Ciencia & Futuro V. 15 No. 3. September-November 2025 p. 213-223 ISSN: 2306-823X
Romero Breffe, L., Santi García, L. Received: 28/4/2025/Accepted: 13/7/2025

Determination of stress in an AISI 1045 structural steel plate using finite elements

Determinación de las tensiones en una placa de acero estructural AISI 1045 mediante elementos finitos

Lisander Romero Breffe lromerob@ismm.edu.cu [1]
https://orcid.org/0009-0009-1791-0071

Leodenis Santi García leosantigarcia91@gmail.com [1]
https://orcid.org/0009-0007-8356-8433
[1] University of Moa, Moa, Cuba

Abstract: The objective of this research is to determine, through the Finite Element Method, the stresses that originate in a plate made of AISI 1045 structural steel. ANSYS 17.2 software was used to determine the stresses, through which the geometric and parameterized modeling, the plate's meshing, the definition of the load cases and their parameters, as well as the model boundary conditions were performed. It was determined that the Von Mises stress is 50.295 MPa; furthermore, according to the obtained results, the total displacement occurs at a value of 0.021181 mm.

Key words: numerical method, metal alloy, three-dimensional modeling

Resumen: El trabajo tiene como objetivo determinar a través del Método de los Elementos Finitos, las tensiones que se originan en una placa fabricada de acero estructural AISI 1045. Para la determinación de las tensiones se empleó el software ANSYS 17.2, a través del cual se realizó la modelización geométrica y parametrizada, el mallado de la placa y la definición de los casos de carga y de sus parámetros, así como las condiciones de frontera del modelo. Se determinó que, la tensión de Von Mises es de 50,295 MPa; además según los resultados obtenidos el desplazamiento total ocurre a un valor de 0,021181 mm.

Palabras clave: método numérico, aleación metálica, modelación tridimensional

Introduction

Baran (1989) defines Finite Element (FE) analysis as a group of numerical methods to approximate the equations governing any continuous system. FE principle is a mathematical theory based on variations calculation, energy theorems, the elasticity principle, and equations from physics and engineering (Benito Muñoz *et al.,* 2023). An FE program main function is to reduce the differential equation to a system of simultaneous algebraic equations which can be solved by a computer, such as Poisson´s second-order partial differential equation (Felix, Nogueira & Maíz, 2023; Mausa, 2024), which governs the deflections of a membrane in a bent prismatic beam and heat conduction with its source.

The scope of the method is considered broad (Otero *et al*., 2021; Cangelosi & Sanzi, 2022; Castillo, 2025). The analytical representation means the ability to combine, in a simple model with different model procedures, each type of element, being described by a differential equation, as already explained. The geometric representation capability is generally considered the first advantage of the method. Triangular elements can be arranged in any desired way to approximate the shape of the structure or body, in the plane or space (Ramos *et al*., 2009).

Two-dimensional (2D) vs. three-dimensional (3D) finite element models

In the 70s and 80s, part modeling was performed by using simplified two-dimensional models due to the computational limitations of that time. 2D modeling has the advantage of producing simpler models and reducing, significantly, calculation time, sometimes leading to acceptable results. Hibbitt and Marcal (1973) were pioneers in the application of a 2D model to determine residual stresses. Currently, despite computational advances, many authors continue to work with 2D modeling using plane or axisymmetric models and to assume certain suppositions and simplifications (Siddique et al., 2005; Ferro, Bonollo & Tiziani, 2010; Moreira, Lopes & Sanches, 2023).

Authors such as Mercado Bautista (2020), Sánchez *et al.* (2022); Quirón Avendaño & Cruz Zuñiga, (2024) Lorza *et al.* (2023); Estrada Garibay et al. (2024), validate the effectiveness of 3D modeling and its applicability to various technical spheres for problem-solving. Siddique *et al*. (2005) Bezerra, Rade & Scotti (2006) report that the thermal and thermodeformational response of all assemblies is three-dimensional and that only these models are capable of fully reproducing the thermodeformational phenomenon. Lu and Hassan (2001) have used three-dimensional modeling for the simulation of a mechanical part over four passes using a 3D model. They mention that, at the start and end points of the weld, a high concentration of residual traction and compressive stresses is observed which cannot be captured by two-dimensional models.

When creating a model, convergence analysis must be used to select the correct element size in the mesh and thus obtain minimal error in the results. Convergence defines that, by progressively refining the mesh, the numerical solution approaches, as closely as desired, the exact solution of the phenomenon.

This is a factor that must ensure that the calculation results do not depend on the mesh density (Goldak and Moashi, 1999; García y Bejar, 2003). As an approximate solution method, it can be stated that FE packages allow to find unknown quantities at specific points and with a certain proximity to the exact solution; an accuracy that, as mentioned, improves with an increase in the number of elements, in oder words, by increasing the mesh density.

