International Scholars Journals

Global Journal of Business Management ISSN 6731-4538 Vol. 19 (1), pp. 001-011, April, 2025. Available online at www.internationalscholarsjournals.org © International Scholars Journals



*Full Length Research Paper*

# The use of finite element method in the furniture industry

**K. Hüseyin Koç[1], Koray Kizilkaya[2], E. Seda Erdinler[1] and Derya Sevim Korkut[3]***

[1]Istanbul University, Faculty of Forestry, Department of Forest Industrial Engineering, Istanbul, 34473, Turkey.
[2]Forest Industrial Engineer, Turkey.
[3]Duzce University, Faculty of Forestry, Department of Forest Industrial Engineering, Duzce, 81620, Turkey.

Accepted 20 April, 2025

**Computer Aided Engineering (CAE) techniques are getting more common everyday in the industry. The reason of this can be determined as the possibility of application of the CAE system besides the development of administration comprehension and industry with personal computers. In this study, Finite Element Method (FEM) application, which is an important engineering technique, was investigated with SolidWorks/CosmosWorks system. It is aimed to discuss the difficulties of FEM application based on the structure of the wood material and the conditions to transfer these to the Furniture Industry. For this purpose, proper analysis coefficient and virtual resistance values were developed with available standard, theoretic information and industrial application examples. At the second stage, the analysis was applied on a real product and the accuracy and applicability of the analysis approach have been evaluated by comparing the theoretic behavior of the product based on the software with the real results in practice.**

**Key words:** Finite element method, FEM, furniture, Computer aided engineering (CAE).

## INTRODUCTION

With the technological development, the modern management and production techniques integrated with the mathematical optimization techniques are quickly established in the businesses. Therefore, the businesses are required to catch up with the technologic develop-ment by using the computer and optimization techniques as a tool for maintaining their existence within the intense competition environment (Koç, 1994). During the technologic development process, direct or indirect use of computer aided design (CAD) and Computer Aided Ma-nufacturing (CAM) techniques is vital for the businesses. For example, product creation and development is facilitated in a manufacturing system supported with computer aided design, while an ordered or developed product can be quickly designed and analyzed and the required values related to the manufacturing such as unit manufacturing cost, manufacturing times and waste rates and similar can be calculated without any additional manufacturing cost. Furthermore, performing possible load analysis during the design process and calculating the magnitude and distribution of the forces occurring under load are the prerequisites for the reliability and economy of the product.

In finite element method (FEM), use of Poisson constants of bending properties in 3 dimensions helps prevent the problems due to non homogenous and anisotropic conditions (Guntekin, 2004). Techniques such as limited element analysis and finite element analysis available for the engineering analysis in CAD systems make the engineer displaying higher performance. This allows the removal of the indefiniteness before the manufacturing and making the decisions related to manufacturing in a more healthy and economical manner. Today, strength (engineering) design of furniture can be accomplished by utilizing solid modeling and structural analysis software. All members of the product can be modeled parametrically and required changes can readily be optimized via advantages that are provided by the solid modeling. Likewise, strength calculations of the designed product could be made by means of the computer aided structural analysis software (Kasal, 2006).

According to the results of the three-dimensional

*Corresponding author. E-mail: deryasevimkorkut@duzce.edu.tr. Tel: +90 380 5421137. Fax: +90 380 5421136.

structural analyses made by the finite element methods; computer aided analysis programs provide reasonable estimates overall strength of the sofa frames tested (Kasal, 2006). The shape and structural optimizations can be performed with this software. In addition to this, it is difficult to evaluate the construction with CAD tools. It is ensured that the model is sized properly by preventing the gradual changes and stress piles in sizing after the analysis. Adding the tolerances, performing tolerance analysis and cost analysis and using the factors that might affect the production are very valuable tools (Kurtoglu, 2003). Today, computer aided engineering (CAE) applications are coming out of theoretical basis and quickly taking their places in actual industrial applications.

In addition to the awareness level in industrial development and business administration understanding, particularly the easy use of the developed CAE systems in Personal Computers (PC) can be considered as important reasons for the development and generalize-tion of engineering analysis applications. The raw materials used in the furniture industry to be relatively more valuable and the quick exhaustion of the raw material resources make it obligatory to use the materials in question more effectively. With the use of CAD, manufacturing and analysis systems in the furniture industry, it is possible to calculate the costs of the parts during the designing phase and it also provides an opportunity for using the resources more effectively and for the optimization applications. FEM is one of these methods that provide such opportunity.

FEM is a numeric method used in the analyses of the behavior of materials or systems due to external factors (force, heat, electric, etc.). FEM covers the numerical solution of a mathematical model framed by the equalization of rigidity matrix {K} and deformation matrix {U} with force matrix {F}. There are numerous computer programs such as, ALGOR™, COSMOS/M™, NASTRAN™, ADINA™ and ANSYS™ (Guntekin, 2004). Generally, it is very important to define the wooden material correctly in the engineering analyses performed during the computer aided design process. Most of the physical tests are arranged for analyzing the characteristics of the homogenous materials. Transferring the materials such as trees and plants, which are not homogenous and displaying an anisotropic structure, to the computer environment and developing solutions for this purpose have a significant importance. In this study, the possibility of applying the finite element analysis technique on the wooden material by applying finite element analysis application on the sample products has been reviewed.

