Journal of Architecture and Design 8[19]1-8: e10819108
Article

Structural analysis of a lifting platform for autonomous vertical vehicular parking

Análisis estructural de una plataforma de elevación para estacionamiento vehicular vertical autónomo

Betanzos-Castillo, Francisco*[a], Fuentes-Castañeda, Pilar[b] and Cortez-Solis, Reynaldo[c]

[a] Tecnológico Nacional de México - TES Valle de Bravo • AIE-1532-2022 • 0000-0002-7245-703X • 206209
[b] Tecnológico Nacional de México - TES Valle de Bravo • KUD-2889-2024 • 0000-0001-6567-9614 • 428699
[c] Tecnológico Nacional de México - TES Valle de Bravo • KUD-2900-2024 • 0000-0001-7519-1815 • 1113392

CONAHCYT classification:

Area: Engineering
Field: Engineering
Discipline: Mechanical Engineering
Subdiscipline: Mechanical design

https://doi.org/10.35429/JAD.2024.8.19.1.8
History of the article:
Received: January 17, 2024
Accepted: June 30, 2024

* ✉ [francisco.bc@vbravo.tecnm.mx]

Check for updates

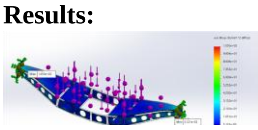
Abstract

A problem that has become latent in urban areas, cities, shopping malls, among others, is the space available for parking cars, coupled with the increase of the vehicle fleet and few spaces established to operate as parking lots. As an alternative solution, the conceptual design of a vertical autonomous parking with a vision oriented to the I4.0 industry was developed. The design, modeling and structural analysis was carried out using computer aided engineering (CAE) of a central platform where vehicles are accommodated upon arrival, this will be raised and finally transferred on a structure divided into different levels, making comparisons according to the configurations and design parameters. Finally, simulations were obtained by means of Finite Element Analysis (FEA), which allowed the realization of different configurations through the use of software to verify if the use of the proposed material is viable for future works in a real way.

Objectives

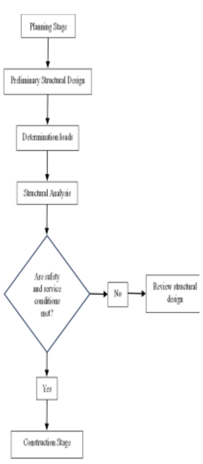
Conceptual Design
Modeling
Structural Analysis
Sustainability
Sustainability

Results:



Contributions:
Development of models of different structural geometric configurations through CAE analysis.

Method



FEA, Simulation, Vertical parking

Resumen

Una problemática que se ha hecho latente en las zonas urbanas, ciudades, centros comerciales, entre otros, es el espacio disponible para estacionar automóviles, aunado al aumento del parque vehicular y pocos espacios establecidos para operar como estacionamientos. Como alternativa de solución se desarrolló el diseño conceptual de un estacionamiento autónomo vertical con visión orientada a la industria I4.0. Se realizó el diseño, modelación y análisis estructural empleando ingeniería asistida por computadora (CAE) de una plataforma central donde los vehículos al llegar son alojados, esta se elevará y finalmente los transferirá sobre una estructura dividida en diferentes niveles, realizando las comparativas de acuerdo las configuraciones y parámetros de diseño. Finalmente, se obtuvieron simulaciones por medio de Análisis por Elemento Finito (FEA), lo que permitió realizar diferentes configuraciones mediante el empleo de software para verificar si es viable el uso del material propuesto para trabajos futuros de manera real.

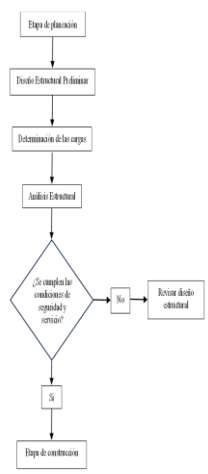
Objetivos

Diseño Conceptual
Modelación
Análisis estructural
Sostenibilidad
Sustentabilidad

Resultados:

Contribuciones:
Desarrollo de modelos de configuraciones geométricas diferentes estructurales mediante análisis CAE.

