Optimization of Thickness in Wood Furniture Structure

María Angélica Rodríguez C[1], David Gallardo G[2], Jacobo Tolamatl M[3], Román Daniel Romero M[4]

[1,2,.3]Engineering Master Bachelor, Tlaxcala Polytechnic University, San Pedro Xalcaltzinco, Tlaxcala 90180 Mexico
[4]Industrial Processes and Operations Engineering, Tlaxcala Technological University, El Carmen Xalpatlahuaya, Huamantla, Tlaxcala 90500, Mexico.

---***---

Abstract – *The correct thickness of a structural piece is very important to ensure the functionality and the economic viability of a product, so the way to achieve this challenge is making use of tools such Computer Aided Engineering and the analysis of the results that this technology gives.*

The scope of this paper encompasses the research of Douglas Pine properties to type them in SolidWorks software, the modelling of structural pieces of this material, the stress simulation, analysis and optimization of this kind of wood.
In order to achieve optimization, it defines the safety factor for this kind of material; taking into account that wood is an anisotropic material as well as the probability of overloading by the final user.

The results allowed reducing the cost of wood furniture and ensuring that these products will not fail due to the stresses when loading nor overloading.

Key Words: Optimization, Wood, Stress

1. INTRODUCTION

The main thing in any furniture made of wood is the functionality, which is given by the material and its corresponding properties like the strength to normal and shearing stresses, but the dimensions of the cross section are also important to resist the load that the objects placed on the furniture. These loads produce tension and compression stress in the material and deformations that can be small or large depending on the dimension of the cross section of the wood and the weight of the objects that are on the furniture. The cross section is integrated by the thickness and the width of the components that encompass the furniture. The width is a variable that cannot be changed in the furniture because it provides the area available to place things in the furniture along with the length. So the thickness is the only variable that can be changed in order to optimize the functionality of the product. This optimization must accomplish with two main functions, the first one is the strength enough to withstand the loads in the elastic zone of the material and the second one is the quantity of material must be the necessary to work in the elastic zone. It means that a small quantity of material or less material than necessary can lead to a failure in the structure, on the other hand the excessive quantity of material can produce a heavy and expensive product.

The method of assessment this kind of products is the using of Computer Aided Engineering that allows to make decisions based on the maximum stress produced by the loads applied on the cross section of the wooden furniture and the comparison of this parameter with the elastic limit and considering a safety factor that must be defined for this type of material.

Thus the proper use of this important material can yield benefits like:

• Less cost of raw material

• Assurance of functionality

• Better exploitation of material properties

• Economical profits

All of this is possible to achieve by means of the use of Computer Aided Engineering which is friendly in its use through the modelling, stress simulation, assessment and optimization if necessary of the interest variable that in this case is the thickness.

The wooden furniture can be a wardrobe, a chair, a table, a bookshelf, in all these cases there is always a load that must be resisted by the product, in the first case the clothes are the load, in the second case, a person is the load, in the third case the plates, vases, jars and food are the load and in the fourth case the books are the weight to bear. The loads produce stresses and deformations and even breakages in this products, so the challenge for the manufacturer of wooden furniture is to design a product that resist the loads in order to keep the stresses in the elastic zone of the material but without falling into excess or running the risk of failure.

This paper develops the methodology to evaluate the strength of the structural components that integrate the wooden furniture, especially those ones that are made of Douglas Pine. The basics of this methodology is the Finite Element Method that is mentioned as follows.

1.1 Finite Element Method

The Finite Element Method is a numerical method that looks for an approximate solution of the field variables distribution that is difficult to obtain analytically. This makes dividing the problem domain in several elements as shown in figure 1 [1].

The Finite Element discretization procedures reduce the problem to a finite number of unknown quantities by dividing the solution region in elements and expressing the field variable of the unknown quantities in terms of the approximation functions assumed in each element. The approximation functions are defined in terms of the field variable values in points specified called nodes or nodal points. The nodes usually fall in the element borders where

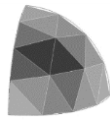the adjacent elements are connected, as can be observed in figure 2 [2]

Figure 1. Discretized hemispheric section in several elements.