## MATERIALS

In the study, wooden board and the cabinets and bookcase made from wooden boards have been used as the test material. The boards and the products are based on melamine coated chipboard. In determining the resistance values of these boards, the results of the various previous studies (Bozkurt and Goker, 1990; Goker, 1978) and the standard values have been used for representing the general industrial average. The chipboard resistance values used in virtual environment are as follows:

Elasticity module : 28733 kp/cm$^2$ for 18 mm
Bending strength : 291.2 kg/cm$^2$
Compressive strength: 104.6 kg/cm$^2$ (Parallel to the surface)
Board density : 0.662 g/cm$^3$
Shearing module : 14363 kp/cm$^2$, determined by the software based on E module.

Determining the resistance characteristics in question and the application of finite element analysis are performed with the following four basic stages as a method. First, based on the available standard and theoretic information, the analysis coefficients suitable for the wooden boards have been developed. For this purpose, the laboratory tests suitable to the standards of the chipboard are simulated in virtual environment and the resistance values giving the same results are calculated and the accuracy of the parameters has been checked with various sample applications.

At the second stage, a bookcase used for three years and displaying different strains under different loads has been analyzed. For this purpose, measurements have been performed on three main cabinet groups and a bookcase comprised of five shelves for each cabinet. Each shelf is systematically divided into 4 equal parts and the loads on each part are measured with precision balance. For the deflection amounts occurring on the shelves, the deviation amounts from the horizontal plane are determined by performing a measurement in every 5 cm. The bookcase, given in Figure 1, has a width of 240 cm and a height of 210 cm. The length of each shelf is 76 cm and the depth is 29.5 cm. At the third stage, the finite element analysis application is performed by using the resistance values determined at the first stage. In this application, the real load values of are applied to the product and the theoretical behavior of the shelves based on the software are examined.

At the fourth stage, the accuracy and applicability of the coefficients and analysis approach developed by comparing the software based result and the real results in the application are evaluated. During the study, SolidWorks software is used for designing the application samples and creating the solid models and CosmosWorks software for analysis. Solid works is a comprehensive design software allowing the user to work on solid model and saving time at the design stage compared to the other CAD software. It is possible to assemble the prepared parts and the technical drawing of the part can be prepared automatically. SolidWorks allows the user to obtain a wide range of data. The parts contain data enabling the use of engineering analyses and computed aided manufacturing software at the design stage (Solidcam, 1999, 2004; Umtas, 2003). Cosmos works analysis software has been generally used in the metal industry. The system is an engineering software working with solid works support and performing finite element analysis (Kurtay, 2000; Turkcad, 2004).

Finite elements method (FEM) is a numerical method and is a very developed technique particularly used in computer aided solving of the problems such as solid mechanics, fluid mechanics, heat transfer and vibration. In FEM, the models are divided into finite number of elements. These elements are connected to each other at certain points at these points are called nodes (Hunter and Pullan, 2000; Karadag, 2003). In solid models, the displacements of each element are directly related to the displacements at the node points. The displacements at the node points are related to the tensioning of the elements. The finite elements method solves the displacements in these nodes. In this way, tension is found as approximately the same with the applied load. These node points must be fixed statically at some certain points (Piskin, 2000;

**Figure 1.** General view of the cabinet analyzed.

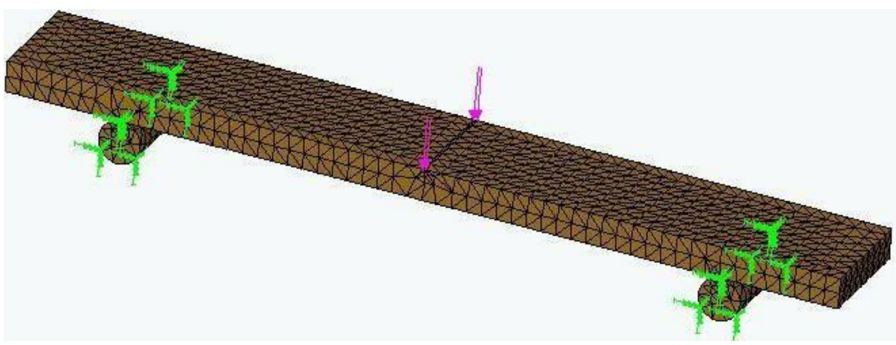

**Figure 2.** Sample model.

Anonymous, 2003, 2004). In finite elements method, the following steps are taken for the performance of the static analysis:

i.) Drawing the model.
ii.) Assigning material to the part.
iii.) Applying the limit conditions.
iv.) Applying the loads.
v.) Building a mesh for the part
vi.) Starting the static analysis
vii.) Determining the safety factor dispersion.

The use of FEM in design processes has significantly reduced the need for creating and testing a physical prototype in design. In this way, an important interaction has occurred between CAD and FEM and made the use of FEM in first and final stage of the design/manufacturing very practical and general. It is also possible to perform weight and volume optimization while determining the safety limits in design made based on the information received from FEM. However, FEM is not sufficient all alone. FEM must be supported with an axial design and with the results of the physical tests.