Método



FEA, Simulación, Estacionamiento vertical

Citation: Betanzos-Castillo, Francisco, Fuentes-Castañeda, Pilar and Cortez-Solis, Reynaldo. [2024]. Structural analysis of a lifting platform for autonomous vertical vehicular parking. Journal of Architecture and Design. 8[19]-1-8: e10819108.

Introduction

Since the middle of the last century, Latin American cities have experienced a rapid growth of their urban centers, which has resulted in a variety of problems such as traffic congestion, environmental pollution, infrastructure deterioration, among others (Rodríguez A. G. y Ramos J. L., 2009). Aiming to mitigate the limitations in the current parking infrastructure and address the global challenge of traffic congestion, propose a vehicle design scheme with vertical lift mechanisms and folding wings, which represents an innovative change for environmental protection, sustainable development and artificial intelligence (Yixu, Junying, & Kun, 2024). Nowadays it is very important to know that urban space is becoming more and more limited, the need for innovative solutions to optimize land use is becoming more and more constant. In this context, conceptualizing vertical parking support structures, in this case, emerges as an efficient and practical response to the growing demand for parking spaces.

Simulation of working machines is a phase of the process that makes it possible to substantially improve performance by facilitating the detection of anomalies not foreseen in the design phase, reducing the amount of real-life testing and also making it possible to analyze and evaluate the safety factor, reducing the probability of generating failures due to fatigue or overload (Galán Ávila, Avendaño Rodríguez, & Villalobos Correa, 2022). For this purpose, a design guideline involving system dynamics, load types and capacities must be defined. A prior strength analysis rule is established, which must be specified and applied (Halicioglu, Canan Dulger , & Tolga Bozdana, 2016).

Finite Element Analysis (FEA) is referred to as Finite Element Method (FEM), which is used in engineering to reduce the number of physical prototypes and run virtual experiments to optimize their designs (Rameshrao Yawale & Nivrutti Naik, 2021) (J.C.TORRES, 1998).

The basic idea of FEM is to divide the complex structure into a finite number of interconnected elements, to determine the loads acting on each node and to calculate the displacements in the direction of those loads, and thus obtain a result for the whole (Oguz Örmecioglu, Aydogdu, & Tugba Örmecioglu, 2024).

This project shows the results of the axisymmetric analysis of a lifting structure for an autonomous vertical parking, simulating the deformation with loads on the structure. The analysis is performed with simulation software, Ansys Workbench and SolidWorks, making a comparison between them to obtain a simulation closer to the real conditions requested at the time of its operation. With this structural analysis, we seek to explore the characteristics, advantages and key considerations associated with this type of structures, with the objective of understanding their performance and their impact in any environment.

The first section contains the project summary with a general description of the research conducted. The second section deals with the introduction of the project, in which the reader will understand the comparative and the main point of analysis. The third section shows the theoretical framework as the main support of the project to know terms and concepts that will be addressed in the CAE simulation, then the fourth section will show the methodology to be applied on the analysis of the structure, materials added to the 3D model, parameters, types of models to be used, meshing and comparisons made, finally the fifth section shows the results and conclusions found.

Methodology

Structural engineering is the science and art of safely and economically planning, designing and constructing structures to serve those purposes. Structural analysis is an integral part of any structural engineering project, whose function begins with the prediction of the behaviour of the structure.

Structural analysis is the prediction of the performance of a structure under prescribed loads and/or external effects, such as support movements and temperature changes. The characteristics of interest in the design performance of structures are (1) stresses or stress results, such as axial forces, shear forces, and bending moments; (2) deflections; and (3) support reactions, therefore, the analysis of structures usually involves the determination of these quantities as the cause of a loading condition.