Materials and Methods

Stress modeling by finite elements

The proposed FEM model is automatically defined from a job file programmed in PCL (Patran Command Language), which is subjected to special MSC commands. The FEM model resulting from this file execution contains the following main characteristics:

1. Geometric and parameterized modeling, which allows to define the matrix arrangement, metrics, and main dimensions of the plates as the user decides.
2. Automatic meshing of the plates. For the plates, beam elements according to Euler-Bernoulli formulation.
3. Automatic definition of the load cases and all necessary calculation parameters to generate the input file to the calculation program.

The generated text file is executed by the user in the finite element solver used, in this case, MSC.NASTRAN. Finally, the results are displayed in the post-processor, allowing for the treatment of all data considered interesting (stresses, system deformations, precise contact separations). Modeling using ANSYS consists of three fundamental stages: a) preprocessing, b) solution, and c) postprocessing. This methodology

(Carnicero, 2001) considers the working modes, the user graphical interface, and the ordered block programming mode. Figure 1 shows the user graphical interface.

Figure 1. User interface.

When running the software, the window titled "Welcome to ANSYS Workbench" closes and the analysis to be performed is selected; the one highlighted in red is chosen, which is a static analysis (Static Structural). In the identified toolbox, double-clicking will display what is shown on the screen in figure 2.

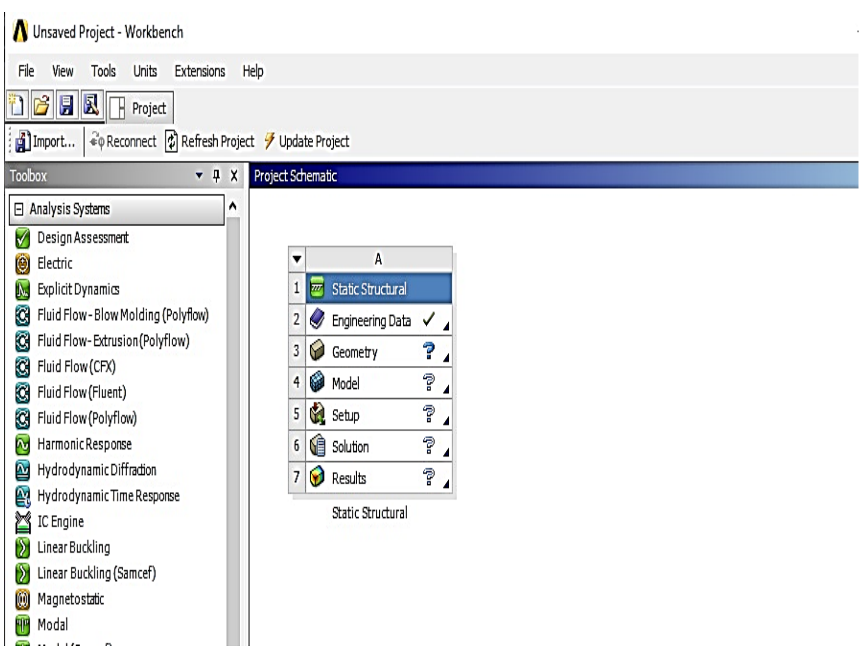

Figure 2. Selection of the analysis type.

This analysis was performed primarily in programming mode.

· The analysis type was declared as "Static".

· "FULL NEWTON RAPHSON" iterative method for solving nonlinear equations was used, which constitutes ANSYS standard nonlinear method for plasticity problems.

During the simulation process, after defining the material and its properties, and generating the geometry, the mesh is built with the chosen element type associated with that geometry. With that purpose, the element´s average size must be specified, which can be reduced later. A right-click is executed on the word *mesh>>Generate Mesh*, which produced the meshed model shown in figure 3.

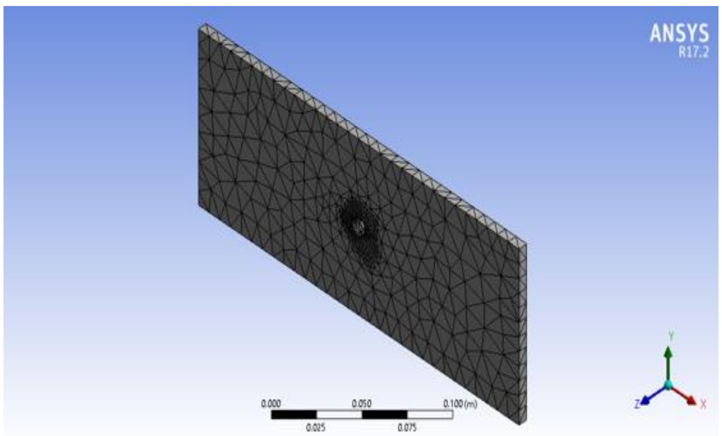

Figure 3. Model generation and meshing.

Since the stresses and strains will undergo greater changes in the hole, it is very useful to use a fine mesh only near this area. By using the h-method of solution, different element sizes can coexist in the mesh, thus achieving accuracy and resolution while adjusting additional computational costs. Thus, the AISI 1045 steel areas will have a relatively coarse mesh that will be refined near the hole, where meshing elements will be significantly smaller.