Besides, Computer Aided Engineering has as fundament the Finite Element Method and it is in several Computer Aided Design software is a powerful tool that can be used to evaluate and optimize the design of products.
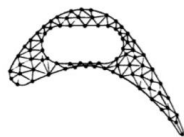

Figure 2. Nodes of a turbine blade profile

The goodness of the use of this tool can be reflected in the cost reduction of raw material and in setting a certainty that the products will work without any problem in relation to mechanical area, in specific way regarding to the strength that can define to withstand the loads that produce the stresses and deformations in structural parts.

When an engineer develops a project has to make modelling, simulation, visualization, analysis, design, prototyping, test and finally the project elaboration processes. It is in these processes where the Finite Element is utilized like a powerful technique of modelling and engineering system analysis, saving effort and resources [3]

Finite Element Method uses in a massive way in the static and dynamic analysis of structural systems [4].

In static analysis, where investigates stresses and displacements, also complicated optimization problems, where the goal is finding the best design suit for the given conditions for which, it carries out a finite element analysis [5]

Due to the great capacity of the current computers, the use of this method has become massive and established in a big community of engineers and scientists [6]

The discretization tools that have many software, let the user generate meshes of finite element in an automatic way, requiring only of some parameters like tentative length of the element or the kind of element (rectangular, triangular, hexagonal, tetrahedron, etc.)

Regarding, the geometrical shape of the elements that integrate the mesh, in flat stress problems [7] compares the accuracy between rectangular and triangular shape element, concluding that rectangular element is more accurate that triangular one because of the strain state is approximate in linear terms, while the triangular element has only constant terms, so it must increase the mesh density significantly to obtain a similar precision.

The continuum means mechanics let formulate the big part of the engineering problems at a macroscopic level where the means is considered continuum and therefore the effects of the material molecular composition are not considered. The most of the problems associated to this branch of mechanics are ruled by differential equations and their solution in the majority of the cases is only possible through a numerical method like the Finite Element Method [8]

The steps involved in the finite element formulation are the rigidity matrix assembly and the load vector (also called force vector), this matrices for all of the elements will produce the global rigidity matrix and the global load vector to formulate the equations system [9]

2. Pine Douglas Mechanical Properties

Due to in the CAE software there are no parameters of Pine Douglas mechanical properties in order to simulate the stresses and deformations, it had to investigate about them. The selected wood is Douglas Pine, the parameters for this material that were found in several information sources are shown in table 1.

This information summarizes the parameters of the mechanical properties that are necessary to run the simulation of stresses and deformations in Computer Aided Manufacturing.

Table-1: Mechanical properties of Douglas Pine to simulate stresses and deformations.

Mechanical Properties	Units	Parameter	Source of information
Density	Kg/m^3	470	Beer, 2004
Ultimate strength (tension)	N/m^2	100×10^6	Beer, 2004
Ultimate strength (compression)	N/m^2	50×10^6	Beer, 2004
Ultimate strength (shear)	N/m^2	7.6×10^6	Beer, 2004
Elasticity modulus	N/m^2	13×10^9	Beer, 2004
Poisson coefficient (Longitudinal orientation – low density)	---	0.40 ± 0.05	atcp.com.br, 2018
Thermal conductivity	W/m.K	0.12 – 0.04	didactica.fisica., 2018
Specific heat	J/Kg-	1760	valvias.com,

	°K		2018
Elastic limit	N/m²	15.2 x 10⁶	fao.org, 2018
Thermal dilatation coefficient	°C⁻¹	3.4 x 10⁻⁵	vaxasoftware, 2018
Damping coefficient	----	0.23	fau.ucv.ve, 2018

3. Procedure to edit the parameters of material selected

To make this activity, it proceeds the next way.

1. It selects Material option without specifying the file of a new part (part to draw), and with right click selects option Edit material.

2. Then, it selects the folder of Customized Materials and with right click activates the window where appears New Library option.

3. Once selected this option, it types the name of the new library and saves it.

4. Right away it selects the new library and with right click activates the window to create a new category of materials.

5. When selecting the new category option activates a new option and there types the name of material to customize.

6. Then, it looks for in SolidWorks materials, the most similar material that it will be customized, in this case it selects the option Pine of the Wood library and when activating the right click, appears the window that allows to copy this material to edit it later. All of the mechanical properties type directly from the investigated sources.

