International Journal of Emerging Research in Management &Technology
ISSN: 2278-9359 (Volume-6, Issue-4)

Research Article | **April 2017**

Resistive Analysis with Solidworks of an Element of a Conveyor Belt

Johann Mejías Brito[1], Idalia Isla Vilachá[2], Jesús F. Tejeda Castrejón[1], Pedro I. Silva Medina[1], Olimpo Lúa Madrigal[1]
[1]Electrical and Electronic Engineering Department/ Instituto Tecnológico de Colima, México
[2]Faculty of Engineering / Universidad de Holguín "Oscar Lucero Moya"

Abstract—

This paper presents the results of modeling and simulation, related to an element of a transport machine. For this study, 3D modeling with SolidWorks 2015 software was performed, the static analysis of the element to be studied was performed using computer-aided engineering techniques, using the SolidWorks Simulation finite element analysis kit. Derived from the analysis we obtain the graphical views of the distribution of tensions and numerical values of the calculated parameters, which allows to propose solutions to eliminate or reduce this problem in the sugarcane industry.

Keywords— resistive analysis, computer aided design; MEF; static analysis; SolidWorks Simulation

I. INTRODUCTION

The sugar industry in Mexico is one of the activities with greater tradition and importance in the economic development of the country (Aguilar et al., 2011, Domínguez et al., 2014). Sugarcane is the second crop in importance, surpassed only by maize (Campos & Oviedo, 2013). The industrialization of this sector is carried out by sugar mills, which through various machine and equipment systems are responsible for transforming the raw material into a finished product. As described in Mejias et al., (2016), the increase in milling capacity in the mills caused the need to analyze the production process to improve its efficiency, which leads to design modifications in different mechanical drives.

As observed by Rodríguez et al. (2006), the mill and conveyor shafts and shafts are highly stressed critical elements that are subject to considerable loads, causing their rupture in a relatively short number of work cycles, which is a high cost for any sugar mill. Similarly Gómez, et al. (2005) corroborated that faults in these machine elements arise as a result of fatigue, and in many cases, the synergy between this and the corrosion caused by sugarcane juice. Among the most frequently detected faults in sugar mills is the fracture of the conveyor belts. This happens to a greater extent by the increase of the movement of the quantity of material; This affectation generates frequent stops of the machinery, causing losses in the productivity of the milling.

To solve these deficiencies and satisfy the need for new technologies that, in accordance with Aguilar (2014), allow to guarantee the food security provided by this basic carbohydrate, as well as to address productivity challenges in the face of climate change; In this work the resistive analysis with SolidWorks of a tree of a bagasse transporter was carried out that had to be replaced because of the fracture of the same one. The results of this study allow the virtual analysis of the behavior of this element under the loads it supports during its operation and propose solutions that solve this problem.

II. MATERIALS AND METHODS

An In this paper we used Computer Aided Design (CAD) tools, which allowed the modeling of the element to be analyzed, as well as the Finite Element Method (FEM). This last method, according to Pérez & Rodríguez (2014), is a process that consists of a logical sequence of work, which can be divided into several stages. In order to develop this study, elements of the methodologies proposed in the work of several authors were taken into account (Estrada and Gómez, 2012, Estrada et al., 2013, Ocampo et al., 2014, Leslie & Águila, 2015, Llanes et al. 2015).

The 3D modeling was performed with CAD software SolidWorks 2015. The defined model comprises a detailed 3D representation of the object tree (Figure 1), with the inclusion of all the constructive details of the tree. In the same way, the properties of the materials that make up the structure of the same, the boundary conditions, the fixtures and loads are declared, and finally the meshing of the element to be analyzed is carried out. Static analysis was performed using computer-aided engineering techniques, using the SolidWorks Simulation finite element analysis palette.

For determination of the stress distribution equivalent (σ_{eq}) the Von Mises criterion is applied to determine the most dangerous area of the device, which is given by the expression:

$$\sigma_{von\ Mises} = \sqrt{\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_1-\sigma_3)^2}{2}} \qquad (1)$$

σ_1, σ_2 y σ_3: Principal stresses.

Determining the safety factor from obtaining the equivalent stresses for the most dangerous area and limits or yield stress according to the type of material used for construction of the device is performed according to the following expression:

$$n = \frac{\sigma_{lim}}{\sigma_{eq}} \tag{2}$$

σ_{lim} : Allowable stress.
σ_{eq} : Equivalent stresses.
For which it must meet the following condition of strength:

$$\sigma_{eq} \leq \sigma_{\lim} \tag{3}$$

Fig. 1 Physical device model with SolidWorks.