## RESULTS

### Determining the test simulation and virtual resistances

The laboratory tests performed for the chipboard are simulated in virtual environment and the resistance values giving the same results are calculated. For this purpose, the test model designs are made in solidworks design software and the constants on these designs are determined and the loads according to the standards are applied to the product. The sampled model developed is given in Figure 2 and the limit conditions are given below.

$$E = \frac{M \times l^3}{4 \times b \times d^3 \times \Delta}$$

Limit conditions

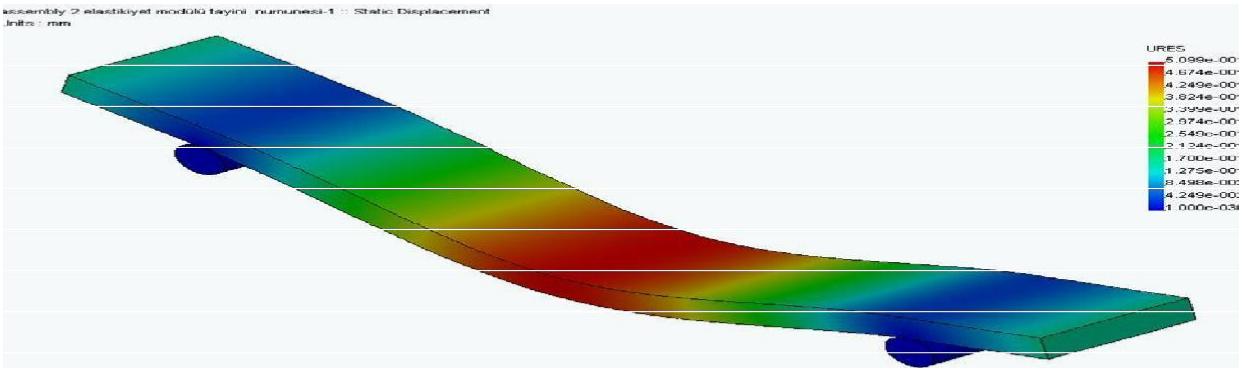

**Figure 3.** The display of the deviation on the sample based on the software.

**Table 1.** Calculations made With E-module formula.

| Parameters | Values 1 | Values 2 |
|---|---|---|
| M | 12.52 kg | 20 kg |
| I | 36 cm | 36 cm |
| | 0.116196244 cm | 0.18562 cm |
| B | 7.5 cm | 7.5 cm |
| D | 1.8 cm | 1.8 cm |
| E | 28733 kg/cm | 28733 kg/cm |

M = 0.64 × 18 = 11.52 + 1kg = 12.52 kg
I = 36 cm
b = 7.5 cm
d = 1.8 cm
E = 28733 kp/cm$^2$

In cosmosworks software, E module is determined as 28733 kp/cm$^2$ and the analyses have been performed and bending amount is found. The values found at the end of the analysis are compared with the values found by using the formula for bending elasticity and the deviation amounts are determined. In Figure 3, the displacement amount occurring on the sample is given as numerical value. However, based on the software, the visual displacement amount is sampled exaggeratedly. The total applied here is 12.52 kg and the amount of displacement found in the virtual environment as a result of this load is = 0.509 mm. The results found when the same displacement amount is calculated according to the bending elasticity module formula are as given in Table 1.

As it can be seen in Table 1, the amount of bending found according to E-module formula is 1.161962 for a load of 12.52 kg. The value of the analysis result based on the software is 0,509 mm. With a load of 20 kg, the bending occurs as 1.856 mm based on E-module formula and 0.771 mm based on the software. In order to eliminate the difference between the formula and the software, loads increased by 10 kg have been applied at the second stage and the results given by the software are compared with the values found by the formula. The resulting values in CosmosWorks are given as graphic in Figure 4. The data based on E-module formula and software are given in Table 2. When the value found as a result of arithmetic mean is multiplied with the E-Module value used in the analysis, 28733*0, 4161315 = 11956,706 is found. If this value is redefined as E-module in CosmosWorks analysis software, the change in the analysis results is as given in Table 3. As it can be seen in Table 3, the deformation values found for the chipboard in CosmosWorks almost give the same results with E-module formula with the changes made. This result found for chipboard will make it possible to see the zonal displacements in design analysis of big size assembled designs in more detail.

As a result of a second test simulation, the sample used in determining the bending strength is drawn and analyzed in virtual environment. The value in bending elasticity module is used here and Pmax is determined (Figure 5). Based on the results of the previous studies, the bending strength was determined as 291.2 kp/cm$^2$. The max P force provided by this is 157,248 kp as seen in Table 4 according to the following bending strength formula.

$$\sigma eg = \frac{3}{2} \times \frac{P_{max} \times L_s}{b \times a^2}$$

When the analysis is performed by applying a Pmax force of 157.248 kp in CosmosWorks, the bending strength in max. Deformation is calculated as 289.9 kp/cm$^2$ . In this