7. When finishing the typing of parameters of the necessary properties to run the simulation, it selects the options apply and close and it observes in the screen how it defines the material in this case Douglas Pine in a realistic way, as it can see in figure 3.

4. Drawing of a bookshelf parts

Once it achieved to define the material and the parameters of its mechanical properties, it proceeds to select the kind of furniture that will be simulated and optimized if necessary, for that a bookshelf is selected, due to it is exposed to loads that can deform permanently, so the aim of this work is to design the parts of the furniture in such a way to have a strength and economic structure in order to resist the loads and not to waste raw material from the thickness parameter.
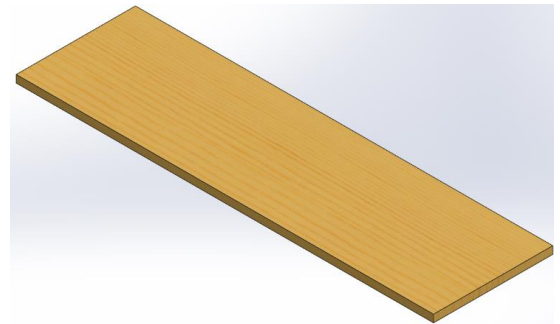

Fig -3: Definition of material (Douglas Pine) in SolidWorks

So, it proceeds to draw the parts of the bookshelf as suggested by professional carpenters, the thickness of the tables to use is 1 inch (25.4 mm).

The aim of doing this activity is to prepare the model to simulate the stresses and deformations, this preparation includes the material definition, the dimensions setting for each component of a bookshelf.

Then, it proceeds to assembly all of the parts to have a complete product.

Once the parts are modelled, it makes the virtual assembly to see how the bookshelf looks complete, this can see in figure 4.
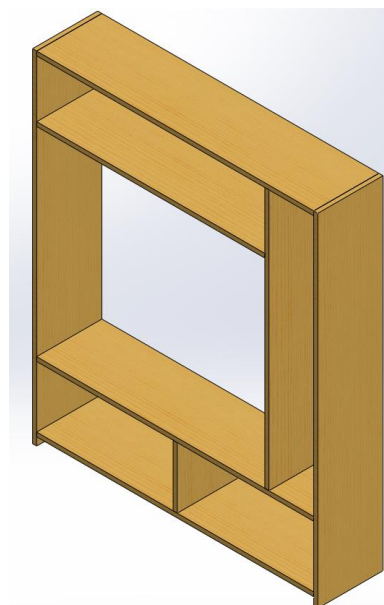

Fig -4: Isometric view of the bookshelf virtually assembled

5. Procedure to simulate stresses

The first step is to define the kind of study to make, in this case is a statically analysis.

Due to the nature of loads that will evaluate in this paper (weight of books on bookshelf tables), the loads are static.

After defining the kind of study, the next step is to set the fixation that in this case are the longitudinal ends of the table for tables that work like a beam, as it can observe in figure 5.
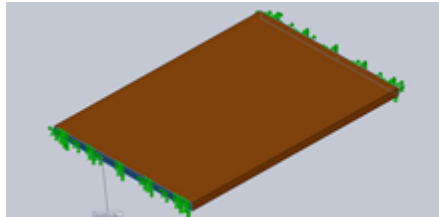
Fig -5: Definition of fixation

Then, it is necessary to select the corresponding material with the respective parameters previously edited, because this information is not available on the regular software of CAE.

The definition of direction and magnitude of loads is the next step in this procedure, in this case the loads are distributed uniformly on each table with a downward direction and different magnitude for each table, for the first table the load is 92.214 N. In figure 6, it can see this.
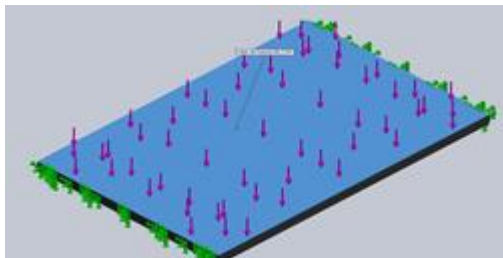

Fig -6: Definition of direction and magnitude of the load

Then, it proceeds to mesh the model according to the accuracy level required, that in case selects the medium level in order not to have a rough accuracy and not to need a lot of time to process the calculation.

