Article Journal Computational Simulation
June 2023, Vol.7 No.17 9-16

Mechanism validation after stress concentration analysis mathematical calculated with safety factor requirements using dedicated software with friction factor mate

Validación de mecanismo después del cálculo matemático análisis de concentración de esfuerzos con requerimientos de factor de seguridad utilizando software dedicado por factor de fricción

BRIANZA-GORDILLO, Gerardo†*, ZAMARRIPA-MUÑOZ, Miguel Ángel and HERRERA-PIAD, Luis Alejandro

Universidad Tecnológica de Aguascalientes

ID 1st Author: *Gerardo, Brianza-Gordillo* / **ORC ID**: 0000-0002-9384-643X

ID 1st Co-author: *Miguel Ángel, Zamarripa-Muñoz* / **ORC ID**: 0000-0002-4589-1455

ID 2nd Co-author: *Luis Alejandro, Herrera-Piad* / **ORC ID**: 0000-0002-6204-0193

DOI: 10.35429/JCS.2023.17.7.9.16 Received: January 15, 2023; Accepted June 30, 2023

Abstract

It is feasible to use Computer Aided Design (CAD) and Finite Element Analysis (FEA) numerical simulation to validate the mathematical results obtained. Testing mechanisms in real world situations, prior to manufacturing, results in optimal designs and more reliable products. Simulation software tools evaluate the behavior of a system, improve the quality of data interpretation, and even increase product innovation. The present work shows the calculation of a simple mechanical system in two dimensions, involving the mechanical properties of the materials used, obtaining the maximum allowable load due to a required safety factor. The behavior of a mechanical element while in a stress concentration is shown along with the results obtained mathematically and with the dedicated software. Once the validity of the theoretical behavior (simulation) is known, the original design will be submitted showing the assembly with non-coincident meshing, the results obtained by the friction factor, the ISO clipping showing the volumes involved in a real situation, and the acquirement of the safety factor. The designer is shown a reliable method for decision making in the development of new equipment, modifications or even changes in the geometries and materials involved.

Analysis, Calculation, Motion study, Interpretation

Resumen

Es factible el uso del diseño asistido por computadora CAD y la simulación numérica FEA para la validación de los resultados matemáticos obtenidos. Probar mecanismos en situaciones del mundo real, antes de la fabricación, da como resultados óptimos diseños y productos más fiables aun con el costo que implican los prototipos. El presente trabajo muestra el cálculo de un sistema mecánico simple en dos dimensiones, se involucran las propiedades mecánicas de los materiales utilizados, la obtención de la carga máxima admisible debido a un factor de seguridad requerido. Se muestra el comportamiento de un elemento mecánico en su concentración de esfuerzos los resultados obtenidos matemáticamente y con software dedicado. Al conocer la validación del comportamiento teórico (simulación), se someterá el diseño original mostrando el conjunto con mallado no coincidente, los resultados obtenidos por sujeción con factor de fricción, los recortes de trazo ISO mostrando los volúmenes involucrados de la situación real y la obtención del factor de seguridad. Se muestra al diseñador un método fiable para la toma de decisiones en el desarrollo de nuevos equipos, modificaciones incluso cambios en las geometrías y materiales involucrados.

Análisis, Calculo, Estudio de Movimiento, Interpretación

Citation: BRIANZA-GORDILLO, Gerardo, ZAMARRIPA-MUÑOZ, Miguel Ángel and HERRERA-PIAD, Luis Alejandro. Mechanism validation after stress concentration analysis mathematical calculated with safety factor requirements using dedicated software with friction factor mate. Journal Computational Simulation. 2023. 7-17: 9-16

† Researcher contributing as first author.

Introduction

A great number of mechanisms and mechanical systems that have practical utility in the engineering field can be analyzed considering uniplanar motion. For the resolution of related problems even in the industrial field, it is necessary to have the mastery of vector algebra and skill in handling the basic concepts of particle kinematics, as well as the fundamental principles of rigid body motion where the dimension between the kinematic pairs of the mechanism under study is kept constant (Alcedo, 2018).