ISSN: 2531-2162
RENIECYT-CONAHCYT: 1702902

Betanzos-Castillo, Francisco, Fuentes-Castañeda, Pilar and Cortez-Solis, Reynaldo. [2024]. Structural analysis of a lifting platform for autonomous vertical vehicular parking. Journal of Architecture and Design. 8[19]-1-8: e10819108.
DOI: https://doi.org/10.35429/JAD.2024.8.19.1.8

A structural engineering study is described by various stages using a flow chart, this indicates that it is an iterative process, and generally consists of the following steps (Kassimali, 2015):

Box 1

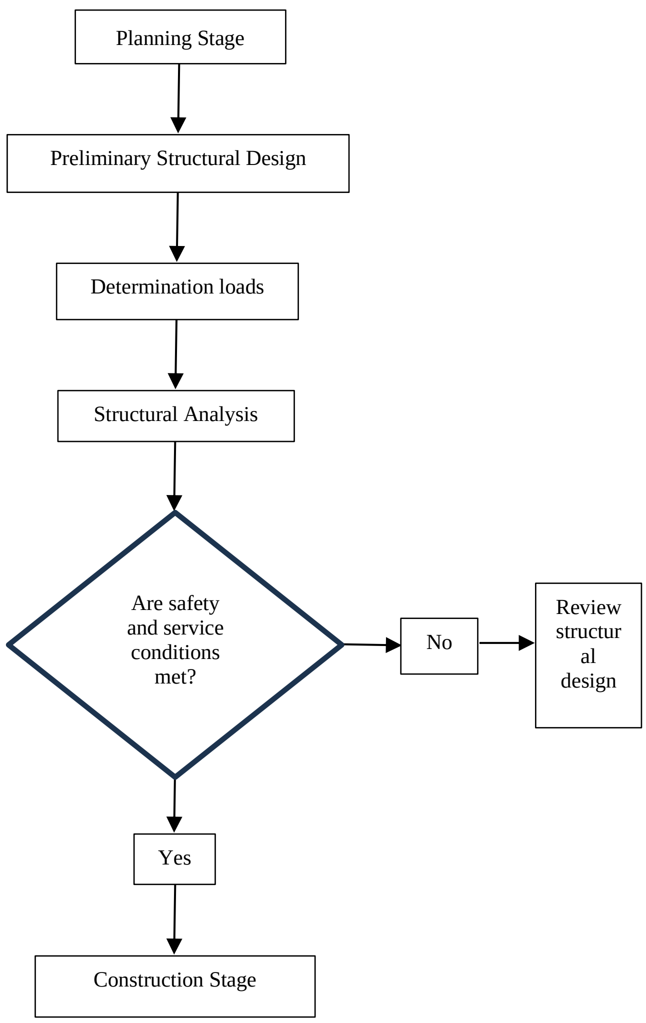


Figure 1
Stages of structural design

Source: adopted from (Kassimali, 2015).

- **Planning Stage**. The planning phase usually involves the establishment of the functional requirements of the proposed structure, the general layout and dimensions of the structure, general considerations of the possible types of structures (e.g., rigid frames or trusses) that may be used, and the types of materials to be used (e.g., structural steel or reinforced concrete).

This stage may also take into account other considerations of non-structural factors, such as aesthetic aspects, environmental impact of the structure and some others.

Its result is generally a structural system that meets the functionality requirements and is expected to be the most economical. This stage is perhaps the most crucial of the entire project and requires experience and knowledge of construction practices, as well as a thorough understanding of the behaviour of structures.

- **Preliminary structural design.** In the preliminary structural design stage, the size of the structural system elements selected in the planning stage is estimated based on an approximate analysis, previous experience and code or regulation requirements. Thus, the sizes of the selected elements are used in the next stage to calculate the weight of the structure.

- **Determination of loads.** Load estimation involves the determination of all loads that can be expected to act on the structure.

- **Structural analysis.** In the structural analysis, the load values are used to develop an analysis to determine the resulting stresses in the elements and the deflections at different points of the structure.

- **Safety and serviceability check.** The results of the analysis are used to determine whether or not a structure meets the safety and serviceability requirements of the design code. If these requirements are satisfied, then the design drawings and construction specifications are executed and the construction phase begins.

- **Structural design review.** If the requirements of the structure are not satisfied, then the element sizes are reviewed, and phases 3 to 5 are repeated until all safety and serviceability requirements are met (Kassimali, 2015).

Mathematical model FEM

Different problems treated in science and engineering are often described in terms of differential equations, formulated by using continuous mechanics models.