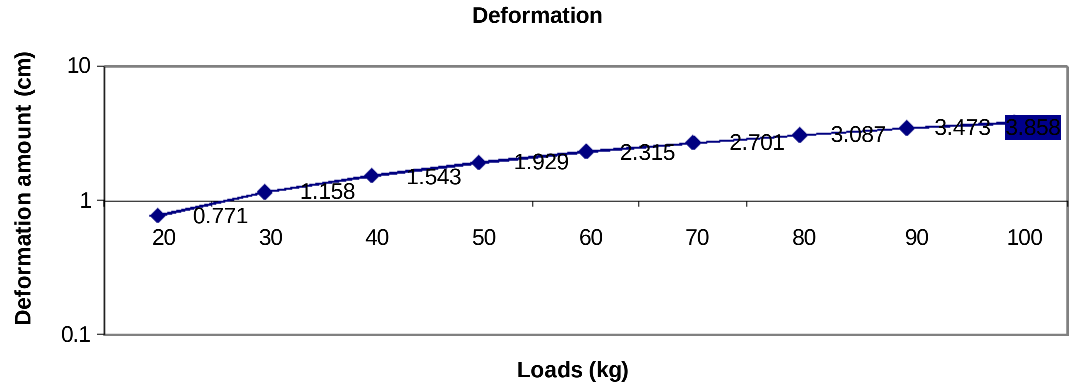


**Figure 4.** Loading-deformation amount relation in cosmos works software.

**Table 2.** Comparison of software based results with formula based results.

| Load (kg) | Displacement (cm) | E-Module Formula A (mm) | CosmosWorks Result B (mm) | A/B |
|---|---|---|---|---|
| 20 | 0.185 | 1.85 | 0.771 | 0.416756757 |
| 30 | 0.278 | 2.78 | 1.158 | 0.416546763 |
| 40 | 0.371 | 3.71 | 1.543 | 0.415902965 |
| 50 | 0.464 | 4.64 | 1.929 | 0.415732759 |
| 60 | 0.556 | 5.56 | 2.315 | 0.416366906 |
| 70 | 0.649 | 6.49 | 2.701 | 0.416178737 |
| 80 | 0.742 | 7.42 | 3.087 | 0.416037736 |
| 90 | 0.835 | 8.35 | 3.473 | 0.415928144 |
| 100 | 0.928 | 9.28 | 3.858 | 0.415732759 |
| Arithmetic mean | | | 0.416131503 | |

**Table 3.** Coefficient based change of software and formula based results.

| Load (kg) | Displacement (cm) | E-Module Formula A (mm) | CosmosWorks Result B (mm) | A / B |
|---|---|---|---|---|
| 20 | 0.185 | 1.85 | 1.858 | 1.004324 |
| 30 | 0.278 | 2.78 | 2.786 | 1.002158 |
| 40 | 0.371 | 3.71 | 3.715 | 1.001348 |
| 50 | 0,464 | 4.64 | 4.644 | 1.000862 |
| 60 | 0,556 | 5.56 | 5.573 | 1.002338 |
| 70 | 0.649 | 6.49 | 6.501 | 1.001695 |
| 80 | 0.742 | 7.42 | 7.43 | 1.001348 |
| 90 | 0.835 | 8.35 | 8.359 | 1.001078 |
| 100 | 0.928 | 9.28 | 9.288 | 1.000862 |
| 100 | 0.928 | 9.28 | 9.288 | 1.000862 |
| | Arithmetic mean | | 1.001779 | |

case, the E-module value determining the bending strength and the values in CosmosWorks have provided almost the same result. The deviation is 0.45%. As a result of this, the opportunity that the analysis applied in CosmosWorks with the use of the virtual resistance values created reflects the wooden board by being different from the metal material. The virtual resistance values giving the same min. deviation for 18 mm chipboard in CosmosWorks according to the analysis results are given in Figure 6.

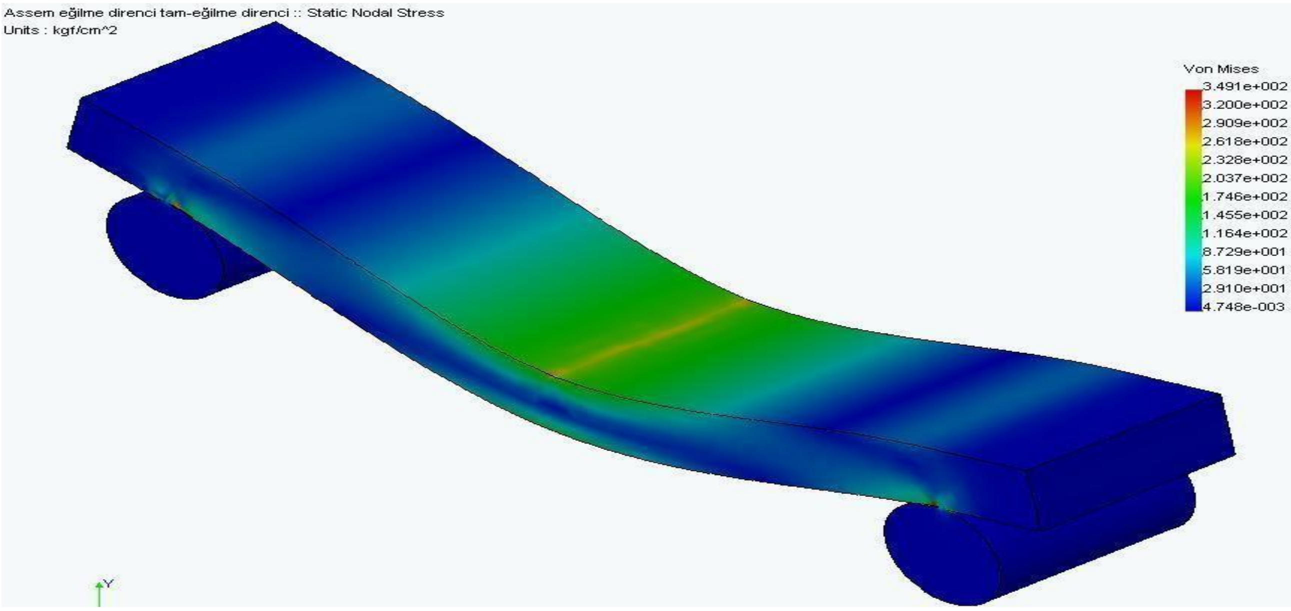


**Figure 5.** Zonal tension amounts are displayed.

**Table 4.** Calculations made with E-module formula.

| Parameters | Values(cm) | Pmax | Bending Strength |
|---|---|---|---|
| L | 25.0 | | |
| Ls | 20.0 | 157.248 | 291.2 kp/cm$^2$ |
| A | 1.80 | | |
| B | 5.00 | | |