In a machine the synchronization of its mechanisms is critical, with the study of motion, the velocity analysis of the trajectories of interest involves defining how fast certain points travel over the links of a mechanism as a function of time, it is necessary to know all the velocities in the mechanism or machine, both for the calculation of stored kinetic energy and to determine the accelerations of the links required for the calculation of the dynamic forces that turn out to be of importance in the productive system (Orozco, 2022).

There are many approaches for the kinematic analysis of machines and mechanisms, there are graphical, algebraic, vector and matrix methods (MESA P. & DURANGO I., 2005). The graphical technique for the kinematic analysis presented in the analysis of the mechanism shown here is focused on this particular mechanism.

In the case of complex mechanisms it is desirable a general method that allows to solve the kinematic analysis in a clear and simple way (MESA P. & DURANGO I., 2005). SolidWorks Simulation is a tool available for motion study users, it provides a complete set of structural analysis capabilities to guide design decisions and improve product performance and quality, it also offers cloud-enabled solutions (Dassault Systemes SolidWorks Corporation, 2022).

An overview on the concept of optimization in the engineering field is frequently employed for a given task to enable an accurate and reliable decision on changes or adjustments made to a mechanical system, which in turn allows not losing sight of the importance of the system's power consumption.

It could also be optimized according to other parameters such as maximum deformation, maximum range, etc., in which case the dimensions of the mechanism should be optimized by modifying the thickness, length of the links, construction material or a combination of all of them. Therefore, optimization is always done with respect to one or more geometrical variables.

Methodology

Proposed solution to a mechanical system:

The following mathematical solution situation is presented for the mechanical system shown in Figure 1. Taking into account that each of the links AB and CD are connected to a support and to the ECB element by means of steel pins of 25.4 mm diameter acting in single shear. It is known that the ultimate shear stress for the steel used in the pins is 324 MPa, taking into account that the shear stress is created by the tangential stresses with respect to the part directrix. The ultimate normal stress for the steel used in the links is 579 MPa, the equivalent of a carbon steel casting. The allowable load P is determined considering a safety factor 3.

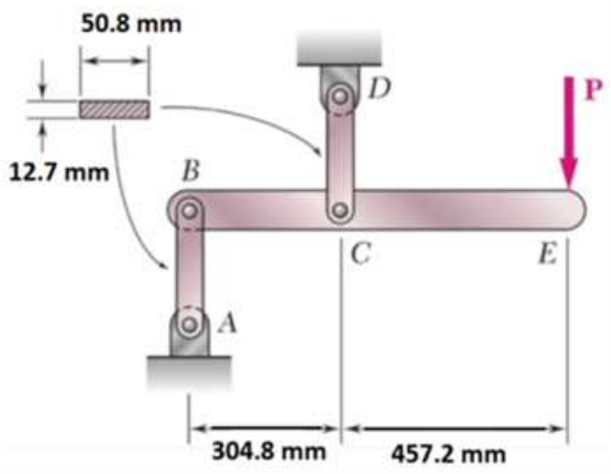

Figure 1 Schematic of the mechanical system to be solved
Source: Own Elaboration based on (BEER, 2010)

Considering that each of the links AB and CD are connected to a support and to the ECB element by means of steel pins of 25.4 mm diameter acting in single shear, we have that:

Data

The dowel diameter (d) = 25.4 mm
Ultimate shear stress for dowels (Tu) = 324 MPa
The ultimate normal stress for the links (σu) = 579 Mpa

Link width (b) = 51 mm FACT
Link height (h) = 12.7 mm FACT

The components of the tangential stresses, normal to the edge in intersection planes, of two orthogonal planes are equal in absolute value, and their directions are such that both are directed towards the edge, we can deduce the property: "The shear stress at a point on the contour is tangent to it" (Gutierrez, 2022). The ultimate shear stress (Tu) for the pins is related to the area and the ultimate load, we say that:

$$\tau_u = \frac{P_u}{A} \tag{1}$$

Where:

τ_u *is the ultimate shear stress in the pastr* (Pa)

P_u *is the ultimate load for the pin* (N)

A *is the cross section of the pin* (m^2)

We know that the area of the pin $A = 5.067\ x\ 10^{-4}\ m^2$

Therefore, from equation (1) we obtain that $P_u = 164{,}584\ N$ Considered as the ultimate load for the pin.