ISSN: 2531-2162
RENIECYT-CONAHCYT: 1702902

Betanzos-Castillo, Francisco, Fuentes-Castañeda, Pilar and Cortez-Solis, Reynaldo. [2024]. Structural analysis of a lifting platform for autonomous vertical vehicular parking. Journal of Architecture and Design. 8[19]-1-8: e10819108.
DOI: https://doi.org/10.35429/JAD.2024.8.19.1.8

In general, elasticity problems are reduced to solving the differential equations, known as equilibrium equations together with stress-strain relations or the strain-displacement relations and the compatibility equation under given boundary conditions.

Equilibrium equations in an elastic body in two dimensions:

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + F_x = 0 \quad [1]$$

$$\frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + F_y = 0 \quad [2]$$

Where σ_x and σ_y are normal forces in the *x* and *y* axes respectively, τ_{xy} and τ_{yx} are shear forces acting in the xy plane.

The strain-displacement relationships are:

$$\varepsilon_x = \frac{\partial u}{\partial x} \quad [3]$$

$$\varepsilon_y = \frac{\partial v}{\partial y} \quad [4]$$

$$\gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \quad [5]$$

Where $\varepsilon_x$and ε_y are the normal deformations in the *x*- and *y*-axis directions respectively, the engineering shear deformation in the *xy* plane is γ_{xy}; *u* and *v* are infinitesimal displacements in the *x*- and *y*-axis directions respectively.

Constitutive equations (stress-strain relationships). These relationships describe the state of deformation, deformations induced by internal forces or stresses resisting against applied loads. These relationships depend on the material properties, they are determined experimentally. Hooke's law relates six components of the three-dimensional stress tensors to the strain tensors, as follows:

$$\sigma_x = \frac{\nu E}{(1+\nu)(1-2\nu)} e_v + 2G\varepsilon_x \quad [6]$$

$$\sigma_y = \frac{\nu E}{(1+\nu)(1-2\nu)} e_v + 2G\varepsilon_y \quad [7]$$

$$\sigma_z = \frac{\nu E}{(1+\nu)(1-2\nu)} e_v + 2G\varepsilon_z \quad [8]$$

$$\tau_{xy} = G\gamma_{xy} = \frac{E}{2(1+\nu)} \gamma_{xy} \quad [9]$$

$$\tau_{yz} = G\gamma_{yz} = \frac{E}{2(1+\nu)} \gamma_{yz} \quad [10]$$

$$\tau_{xz} = G\gamma_{zx} = \frac{E}{2(1+\nu)} \gamma_{zx} \quad [11]$$

Or inversely:

$$\varepsilon_x = \frac{1}{E}\left[\sigma_x - \nu\left(\sigma_y + \sigma_z\right)\right] \quad [12]$$

$$\varepsilon_y = \frac{1}{E}\left[\sigma_y - \nu(\sigma_z + \sigma_x)\right] \quad [13]$$

$$\varepsilon_z = \frac{1}{E}\left[\sigma_z - \nu\left(\sigma_x + \sigma_y\right)\right] \quad [14]$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G} \quad [15]$$

$$\gamma_{yz} = \frac{\tau_{yz}}{G} \quad [16]$$

$$\gamma_{zx} = \frac{\tau_{zx}}{G} \quad [17]$$

Where E is Young's modulus, ν is Poisson's ratio, G is the shear modulus and e_v the volumetric strain expressed by the sum of the three normal strain components, $e_v = \varepsilon_x + \varepsilon_y + \varepsilon_z$.

The FEM assumes an object of analysis as a set of elements having arbitrary shapes and finite sizes (called finite element), approximates partial differential equations by simultaneous algebraic equations and numerically solves various elasticity problems (Nakasone, Yoshimoto, & Stolarski, 2006).

The Finite Element Analysis method requires the following main steps:

- Discretization of the domain into a finite number of subdomains (elements).
- Selection of interpolation functions.
- Development of the elementary matrix for the subdomain (element).
- Assembly of the elementary matrices of each subdomain to obtain the global matrix of the complete domain.
- Imposition of the boundary conditions.
- Solution of the equations.
- Additional calculations (if required).