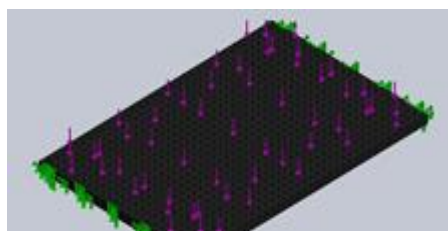Once this level selects, the shape of the model changes showing the mesh, as it can see in figure 7.

Fig -7: Model meshing

After that, it runs the deformation simulation in order to have an image about how the material will work, this presents in figure 8.
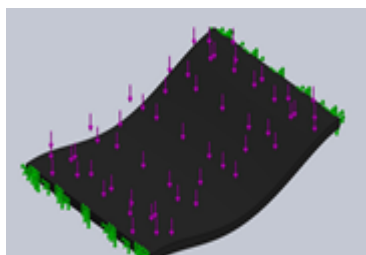

Fig -8: Deformation simulation

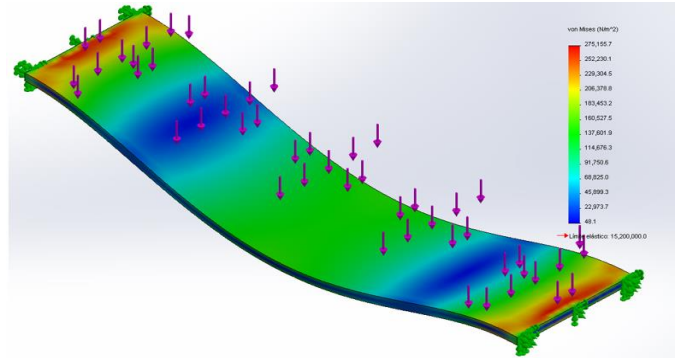Now, it can have the results of stress, these is shown in figure 9.

Fig -9: Results of stress simulation

Due to the maximum stress in this first case is 275 155.7 N/m^2 and the elastic limit for the material used is 15.2 x 10^6 N/m^2, it notices that there is a lot of material in excess and taking into account the safety factor criterion, it finds that is 55.24 which is very high.

The results for all of the analyzed tables shown a similar behavior, so it run the optimization of the cross section for all of the tables that integrate the complete furniture.

6. Optimization of wood cross section

It selects the optimization option in the CAE, which has two main windows that are: variable view and results view. The first window suggests a range of dimensions where the optimization can take place regarding the thickness of the table, these values can be change by the user or they can remain as suggested by the CAE, there is also a constraint that the user can use that is the safety factor in this window, so it chooses the parameters of range and safety factor and the optimization executes. This shows in figure 10.

When executing the optimization with the constraints defined by the user, it gets the results, as it can see in figure 11.

Fig -10: Optimization window

Variable view	Results view		
		Initial	Optimum
D1 Protruding1 (0.0254)		25.4 mm	12.7 mm
Safety factor		55.241447	14.554417
Mass		8.11784 kg	4.05892 kg

Fig -11: Results of optimization

When executing optimization, it observes an important reduction of thickness and therefore, safety factor and mass reduce.

All of the parts of the furniture studied behaved in the same way. They required optimization execution because the safety factor values were high.

5. Results

By comparing the initial and final parameters of the cross section, safety factor and mass of the tables used to build the furniture, which is a bookshelf in this case, it can realize how they changed when optimizing, in general there was a reduction in the cross sections, therefore, it carries to use less material and make cheaper the cost of raw material for this kind of product, and at the same time it assures that the furniture will work without any problem under the loads defined, even, it can support at least 8 times an overload, this value is also defined because the wood is an anisotropic material, this is a very good practice because allows to do a good use of the material,

The initial conditions correspond to the parameters of thickness that the furniture manufacturer has defined based empirically, so these values generate the safety factor and the respective mass for each table.

The final conditions correspond to the optimized ones through CAE. These results show an important reduction with regard to the values defined in an empiric way.

In table 2 shows the summary of all the tables analyzed and in table 3 can see the reduction of thickness and mass of the same parts, which are the tables.