In the literature of mechanics of materials and structural design, the ultimate normal stress is mainly based on deformation theory and experimental tests of structural elements, although some probabilistic methodologies assume the existence of the normal stress without specifying its mode of determination (Molina, 2019). The ultimate normal stress (σ_u) for the links is also related to the area remaining from the bolt support (considered the smallest in tension) and the ultimate load, we have that:

$$\sigma_u = \frac{P_u}{A} \tag{2}$$

Where:

σ_u *is the ultimate normal stress in the link*

P_u *is the ultimate load for the link* (N)

A *is the critical trans. sec. of the link* (m^2)

We know that the critical area of the link:

$$A_c = bh - dh \tag{3}$$

And from equation 3 we obtain:

$$A_c = 3.226\ x\ 10^{-4}\ m^2$$

So from equation (2) $P_u = 186{,}825\ N$ considered as the ultimate load for the link.

From here the ultimate load for the pin can be considered as the load that will drive the design, since it is less than the ultimate load for the link.

Figure 2 shows the free body diagram for the analysis of the applied load on each element with respect to the ultimate load considered.

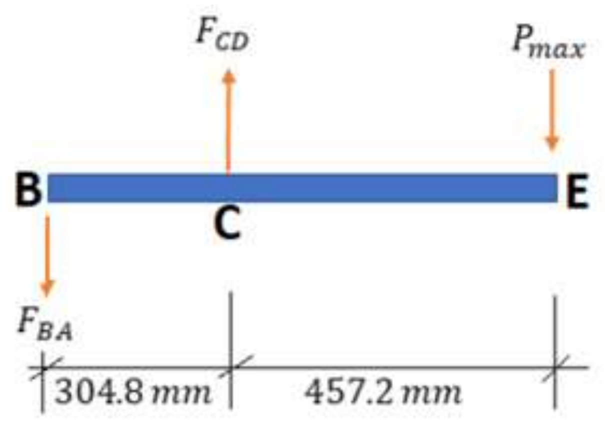

Figure 2. Free body diagram of the mechanical system.
Source: Own Elaboration

The sum of moments of the free body diagram of the member BE is considered in order to find a relationship between F_{CD} and P_{max} and between F_{BA} and P_{max}

For the sum of moments with respect to point C we have that:

$$\sum M_C = 0 = F_{BA}(0.3048\ m) - P_{max}(0.4572\ m) = 0 \tag{4}$$

And from equation (4) we obtain that: $F_{BA} = 1.5\ P_{max}$

For the sum of moments with respect to point B we have that:

$$\sum M_B = 0 = F_{CD}(0.3048\ m) - P_{max}(0.762\ m) = 0 \tag{5}$$

And from equation (5) we obtain that: $F_{CD} = 2.5\ P_{max}$

ISSN 2523-6865

Therefore, the CD busbar is considered as the critical busbar in the design according to F_{CD}. Taking into account the ultimate load for the pin we have that: The maximum allowable load for the design will be:

$$P_{adm} = \frac{164584N}{2.5} = 65{,}833\ N$$

And with the required Safety Factor:

$$FS = \frac{P_{max}}{P_{admp}} \quad (6)$$

Then the allowable load for the pin, with a Factor of Safety of 3: $P_{admp} = 21{,}944\ N$ Analogously, taking into account the ultimate load for the link we have that:

The maximum allowable design load shall be:

$$P_{adm} = \frac{186{,}825\ N}{2.5} = 74{,}730\ N$$

And with the required Safety Factor:

$$FS = \frac{P_{max}}{P_{adme}} \quad (7)$$

Then the allowable load for the link with a Factor of Safety of 3:

$$P_{adme} = 24{,}910\ N$$

Validation and analysis with dedicated software

To analyze complex geometries, computational calculation methods have been developed. For the calculation of solid assemblies, the Finite Element Method (FEM). Founded on the transformation of continuous systems into discrete systems, i.e. the division of the real structure into small substructures (elements) of finite character, which are joined by means of nodes (Mulas, 2019). In machine components the stress concentration is important, elements exposed to the action of one or more forces in its operation, should be considered for its good performance the behavior of the stress concentration factor k which is the ratio of the maximum stress and the average stress calculated in the reduced section of the geometric discontinuity known as critical (Acosta, 2014).