ISSN: 2531-2162
RENIECYT-CONAHCYT: 1702902

Betanzos-Castillo, Francisco, Fuentes-Castañeda, Pilar and Cortez-Solis, Reynaldo. [2024]. Structural analysis of a lifting platform for autonomous vertical vehicular parking. Journal of Architecture and Design. 8[19]-1-8: e10819108.
DOI: https://doi.org/10.35429/JAD.2024.8.19.1.8

Journal of Architecture and Design 8[19]1-8: e10819108
Article

The ability to discretize irregular domains with finite elements makes the (FEA) method a valuable and practical analysis tool for the solution of boundary, initial and eigenvalue problems arising in different engineering disciplines (Madenci & Guven, 2015).

Computational Model

Simulation programs, over time, have improved their analysis, using improvements in the meshing processes, acceptance criteria, implementation of variables and presentation of results, where it is absolutely essential that the user acquires skills and can identify its operation, to have a command of the software (Díaz Iglesias, 2021).

In order to perform simulation and structural analysis, some design software is used, such as: Ansys Workbench® which is an engineering simulation software, provides a wide range of tools and resources for post-processing and visualization of simulation results, allowing to understand and analyze the results effectively (SEMCOCAD, 2024); on the other hand, SOLIDWORKS® is a 3D CAD design software, offers solutions to cover the aspects involved in the product development process from creating, designing, simulating, manufacturing, publishing and managing the data of the design process (SOLIDBI, 2024).

Results

To obtain the structural behaviour of the lifting platform, where the vehicles will be housed and later transferred to their final available location, a bridge type structure was designed, without lateral perforations, square profile 127 mm per side, 12.7 mm thick, 25.4 mm plate, upper and lower central part without plate and lower sides without plate, see figure 2. The creation of the platform geometry was carried out in SOLIDWORKS® and Ansys Workbench®. The geometry consists of the generation of sketches, drawings, 3D operations such as extrusion, enclosure, to mention a few operations.

Simulation in SolidWorks

To determine the structural behavior of the lifting platform model, a load distributed along the entire structure was configured. Figure 3 shows the resulting Von Mises stresses, with a minimum of 4.385e-03 MPa and a maximum of 7.571e-03 MPa.

On the other hand, later during post-processing, the total deformation of the structure was determined, which are 1e-3 mm as minimum and 2.65e+0 mm as maximum. This can be seen in Figure 4.