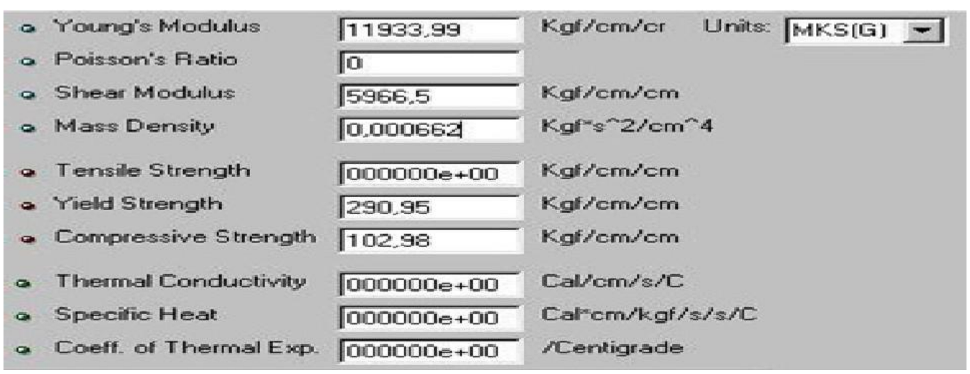


**Figure 6.** The virtual resistance values determined for cosmosworks software.

## EXAMINATION OF THE BOOKCASE IN TERMS OF LOAD AND DEFLECTION

At this stage, the bookcase used for 3 years and modeled in solid works as given in Figure 7 is examined in terms of load and the occurring deflection. The load and deflection values obtained as a result of the measurements performed with a digital compass and precision balance are summarized in Table 5. These data, in general, display the average and maximum values of deflection occurring with the available total load of each shelf. Furthermore, it is shown with graphic on 2 separate shelf examples in order to constitute an example for the deflection formation (Figures 8 and 9).

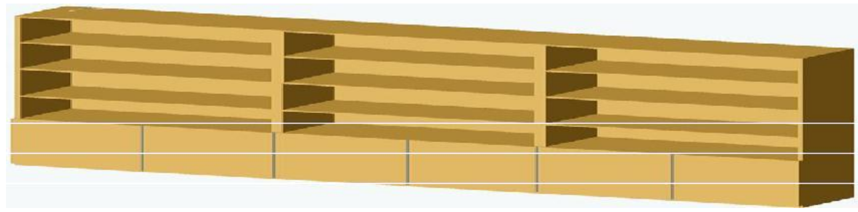

**Figure 7.** Solidworks modeled appearance of the bookcase in use.

**Table 5.** The real load and deflection values of the bookcase.

| Shelves | Load and deflection | 1st Bookcase | 2nd Bookcase | 3rd Bookcase |
|---|---|---|---|---|
| 1ST Shelf | Load amount (kg) | 23.45 | 27.65 | 27.50 |
| | Max. deflection (mm) | 1.11 | 2.64 | 1.29 |
| | Ave. deflection (mm) | 0.50 | 1.76 | 0.61 |
| 2ND Shelf | Load amount (kg) | 22.9 | 24.5 | 29.9 |
| | Max. deflection (mm) | 2.16 | 1.62 | 0.79 |
| | Ave. deflection (mm) | 1.29 | 0.88 | 0.32 |
| 3RD Shelf | Load amount (kg) | 13.9 | 27.9 | 29.8 |
| | Max. deflection (mm) | 1.61 | 1.34 | 1.39 |
| | Ave. deflection (mm) | 1.06 | 0.98 | 0.72 |
| 4TH Shelf | Load amount (kg) | 26.8 | 8.10 | 31.8 |
| | Max. deflection (mm) | 0.58 | 1.96 | 1.25 |
| | Ave. deflection (mm) | 0.38 | 1.28 | 0.62 |
| 5TH Shelf | Load amount (kg) | 22 | 22.2 | 22.1 |
| | Max. deflection (mm) | 8.13 | 8.14 | 0.88 |
| | Ave. deflection (mm) | 5.32 | 2.58 | 0.39 |

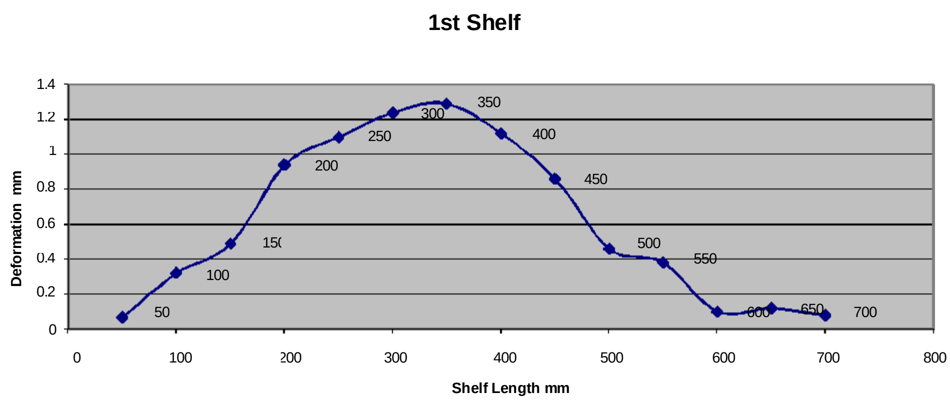


**Figure 8.**The progress of deformation on first shelf of the third cabinet.

## VIRTUAL BOOKCASE DESIGN AND ANALYSIS

After detailed part drawings, combining figures and assembly processes are modeled in SolidWorks, the bookcase, previously given in Figure 7, is analyzed with the previously determined virtual resistance values by applying the real loads on each shelf in CosmosWorks software. In this way, the virtual deflection amounts arising