Figure 3 shows the mathematical analysis of the maximum stress obtained in a rectangular steel plate with centered hole AISI 1020 subjected to axial tension load.

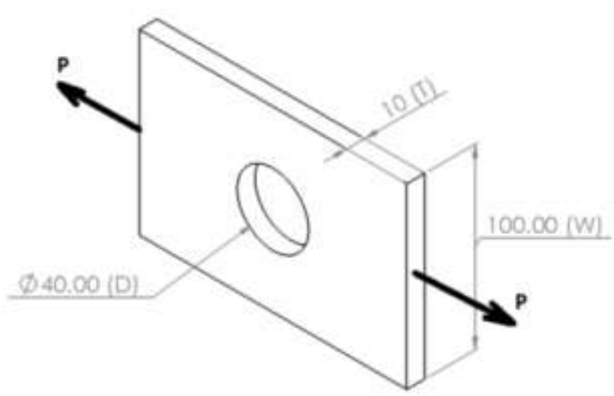

Figure 3 Centered-hole steel plate subjected to 110,000 N axial load
Source: Own Elaboration

A measure of the degree of stress concentration is given by the so-called geometric stress concentration factor k , which is defined as the ratio between the maximum local stress ("stress peak") and the corresponding nominal stress (Peru, 2022):

$$k = \frac{\sigma_{max}}{\sigma_{nom}} \quad (8)$$

Geometric nominal stress σ_{nom} towing to two stress effects: the increase due to the decrease in cross-section and the increase due to geometry (Peru, 2022):

$$\sigma_{nom} = \frac{P}{(W-D)T} \quad (9)$$

The stress concentration factor depends mainly on the geometry, not on the material except when the material deforms under load. Values of k are usually obtained from graphs and formulas and are strictly valid for ideally elastic stiff members, for brittle and high strength materials they are usually sensitive to even minor scratches hence for $0 \leq \frac{D}{W} \leq 1$:

$$k = 3 - 3.14\left(\frac{D}{W}\right) + 3.667\left(\frac{D}{W}\right)^2 - 1.527\left(\frac{D}{W}\right)^3 \quad (10)$$

From equation (8) and with the geometrical data of figure 3 we have that:

$$\sigma_{nom} = 183.33\ MPa$$

From equation (10) and with the geometrical data of figure 3 we have that $k = 2.23568$

ISSN 2523-6865

Finally, from equation (11) the maximum stress in the plate:

$$\sigma_{max} = 409.87 \, Mpa$$

SolidWorks Simulation comprises a set of structural analysis solutions, which aim to predict the actual physical behavior of a product. Using Finite Element Analysis (FEA), this group of tools makes all the work very easy, using virtual testing of CAD models (Systèmes, 2022). Meshing is a crucial step in design analysis. The automatic mesher in the software generates a mesh based on a global element size, a tolerance and local mesh control specifications. Mesh control allows you to specify different element sizes of components, faces, edges and vertices (Systèmes, 2022). The FEA analysis of the calculated plate is shown below. Figure 4 shows the initial static analysis including restraint, applied load.

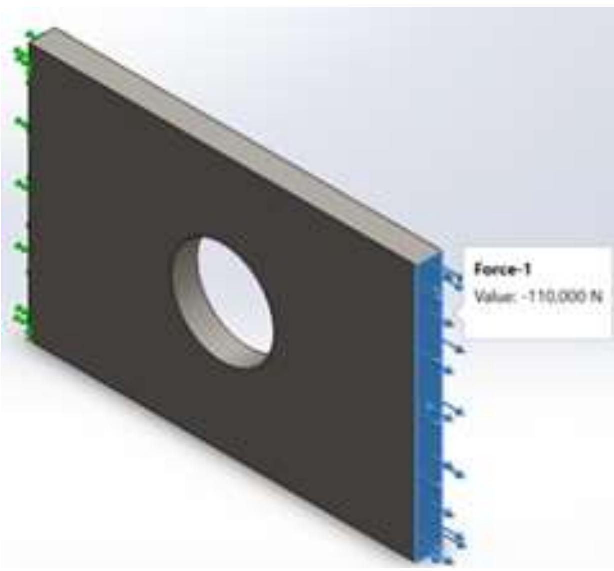

Figure 4 AISI 1020 steel plate with ultimate tensile stress of 420 MPa subjected to axial load of 110,000 N
Source: Own Elaboration

Figures 5, 6 and 7 show the fine, normal and coarse meshes showing the maximum stress value obtained from the FEA simulation study.