The element to be studied can be observed in Fig. 1, it receives the movement and power of a power unit and has to reduce the speed at which the movement of the mat demands. On the tree are located 4 wheels called bagaceras that have 13 teeth and pass 8 inches, on them go the bagaceras chains. The material is steel AISI 1045, has good machinability in normalized as well as the hot rolled condition. Based on the recommendations given by the machine manufacturers, operations like tapping, milling, broaching, drilling, turning and sawing etc. can be carried out on AISI 1045 steel using suitable feeds, tool type and speeds. The mechanical properties of this material can be seen in Table 1.

Table I Mechanical Properties of AISI 1045 steel.

Material	Modulus of Elasticity	Poissons Ratio	Tensile Strength
Steel 1045	205 000 MPa	0.29	530 MPa

III. RESULTS AND DISCUSSIONS

The geometric documentation of the machine element object of study was made, submitting the complete model to the proposed analysis, derived from this the graphical views of the distribution of tensions, deformations, displacements, safety coefficients, etc., animations, as well as the results of the deformations, displacements (mm) and Von Mises tensions (MPa).

When analyzing the extreme tensions it can be observed that the maximum tensions reaches 339.6 MPa (Fig. 2), it can be observed that they are below the elastic limit of the steel AISI 1045 that is of 565 MPa. It can therefore be inferred that the belt conveyor shaft will withstand the loads to which it is subjected.

In the case of static displacement, a maximum of 3.6 mm is obtained in the red area, as shown in Fig. 3, which indicates the critical region of the same. All materials in the model obey Hook's law where stresses are directly proportional to deformations. The result generated by the simulation is observed in Fig. 4.The design check is made when the material, shape and dimensions have already been preset and it is necessary to know if they are adequate to resist the state of the acting loads. This last step is conclusive to determine the accuracy of the results previously performed. The analysis of the modifications introduced to the structure was carried out, the cylinder diameter was increased by 10 mm and the dies were increased by 10 mm, as well as the radios of agreement and bevels, obtaining a safety factor of 1.36 for the new Design, which assures a reliable behavior of the same during the appearance of extreme loads in conditions of operation of the machine.

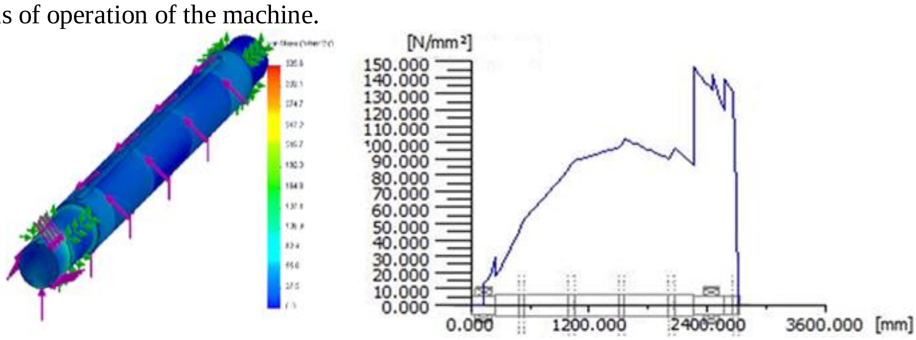

Fig. 2 Equivalent Stresses.

The Figure 5 shows the results of the analysis of the safety factor, obtaining a safety coefficient of 1.36 for the new design, which assures a reliable behavior of the same during the appearance of extreme loads under conditions of exploitation of machine. Finally derived from the results of the studies the tree was redesigned with the optimal dimensions to support the loads that this element of the machine is subjected during the production process in the sugar mill.

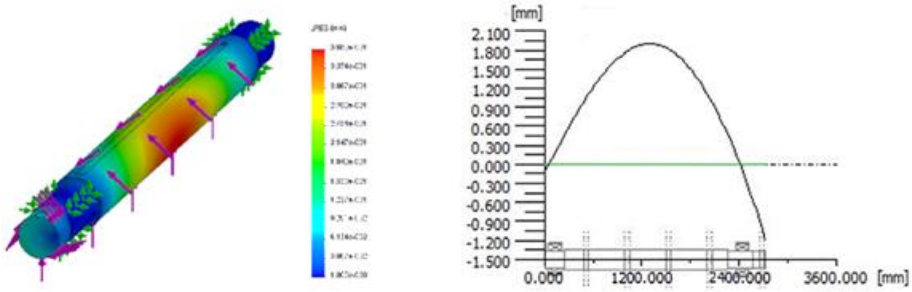

Fig. 3 Static Displacement.

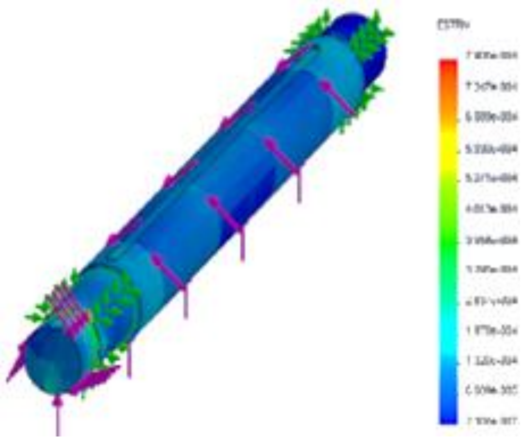
Fig. 4 Unit Deformations.