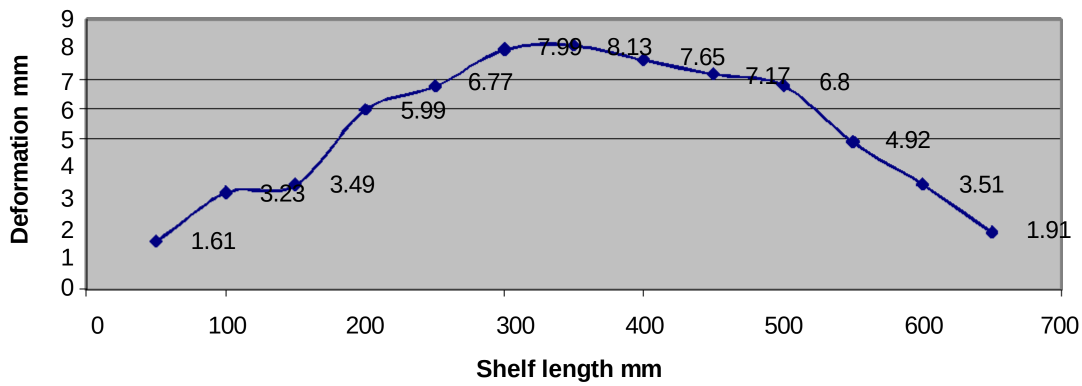


**Figure 9.** The progress of deformation on fifth shelf of the first cabinet.

**Figure 10.** The mesh structure arising in finite element analysis of the bookcase.

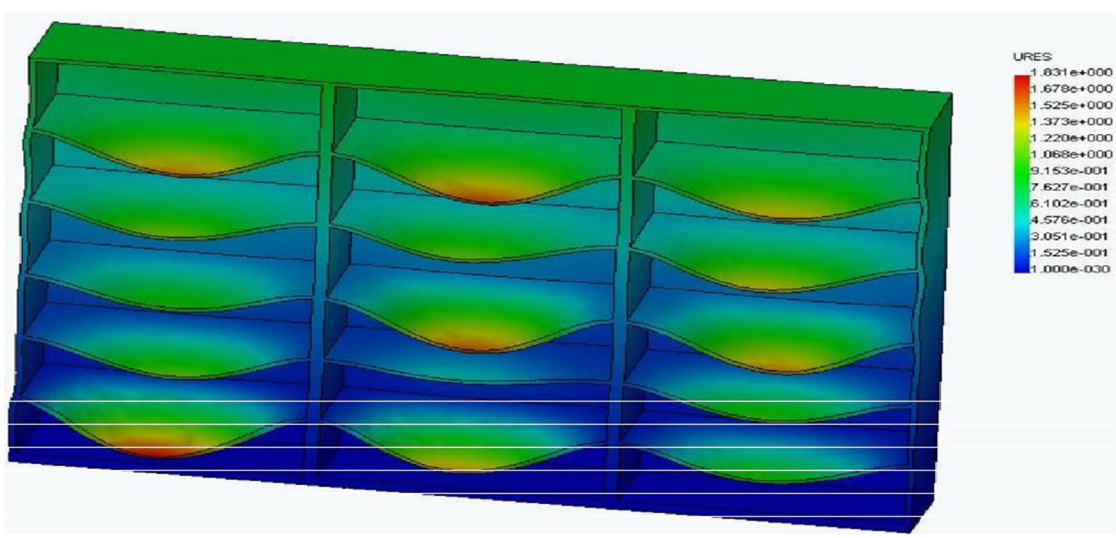


**Figure 11.** Simulated appearance of deformation in bookcase.

arising on each shelf of the bookcase are determined. The mesh structure of the bookcase created for the Finite Element analysis is given in Figure 10 and the simulated appearance of the deformation is given in Figure 11.

## THE COMPARISON OF THE REAL VALUES AND THE SOFTWARE BASED VALUE OF THE BOOKCASE

When each cabinet and shelves of the bookcase are

**Table 6.** Deflection amounts of first and second cabinets (mm).

| | First Cabinet | | | | | Second Cabinet | | | | |
|---|---|---|---|---|---|---|---|---|---|---|
| | 1st Shelf | 2nd Shelf | 3rd Shelf | 4th Shelf | 5th Shelf | 1st Shelf | 2nd Shelf | 3rd Shelf | 4th Shelf | 5th Shelf |
| Cosmos works | 1.0 | 1.1 | 0.9 | 0.6 | 1.83 | 1.6 | 1.06 | 1.35 | 0.4 | 1.5 |
| Real values | 1.11 | 2.16 | 1.61 | 0.58 | 8.13 | 1.76 | 1.62 | 1.4 | 1.96 | 8.14 |