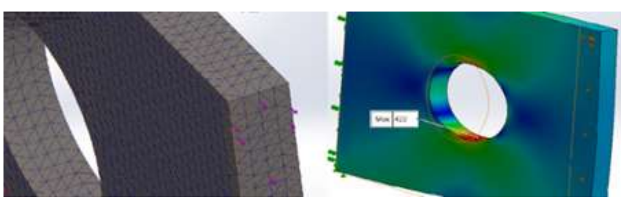

Figure 5 Maximum stress value (422 MPa) with simulation using fine meshing
Source: Own Elaboration from simulation

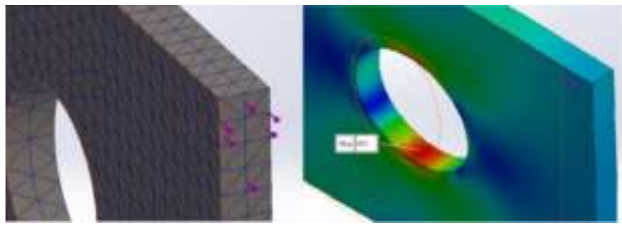

Figure 6 Maximum stress value (423 MPa) with simulation using normal meshing
Source: Own Elaboration from simulation

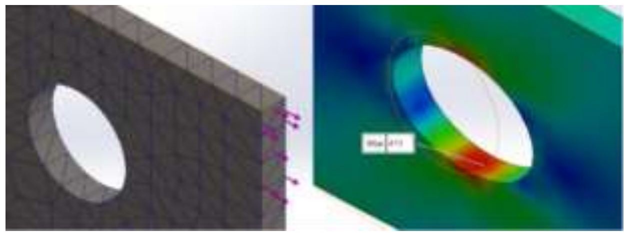

Figure 7 Maximum stress value (410 MPa) with simulation using coarse meshing
Source: Own Elaboration from simulation

The Von Mises maximum stress criterion is based on the Von Mises-Hencky theory, also known as the shear energy theory or the maximum distortion energy theory, provides a good idea in the design in determining that the element in question for its final use will or will not be strong enough. A SOLIDWORKS Simulation study allows us to understand the behavior of forces that will be applied in real life prior to production. Performing a proper finite element analysis of a system can drastically reduce prototyping time and provide design validation or justification for changes and adjustments. The most common indication of this is a Von Mises stress value (Systèmes, 2022). Table 1 shows the comparison of the results obtained by the dedicated software from the mathematical one obtained with mesh density.

Mesh density	Maximum Stress Von Mises (MPa)	Maximum Stress per Concentration Factor (MPa	Fluctuation percentage (%)
Fine	422	409.87	2.87
Normal	423	409.87	3.10
Coarse	410	409.87	0.03

Table 1 Comparison of values obtained in the simulation
Source: From values obtained

The criteria that can be obtained from the results presented in Table 1 will depend largely on the designer's needs. The Finite Element Method (FEM) predicts the behavior of the model by combining the information obtained from all the elements that make up the model. Meshing is a crucial step in the design analysis.

The automatic mesher in the software generates a mesh based on a global element size, a tolerance and local mesh control specifications. Mesh control allows you to specify different element sizes of components, faces, edges and vertices. In the early stages of design analysis where approximate results may be sufficient, however, the designer can specify a larger element size (coarse meshing) for a faster solution, if the need is for a more accurate solution, a smaller element size will be necessary.

For the mechanism proposed in the original study (see Figure 1), Figure 8 shows the model in question of only the parts in contact, bar and pin in single shear with the materials assigned with respect to the proposed ultimate stress value.

