springer –Verlag.

JOŠČÁK, P., 1999: Pevnostné navrhovanie nábytku. DF, TU vo Zvolene, 246 pp.

KOTLÍK, B. *et al.*, 2003: Matematické, fyzikální a chemické tabulky pro SŠ a nižší ročníky víceletých gymnázií. Fragment. 288 pp. ISBN 978-80-7200-521-5.

LANG, E. M., FODOR, T., 2002: Finite element analysis of cross-halved joints for structural composites. Wood and Fiber Science. 34 (2). pp. 251–265.

MERIAM, J. L., 1978: Engineering mechanics volume 1 – statics. SI/English Version. John Wiley & Sons. New York. 398 pp.

MIHAILESCU, T., 2003: Finite Element Analysis of mortise and tenon joint in parametric form. Transilvania University, Faculty of Wood Industry, Brasov, 145 pp.

SEBERA, V., ŠIMEK, V., 2008: Kontaktní analýza a optimalizace CNC ozubového spoje v MKP řešiči Ansys. Nábytok 2008, DF, TU vo Zvolene, pp. 1–11.

SUSNJARA, K., 2006: The New Furniture – How modern technology is changing the furniture and cabinet industry. Thermwood, Dale, Indiana, 210 pp.

ŠIMEK, M., HAVIAROVÁ, E., ECKELMAN, C., 2010: The Effect of End Distance and Number of Ready-to-Assemble Furniture Fasteners on Bending moment Resistance of Corner Joints. Wood and Fiber Science Journal, vol. 42 (1), pp. 92–98.

ŠIMEK, M., 2008: Mechanické vlastnosti nejpoužívanějších demontovatelných spojů korpusového nábytky. Doctoral thesis, MZLU v Brně, 139 s.

VESELOVSKÝ, J., 1996: Nábytkové spoje z natívneho dreva. Sympózium Nábytok "96", DF, TU vo Zvolene, s. 49–58.

ZHANG, J., ECKELMAN, C., 1993: The Bending Moment Resistance of Single-Dowel Corner Joints in Case Construction. Forest Product Journal, Vol. 43 (6), pp. 19–24.

ZHANG, J., ECKELMAN, C., 1993: Rational Design of Multi-Dowel Corner Joints in Case Construction. Forest Product Journal, Vol. 43(11/12), pp. 52–58.

Wood Handbook – Wood as an Engineering Material. Information on engineering with wood, properties of wood and designing with wood. September 28, 2002. Quoted on 15. 8. 2009. Available at: http://www.woodweb.com/knowledge_base/Wood_Handbook.html.

Address

Ing. Václav Sebera, Ústav nauky o dřevě, Ing. Milan Šimek, Ph.D., Ústav nábytku, designu a bydlení, Mendelova univerzita v Brně, Zemědělská 1, 613 00 Brno, Česká republika, e-mail: seberav@gmail.com, milansimek@gmail.com