**Table 7.** The comparison of real and virtual values for third cabinet.

| | 1st Shelf (mm) | 2nd Shelf (mm) | 3rd Shelf (mm) | 4th Shelf (mm) | 5th Shelf (mm) |
|---|---|---|---|---|---|
| CosmosWorks | 1.15 | 0.8 | 1.35 | 1.2 | 0.9 |
| Real Values | 1.29 | 0.79 | 1.39 | 1.25 | 0.88 |

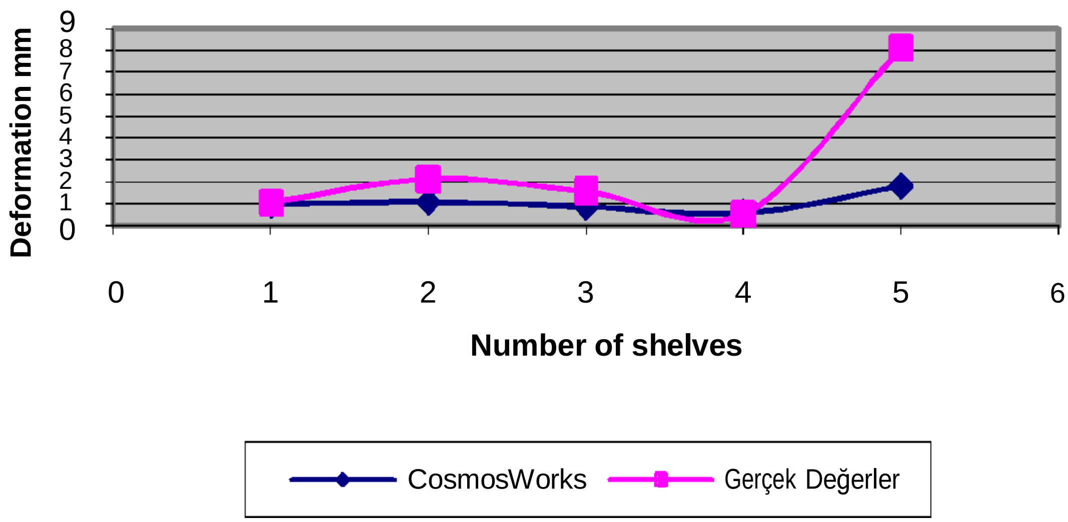


**Figure 12.** Deflection change in 1st Cabinet.

evaluated separately, the results given in Tables 6 and 7 are obtained. In order to facilitate the evaluation, the data in Figures 12, 13 and 14 are graphically interpreted. As it can be seen from the tables and graphics with comparison, the real values are slightly higher than the values generated by the software. Although the amount of real deformation in fifth shelf of the first cabinet is 8.13 mm, the virtual result is only 1.83 mm. The reasons for this can be explained as follows. The tiring loadings for approximately three years decrease the resistance of the board and the homogeneity of the load distribution is impaired by time. Furthermore, the opening arising at the junction places of the shelves can be one of the important reasons for this. As it can be seen in the deformation values of the third cabinet, the results are so close to each other and the average deviation is 11%. At it can be seen in Table 7, the real values are slightly higher than the virtual values.

## CONCLUSION AND SUGGESTIONS

In computer aided engineering applications, particularly most of the analysis software, the wooden material characteristics are not defined. For example, when the features such as elasticity model, Poisson rate, pressure and shearing resistance etc. are used in analyses, the deformation and tension amounts arising are performed by considering that the wood will behave like metal material. The wood, however, displays a specific behavior with its heterogeneous and anisotropic structure and therefore it is needed to develop special coefficients suitable to wood or virtual resistance values with another approach.

Reliability concepts are depicted in general both in their application to furniture and in their incorporation into standards for performance testing. The major objective of reliability and performance testing is to improve the