Box 2

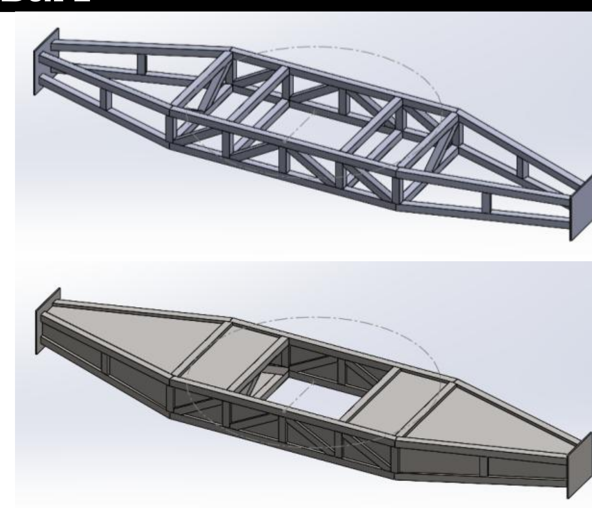


Figure 2
Computational model

Source: Own elaboration

Box 3

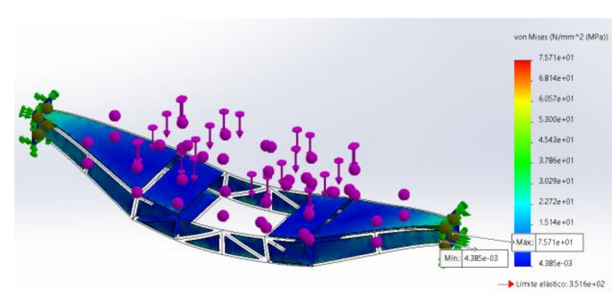


Figure 3
Von Mises forces

Source: Own elaboration

Box 4

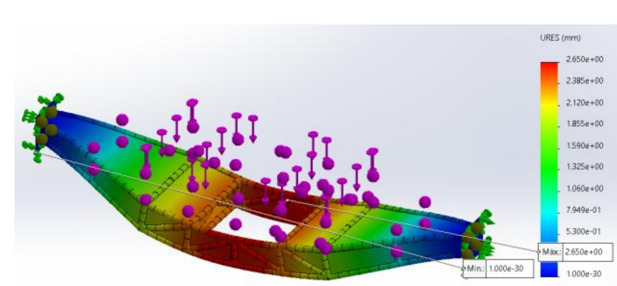


Figure 4
Total deformation

Source: Own elaboration

ISSN: 2531-2162
RENIECYT-CONAHCYT: 1702902

Betanzos-Castillo, Francisco, Fuentes-Castañeda, Pilar and Cortez-Solis, Reynaldo. [2024]. Structural analysis of a lifting platform for autonomous vertical vehicular parking. Journal of Architecture and Design. 8[19]-1-8: e10819108.
DOI: https://doi.org/10.35429/JAD.2024.8.19.1.8

Journal of Architecture and Design 8[19]1-8: e10819108
Article

Simulation in Ansys Workbench

To translate the behaviour from a physical model to a computational model, the problem must be understood, which leads to obtaining an analytical solution using the theory of simple stresses and stress concentration factors. The computational model of the platform for an autonomous vertical parking was designed in DesignModeler of Ansys Workbench, considering the geometry, load, boundary conditions and structural materials, in this case a PTR AISI 1010 material was used, square profile 127 mm per side, 12.7 mm thick, 25.4 mm plate, upper and lower central part without plate and lower sides without plate and derived from the symmetry conditions, an axisymmetric analytical analysis was performed, see figure 5.

In Figure 6, using the Static Structural module of Ansys Workbench 2024 R1 Student software, which was configured to use AISI 1010 structural steel for the proposed computational model, the static structural analysis of the elevation platform supporting the parking transfer was performed, which allowed determining the total deformation and equivalent stress.

Box 5

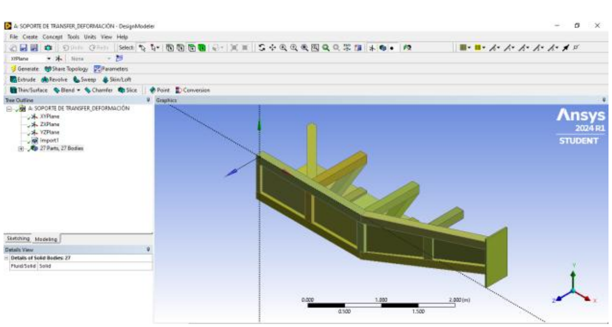


Figure 5
Axisymmetric geometry of the support structure

Source: Own elaboration

Box 6

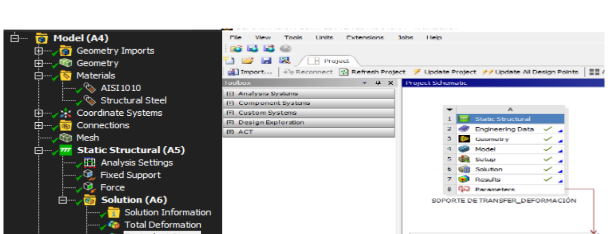


Figure 6
Configuration for static structural analysis simulation

Source: Own elaboration

Derived from the geometry of the computational model, an axisymmetric cut was made in Ansys, then the meshing was performed, with a mesh configuration for a type of mechanical analysis; it has 67086 elements and 1406 elements, being a discretization by triangular polygons to perform the configured analysis, this is shown in Figure 7.

Box 7

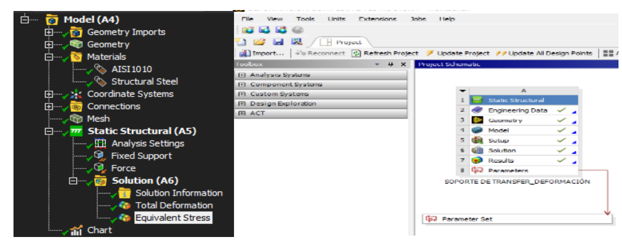


Figure 7
Discretization of mechanical type meshing

Source: Own elaboration

For the static structural analysis, an input force of 49033 N was used on 6 faces of the structure shown in red in Figure 8. According to the configurations made in the Ansys Workbench software, it can be observed that the platform structure presented deformation according to the colour code with a minimum value of 0 and a maximum value of 2.0223e-002 mm, as shown in Figure 9.