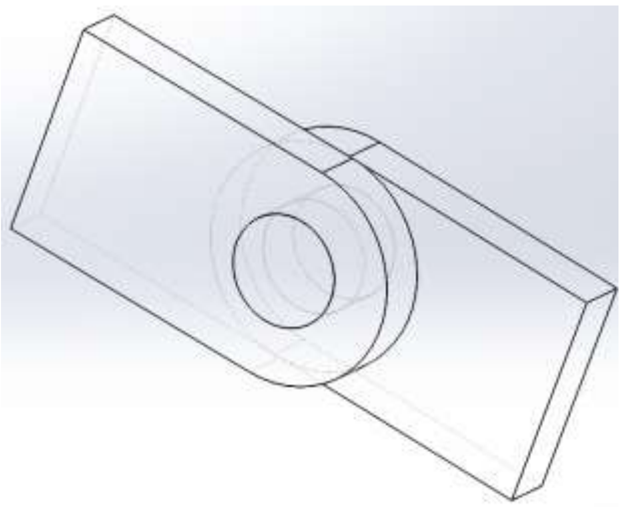

Figure 8 Assembly of the contacting parts of the elements subjected to load in the mechanism
Source: Own Elaboration based on the original model

Assembly by coefficient of friction

It is clear that in the assembly shown in Figure 8 the element is simply supported on the bolt surface and allows its mobility. Performing a geometric assembly does not allow to visualize the real behavior when the elements are in tension and shear. SolidWorks allows to include global friction. The software calculates the static friction forces by multiplying the normal forces generated at the contact locations by the specified friction coefficient. The coefficient of friction should be between 0 and 1.0 (Serway, 2008). A coefficient of friction Steel on iron of 0.74 is taken (Serway, 2008). Figure 9 shows the assignment of the coefficient of friction to the assembly.

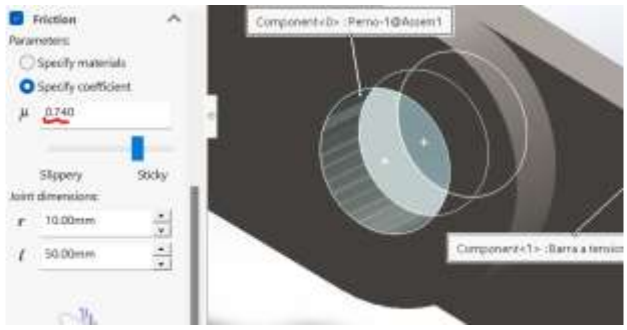

Figure 9 Assembly conditions by friction coefficient
Source: Own Elaboration from the original model

Incompatible meshing

In an incompatible mesh, the entities in contact of the assembly are meshed in such a way that there is no node to node correspondence between the meshes of each entity. Nodes that correspond piece by piece cannot be merged (for a rigid joint contact) or overlapped. The results obtained with an incompatible mesh more closely resemble reality where each of the elements behaves independently in the study according to its own function. Fig. 10: shows the incompatible mesh for the study of the assembly.

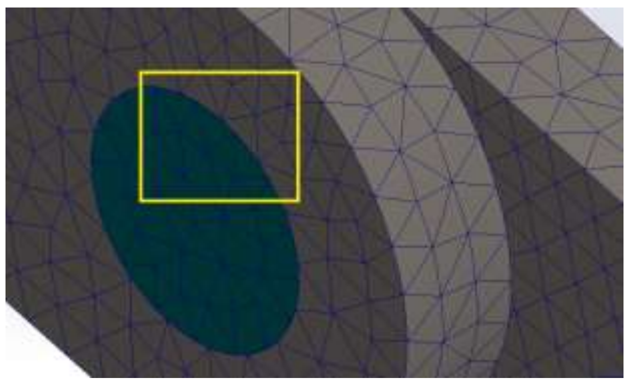

Figure 10 shows the incompatible meshing that is performed for the study of the set
Source: Own Elaboration from the original model

Testing, validation and safety factor

When loads are applied to a solid, the solid deforms and the effect of the loads is transmitted through the solid. External loads induce internal forces and reactions to render the solid to an equilibrium state. Linear static analysis calculates displacements, unit deformations, stresses and reaction forces under the effect of applied loads (Serway, 2008). Figure 11 then shows the assembly in tension situation applying the maximum allowable load for the design on the single shear pin.