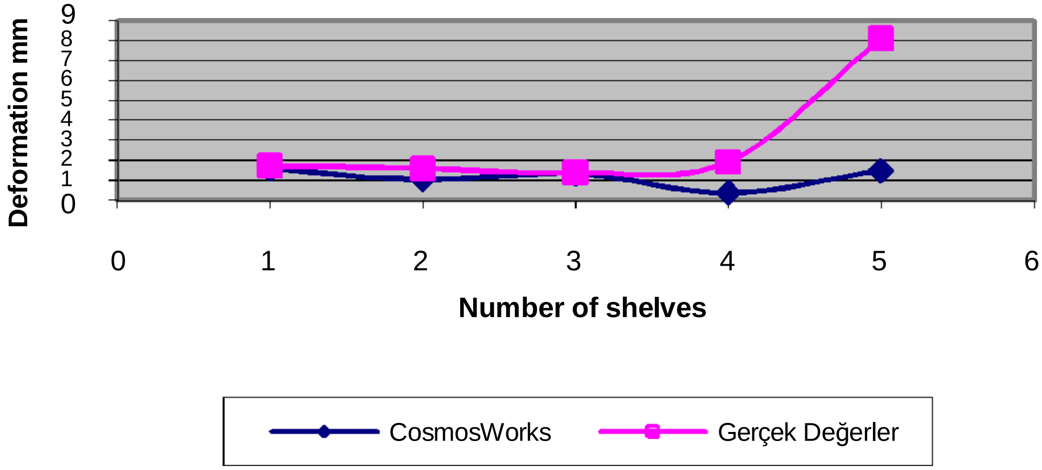


**Figure 13.** Deflection Change in 2nd Cabinet.

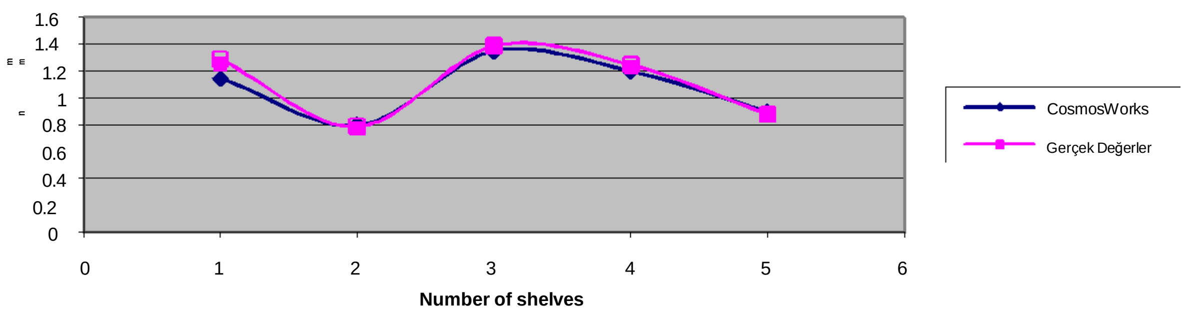


**Figure 14.** Deflection change in 3rd Cabinet.

durability and safety of furniture products and to predict failure or unexpected problems associated with them (Erdil et al., 2004).

Although the wooden board products are developed for obtaining homogenous material from the wooden material, they display difference from metal and similar materials in terms of behavior. The physical characteristics and manufacturing technologies of the wooden type particularly used in chip and fibers cause different resistance values for the boards and cause them display different behaviors against the loads. Therefore, in this study, first virtual resistance values are developed for wooden boards and then by applying these values on a real product, the real values are compared with the software based values.

The high quality furniture that can rival in the world markets could be produced with the widespread use of developed methods, computer aided analyses and performance tests (Kasal, 2006). As a result of the study, it has been revealed that the finite Element analysis applications can be performed successfully by creating proper virtual resistance values on a wooden board based material. However, it is required to separately determine the virtual resistance values for boards with different characteristics in order to generalize the application industrially. New studies must be planned within this scope.

The resistance values determined under CosmosWorks can be used in the design of new furniture and creating the construction without any need for safety tension. This method can also be supported by optical measurement techniques. Optical measurement techniques, when used in material and structural testing, provide considerably more information than conventional techniques do (Enquist, 2005). But it must be considered in the analysis and modeling works carried out that the effects dynamic tiring load in real life can be very variable and the tolerances must be determined according to this.

## REFERENCES


Anonymous (2003). Enterprise Solution for Simultaneous Product Development, website publication, www.impactxoft.com, [03.05.2003].

Anonymous (2004). History of Finite Elements Analysis.

Bozkurt A, Goker Y (1990). Chipboard Industry Book, Istanbul University, Faculty of Forestry.

Enquist B (2005). Use of Optical Measurement Techniques in Furniture Design, Chalmers University of Technology, Göteborg Sweden 2005, Building Design papers nr. 2-2005.

Erdil YZ, Haviarova E, Eckelman AC (2004). Product Engineering and Performance Testing in Relation to Strength Design of Furniture, July 2004. Wood Fiber Sci. J., 36(3): 411-416.

Goker Y (1978). Plywood, Blockboard and Chipboard Industry in Turkey, Development Opportunities, Studies on Technologic Characteristics of These Materials, Istanbul University. Publication No: 2489, Faculty of Forestry Publication No: 267.

Guntekin E (2004). Finite Element Analysis of Doweled Furniture Corner Joints, Süleyman Demirel University, J. Fac. Forest., 159-169. Series A, No.1, ISSN: 1302-7085.

Hunter P, Pullan A (2000). FEM/BEM Notes. Department of Engineering Science, University of Auckland, New Zealand; March 12, 2000.

Karadag V (2003). Finite Elements Management and Applying This on Bending Movement, Istanbul Technical University, Faculty of Mechanical Engineering, Gumussuyu, Istanbul.

Kasal A (2006). Determination of the Strength of Various Sofa Frames with Finite Element Analysis. G.U J. Sci., 19(4): 191-203.

Koç KH (1994). Applying Computer Aided Manufacturing in Forestry Products Industry, Summary of Doctoral Thesis, Series A. Istanbul Univ. Fac. Forest. J., 44(1): 71-79.

Kurtay T (2000). Finite Element Analysis Course Notes, Istanbul Technical University, Faculty of Mechanical Engineering, Gumussuyu, Istanbul, May 2000.

Kurtoglu A (2003). Istanbul University Faculty of Forestry, Department of Forest Industrial Engineering, Wooden Works, Furniture and Construction Course Notes.

Piskin S (2000). Istanbul Technical University Institute of Informatics Calculated Science and Special Topics in Engineering, Mesh Production.

Solidcam (2004). Solidcam Website, www.solidcam.com, [05. 08.2004].

Solidcam (1999). The Integrated CAM Engine for SolidWorks Presentation CD, Sanal Gazete, December, 1999

Turkcad (2004). Turkcad Website, www.turkcadcam.net, [01.08.2004].

Umtas (2003). International Machinery Trade A.ß., SolidWorks/CosmosWorks Training Notes, Introduction to Finite Elements Method with CosmosWorks.