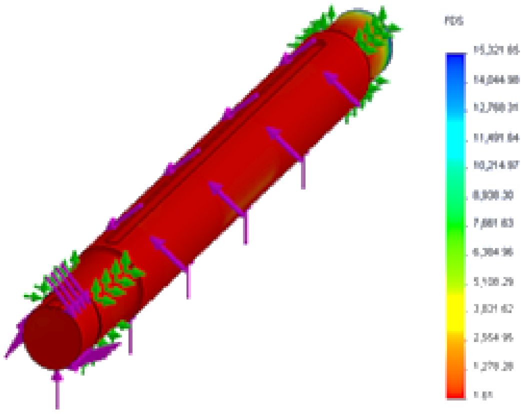
Fig. 5 Security Factor.

IV. CONCLUSIONS

The The geometric documentation of the web conveyor tree was performed using the SolidWorks parametric software.

Static analysis of the machine element under study was performed with the SolidWorks Simulation finite element analysis package.

The study made it possible to offer an alternative solution to the problem proposed as a result of increasing the diameter of the cylinder by 10 millimeters and 10 millimeters to the trunnions, in addition to making the radii of agreement and the bevels, maximum reference to 146.31 MPa.

The modifications made in the design of the tree allowed to establish a safety factor of 1.61, a value that is within the limits permissible so it can be said that the tree resists the loads to which it will be subjected.

REFERENCES

[1] Aguilar, N. (2014). Reconversión de la cadena agroindustrial de la caña de azúcar en Veracruz México. Nova scientia, 6(12), 125-161.

[2] Aguilar, N., Galindo, G., Fortanelli, J., & Contreras, C. (2011). Factores de competitividad de la agroindustria de la caña de azúcar en México. Región y sociedad, 23(52), 261-297.

[3] Campos, F., & Oviedo, M. (2013). Estudio sobre la competitividad de la industria azucarera en México (No. 2013-16). Working Papers, Banco de México.

[4] Domínguez, C. Á., Sosa, R. & Bravo, H: (2014). Prevención, minimización y control de la contaminación ambiental en un ingenio azucarero de México. Ingeniería, investigación y tecnología, 15(4), pp. 549-560

[5] Estrada, R., & Gómez, E. (2012). Análisis del bastidor principal de la cosechadora de caña a través del Método de los Elementos Finitos.//Analysis of the main frame in cane combine-harvester using the Finite Elements Method. Ingeniería Mecánica, 6(3), 45-52.

[6] Estrada, R., Aballes, C. R., & Pérez, R. (2013). Análisis dinámico del mecanismo paralelogramo del cortacogollo desfibrador para la cosechadora cañera cubana CCA-5000. Revista Ciencias Técnicas Agropecuarias, 22(1), 05-11.

[7] Gómez, J. A., Gómez, A. L., & Coronado, J. J. (2005). Análisis del comportamiento a fatiga de un acero AISI 1045 en un ambiente de jugo de caña de azúcar. Scientia et Technica, 1(27). pp. 81-86.

[8] Leslie, J. F. M., & Águila, M. V. G. (2015). Análisis del comportamiento resistivo del órgano de trabajo del Multiarado MAU-250. Acta Agronómica, 64(1), 22.

[9] Llanes, R. V., Cingüalbres, R. A. E., & Dubras, H. C. (2015). Estudio pseudodinámico de la viga de giro de la cosechadora de caña cubana CCA-5000. Revista iberoamericana de ingeniería mecánica, 19(1), 59-70.

[10] Mejías, J., Silva, P., Lugo, R., Martínez, J., Tejeda, J. (2016) International Journal of Engineering, Business and Enterprise Applications, 18(1), 01-05.

[11] Ocampo, A., Hernández, L. H., Fernández, D., Fernández, D., & Cervantes, R. (2014). Análisis estático de los esfuerzos y deformaciones de la estructura de un Invernadero tipo Ventila Cenital ubicado en Veracruz. Revista Ciencias Técnicas Agropecuarias, 23(4), 10-16.

[12] Pérez, Y. C., & Rodríguez, R. F. (2014). Diseño de los acoplamientos cuadrados de hierro fundido para molinos de caña mediante el método de los elementos finitos. Revista Facultad de Ingeniería, (41), 120-131.

[13] Rodríguez, S., Coronado, J. J., & Arzola, N. (2006). Predicción de vida remanente en ejes de masa superior de molino de caña. Ingeniería e Investigación; Vol. 26, núm. 1; 84-91.