ISSN 2523-6865

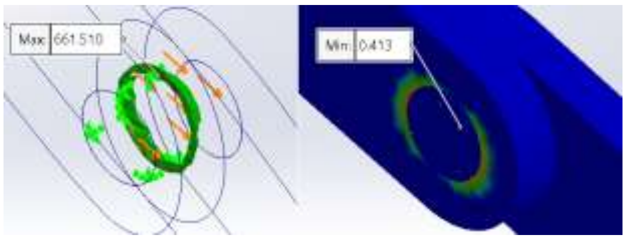

Figure 11 Element loaded to 164584 N. It shows a maximum load of 661 MPa and a Factor of Safety of 0.41
Source: Own Elaboration based on the simulation.

Since the safety factor (equation (6)) is the ratio of the maximum stress to the allowable stress, a safety factor less than 1 indicates that the element will fail.

Figure 12 shows the assembly in tension situation applying the maximum allowable load for the critical bar link CD of 65,677N, (equation (5)).

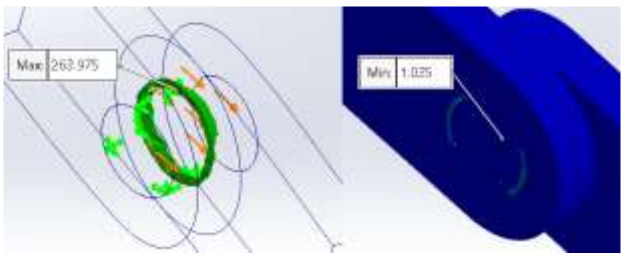

Figure 12 Element loaded at 65,677 N. It yields a maximum load of 263 MPa and a Factor of Safety of 1.03.
Source: Own Elaboration from simulation.

Figure 13 shows the assembly in tension situation applying the maximum admissible load for the pin of 21,894 N, due to the Safety Factor considered.

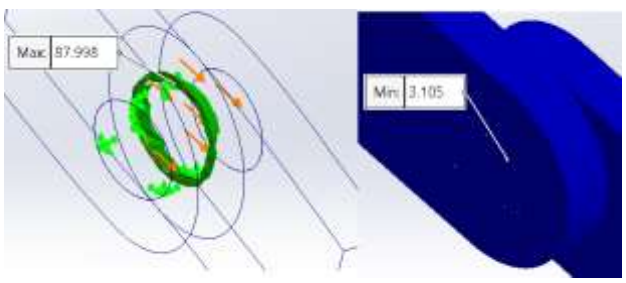

Figure 13 Element loaded to 21,894 N. It shows a maximum load of 88 MPa and a Factor of Safety of 3.1:
Source: Own Elaboration from simulation

Figure 14 shows the ISO cutout that allows to visualize by volume percentage the support with respect to the applied load of the link on the shear bolt.

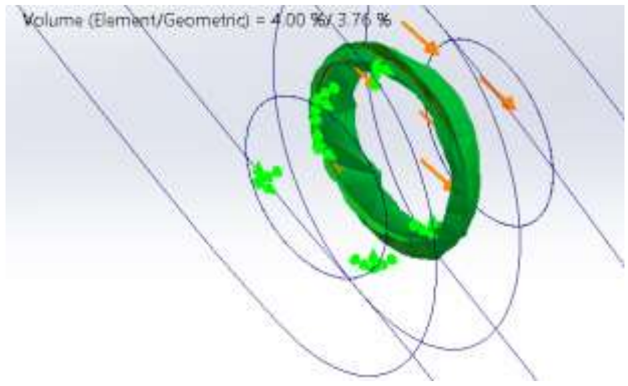

Figure 14 ISO trace cutout of the test element
Source: Own Elaboration from the simulation

Figure 15 shows the ISO trace cutout that allows visualizing by volume percentage the support with respect to the applied load of the bolt on the link.