Box 8

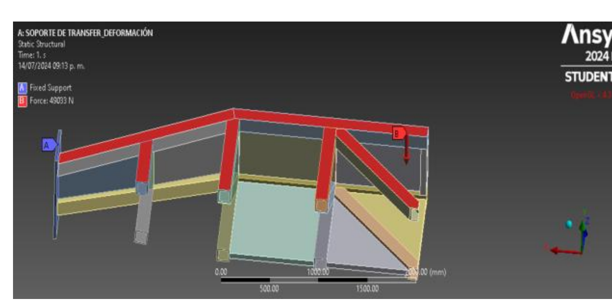


Figure 8
Configuration of forces and supports

Source: Own elaboration

ISSN: 2531-2162
RENIECYT-CONAHCYT: 1702902

Betanzos-Castillo, Francisco, Fuentes-Castañeda, Pilar and Cortez-Solis, Reynaldo. [2024]. Structural analysis of a lifting platform for autonomous vertical vehicular parking. Journal of Architecture and Design. 8[19]-1-8: e10819108.
DOI: https://doi.org/10.35429/JAD.2024.8.19.1.8

Box 9

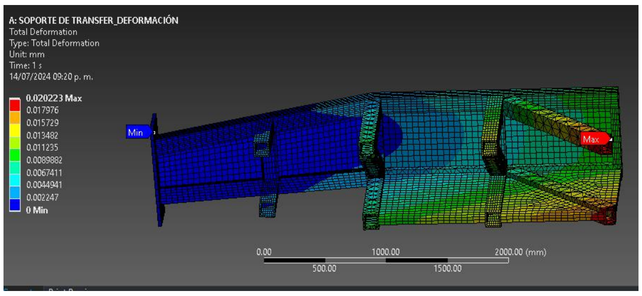


Figure 9
Minimum and maximum total deformation

Source: Own elaboration

Finally, Figure 10 shows the minimum and maximum equivalent stress values, i.e., using the Von Mises criterion, these were 0 MPa and 199.62 MPa, respectively, which are located at the end of the fastening element of the support structure of the lifting platform.

Box 10

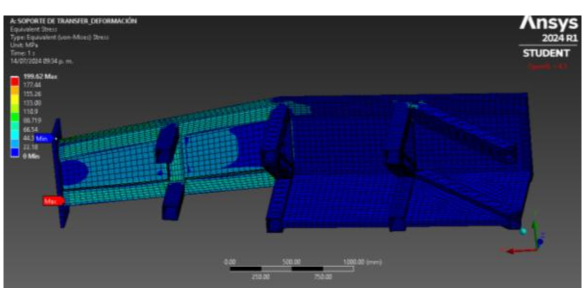


Figure 10
Equivalent (von-Mises) Stress

Source: Own elaboration

Discussion of results

The stress distribution and total deformation were determined numerically using the finite element technique, this type of analysis is very suitable for structural analysis, which allowed to determine the mechanical behaviour of a lifting platform that as a conceptual proposal will be used for an autonomous vertical parking with a vision towards Industry 4.0. In addition, this technique saves time and money in performing strength of materials calculations, as well as in the construction and prototyping of physical models.

The computational model represents and describes the total deformation and the equivalent stress when subjected to equivalent loads, making it possible to visualize the structural behaviour under different working and boundary conditions, which gives an idea of how such behaviour would be in reality and saves time and money.

The results of the present investigation indicate that it is feasible to use computational models with the help of CAD software, which allows with a certain relative ease to perform various simulations emulating with this the real physical conditions of this type of mechanical elements, as well as the comparison of the results obtained, so it is recommended to continue working on this line and take advantage of the benefits of these tools as support for research in the engineering area.