Table-2: Comparison of initial and final design parameters.

Item	Description (Table)	Safety Factor		Thickness (mm)		Mass (kg)	
		I	F	I	F	I	F
1	Lower horizontal	55.1	10	25.4	12.7	8.1	4.0
2	TV Set	29.5	10	25.4	15.8	8.1	4.6
3	Upper	83.8	8.	25.4	9.5	6.6	2.1
4	Lower	540	83	25.4	6.35	1.7	0.2
5	Side	654	78	25.4	3.1	10.2	1.1

Table-3: Percentage reduction of thickness and mass.

Item	Description (Table)	Reduction of thickness and mass (%)
1	Lower horizontal	50
2	TV Set	42.69
3	Upper	68.51
4	Lower	84.26
5	Side	88.46

Table 3 shows that the minimum percentage reduction is 42.69 and the maximum is 88.46, this means that the optimization process had a great impact in the cost and weight of the furniture.

6. CONCLUSIONS

The use of Computer Aided Engineering is a powerful tool to evaluate stresses that generate in materials subjected to loads, as in this case a wood furniture which have to bear the weights that the user places on it.

The assessment obtained by this means must analyze to determine if the value of the maximum stress is allowable by comparing with the elastic limit value. If the maximum value generated in the stress simulation is lower than the elastic limit, it means that the piece has more material than required and it is necessary to optimize it, depending on the safety factor to take into account as engineering reference, on the other hand if the maximum value of stress in bigger than the elastic limit, there is an imminent failure and it is necessary to optimize it based on a defined safety factor.

This work is an input to the engineering area to evaluate wood made products, because in CAE software there are no parameters to simulate stresses for this material, so by following the methodology presented, it can achieve the simulation and optimization of cross section of wood furniture or another product manufactured with it.

The advantage of this kind of technology is that there is no need to make destructive test to assure to evaluate materials regarding their mechanical strength.

A relevant aspect is the designer criterion, who has to take into account the possibility of overload, the nature of the material and the work conditions of the product in order to set a reasonable safety factor to ensure the functionality and the economy of the product.

REFERENCES

[1] Akiyama, H. (2003). Metodología de proyecto sismorresistente de edificios basada en el balance energético. Barcelona: Editorial Reverté

[2] Alter, M. J. (2004). LOS ESTIRAMIENTOS. Barcelona: Paidotribo

[3] Angulo, J. A. (2016). Análisis y diseño de piezas con CATIA V5. México: MARCOMBO

[4] Beer, F. P. (2004). MECÁNICA DE MATERIALES. México: Mc Graw Hill

[5] Budynas, R. G. (2012). Diseño en Ingeniería Mecánica de Shigley. México: Mc Graw Hill

[6] Bustamante, M. d. (2015). Simulación por Ordenador Mediante el Método de Elementos Finitos y Optimización por el Método de Ingeniería Robusta, de

Conectores para Estantería Metálica. Cuenca, Ecuador: Universidad Politécnica Salesiana

[7] Ferdinand P. Beer, E. R. (2004). Mecánica de Materiales. México, D.F.: Mc Graw Hill Interamericana

[8] Firl, M. (2010). Thesis: Optimal Shape Design of Shell Structures. München, Alemania: Technischen Universität München

[9] González, S. G. (2016). SolidWorks Simulation. México: MARCOMBO

BIOGRAPHIES

María Angélica Rodríguez Carmona Engineering Master Bachelor student of Tlaxcala Polytechnics University, author of four papers in the RED Tes Journal of Mexico.

David Gallardo García Strategic Planning and Technology Direction Ph. D. Full time Professor Researcher of the Tlaxcala Polytechnics University, quality management systems consultant, desirable Profile before PRODEP, member of ASQ and 14 years of experience in various automotive and manufacturing industries.

Jacobo Tolamatl Michcol Strategic Planning and Technology Direction Ph. D. Full time Professor Researcher of the Tlaxcala Polytechnics University, quality management systems consultant, green belt certified, leader of Industrial Engineering Academic Body.

Román Daniel Romero Mitre Engineering Master Bachelor, Professor Researcher of the Tlaxcala Technological University, 2017-2020 Desirable Profile before PRODEP.