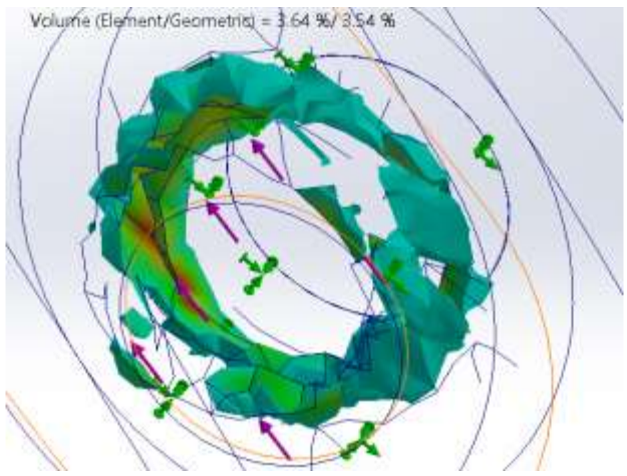

Figure 15 ISO trace cutout of the test element
Source: Own Elaboration from simulation

Results

It is interesting to be able to verify that, from the mathematically obtained calculations for the mechanical behavior of a mechanism with respect to the strength of the materials and the behavior of the solid mechanics, we have within reach dedicated software packages, which allow us to corroborate in a simple way the obtained results.

This gives the reliability in making decisions for modeled designs that result in higher complexity. With the help of SolidWorks software, the safety factor mentioned in the original mechanism is obtained. The analysis of the steel plate with centered hole is taken into account in order to generate the cofinance and reliability of the obtained graphic results.

ISSN 2523-6865

Conclusions

It is important to consider that there is more than one simulation package for finite element, the one to be used will depend on the experience and certification level of the designer. Although the safety factor may vary with respect to the simulation load, it is important to consider that fine meshing offers results with more significant figures, but will require more computer resources. In addition, we should take into account that the proportionality limit is wider in ductile materials than in harder materials, so the value of the maximum stress should be considered for the determination of the allowable stress for the type and mechanical properties of each material or materials involved in the simulation.

References

ACOSTA LANDÍN, R. Y. (2014). Cálculo del factor de concentración de esfuerzos utilizando Solidworks. *repositorio académico digital.* Universidad Autónoma de Nuevo León, Monterrey , Nuevo León, México. doi:ISSN 2395-9029

FERDINAND P. BEER. E. RUSSELL JOHNSTON, J. (2010). *Mecánica de materiales*, México, D. F: Mc Graw Hill. ISBN-13: 978-607-15-0263-6 URL: https://shorturl.at/atLZ7

GUTIÉRREZ, R. I. (2022). Esfuerzo cortante. docplayer. Universidad de Cantabria. URL: https://shorturl.at/cjHLP, DOI:221, 04_ ÚniCo

LUNA, J. M. (2014). Diseño del sistema mecánico de una impresora 3d que utiliza abs en polvo reciclado. *Revista electronica en ingenieria mecanica*, págs. 2 -3. doi:ISSN 1870-1264

MOLINA ALEJANDRO, P.-M. M.-C. (2019). Análisis metodológico del esfuerzo normal $\boldsymbol{\sigma y}$ basado en deflexión elástica. (U. A. Juárez, Ed.) *Revista de Ciencias Tecnológicas (RECIT), 2 No. 4*, 166-180. doi: ISSN 2594-1925

MULAS GONZÁLEZ, D. (2019). Estudio del comportamiento mecánico de piezas realizadas por tecnología FDM mediante la validación de modelos de simulación por Elementos Finitos. *Trabajo Fin de Grado en Ingeniería Mecánica.* Universidad de Valladolid, Valladolid, España. Obtenido de https://uvadoc.uva.es/bitstream/handle/10324/37739/TFG-I-1215.pdf?sequence=1&isAllowed=y

PERÚ, P. U. (diciembre de 2022). *Capítulo 2 concentracion de esfuerzos.* Obtenido de https://bit.ly/3YoHfJa

SERWAY, R. A., & JOHN W. JEWETT, J. (2008). *FÍSICA para ciencias e ingeniería* (Vol. 1). (S. R. González, Ed., & P. f. Engineers, Trad.) México, D.F., ,México: EDITEC S.A. de C.V. doi:ISBN-13: 978-607-481-357-9

SYSTÈMES, D. (1995 - 2022). *Ayuda de SOLIDWORKS. simulación*. Obtenido de https://bit.ly/3e4LZRy