Conclusions

The present study demonstrated the potential of the finite element technique for similar physical phenomena in the area of strength of materials, since it allowed the numerical calculation of the total deformation and equivalent stresses (von Mises) by using two CAD software. This simulation will allow to extrapolate the results of real working considerations, since it is observed that both software have a greater potential, the results obtained are very similar, which undoubtedly will allow to analyze different scenarios of application.

Future work should be directed towards the study of new conceptual designs and make the comparison with theoretical calculations in order to optimize the results and resources. In this way, it will be possible to predict the behavior of the lifting platform in different materials subjected to different working conditions.

Declarations

Conflict of interest

The authors declare no interest conflict. They have no known competing financial interests or personal relationships that could have appeared to influence the article reported in this article.

Author contribution

The contribution of each researcher in each of the points developed in this research, was defined based on:

Betanzos-Castillo, Francisco: Contributed to the project idea, research method and technique.

Fuentes-Castañeda, Pilar: wrote the manuscript with input from all authors and developed the theoretical formalism.

ISSN: 2531-2162
RENIECYT-CONAHCYT: 1702902

Betanzos-Castillo, Francisco, Fuentes-Castañeda, Pilar and Cortez-Solis, Reynaldo. [2024]. Structural analysis of a lifting platform for autonomous vertical vehicular parking. Journal of Architecture and Design. 8[19]-1-8: e10819108.
DOI: https://doi.org/10.35429/JAD.2024.8.19.1.8

Cortez-Solis, Reynaldo: designed the model and the computational framework and analyzed the data.

Availability of data and materials

There are no data available.

Funding

The research did not receive funding, although it has received support from the Tecnológico Nacional de México - TES Valle de Bravo in terms of technical support.

Acknowledgements

Special thanks to the Tecnológico Nacional de México - TES Valle de Bravo, since their support has allowed the development of this project, important stages have been achieved that lead to an advance in technological development and in the training of human resources.

References

Basics

Díaz Iglesias, C. A. (11 de 08 de 2021). Repositorio Insitucional Universidad Distrital Francisco de Jose Caldas. Obtenido de repository.

SEMCOCAD. (19 de 05 de 2024). Soluciones y capacitación CAD/BIM. Obtenido de SEMCOCAD:

SOLIDBI. (19 de 05 de 2024). SOLIDBI-Inspira tu innovación.

J.C.TORRES. (1998). Google Académico.

Kassimali, A. (2015). Análisis Estructural. Cengage Learning. ISBN: 978-1-133-94389-1

Rodríguez A. G. y Ramos J. L. (2009). Renovación urbana del Centro Histórico de Barranquilla: orígenes y evolución del proceso. Revista Digital de Historia y Arqueología desde el Caribe, 11: 46-62.

Rodríguez, F. D. (2019). Academia.edu. Obtenido de Google Academico.

Supports

Galán Ávila, J. A., Avendaño Rodríguez, D. F., & Villalobos Correa, D. E. (2022). Design and simulation of mechanical press for testing of coining tools with nanostructured coatings. Revista Facultad de Ingeniería Universidad de Antioquía(104), 53-70.

Halicioglu, R., Canan Dulger , L., & Tolga Bozdana, A. (2016). Structural design and analysis of a servo crank press. Engineering Science and Technology, an International Journal, 19, 2060-2072.

Madenci, E., & Guven, I. (2015). The Finite Element Method and Applications in Engineering Using ANSYS. New York: Springer.

Nakasone, Y., Yoshimoto, S., & Stolarski, T. (2006). Engineering Analysis with ANSYS Software. Oxford: Elsevier.

Differences

Oguz Örmecioglu, T., Aydogdu, Í., & Tugba Örmecioglu, H. (2024). GPU-based parallel programming for FEM analysis in the optimization of steel frames. Journal of Asian Architecture and Buikding Engineering, 1-22.

Rameshrao Yawale, V., & Nivrutti Naik, N. (2021). Static structural and modal analysis of mechanical component using FEA approach. International Journal of Creative Research Thoughts, 9, 4003-4012.

Yixu Chu, Junying Lin, Kun Li. Design and Simulation of Foldable Wing eVTOL UAV. Academic Journal of Engineering and Technology Science (2024) Vol. 7, Issue 4: 136-143.

ISSN: 2531-2162
RENIECYT-CONAHCYT: 1702902