In order to mesh the model, the boundary conditions are established by executing the command *Supports>>Fix Supports* as shown in figure 4.

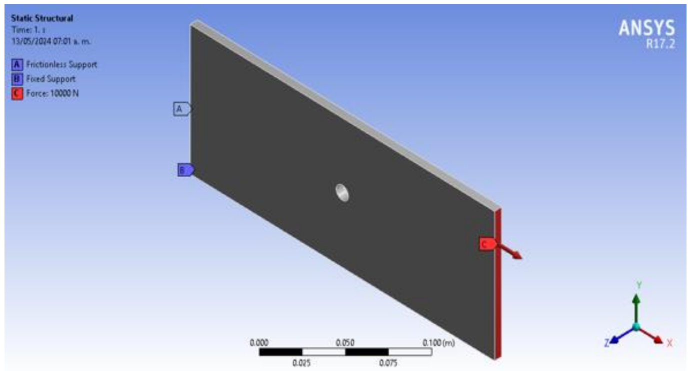

Figure 4. Model boundary conditions.

The boundary conditions were established to prevent rigid body motion and to be able to estimate the state of strains and stresses produced by external loads applied to the system; it was necessary to restrict part of its domain´s movement (u = 0 for Su domain). In this case, the applied forces were treated as a distributed load over the area of the red-colored face. These conditions were defined in ST domain.

Results and Discussion

Analysis by the Finite Element Method

One way to characterize steel´s plastic propertiesis through the Johnson-Cook mechanical model, which reproduces the behavior of materials subjected to large strains, high temperatures, and high strain rates. The formulation of this model defines Von Mises equivalent stress as three decoupled terms where the first one defines the dependence on strain hardening, the second, the strain rate sensitivity and the third, the temperature sensitivity. Figure 5 shows the analysis of the stresses the plate is subjected to.

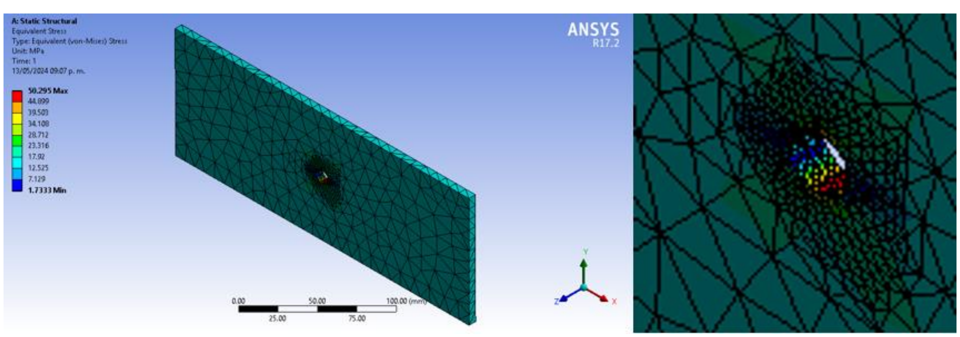

Figure 5. a) Von Mises equivalent maximum stress. b) Hole.

In this situation, the plate acts under rigid conditions and it can be observed that, in the distribution of Von Mises stress, the most critical zones occur in the hole (Figure 5b); it is determined that the maximum stress in AISI 1045 structural steel plate is 50.295 MPa.

Material deformation occurs due to elastic behavior, where the curve growth is linear until rupture occurs. From this moment on, strain returns to zero and begins to increase linearly until it becomes constant (constant stress). Figure 6 shows the total deformation of the plate made of AISI 1045 structural steel.

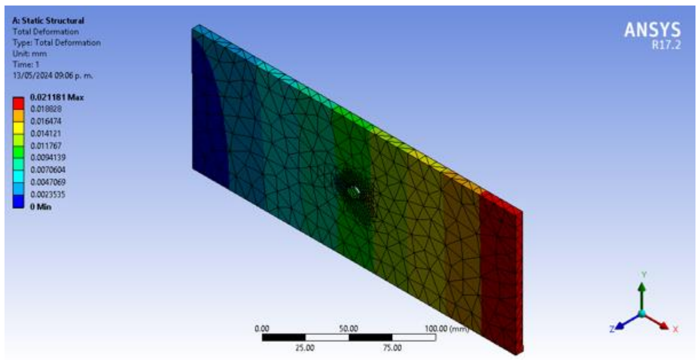

Figure 6. Total deformation.

The map of strain suffered by the material at the first end (red zone) is shown, as it is the one undergoing the greatest deformations. According to the obtained results of a total deformation of 0.021181 mm, it is observed that the load is mainly distributed in the first three parts of the plate where the first one takes about 55% of the load, the second 25%, and the third 15%. León *et al.* (2001) have stated that the theoretical distribution suggests that most failures would occur in the first chord; they also demonstrated that 90% of the failures occurred in the first chord when subjected to an axial load.

Once the load distribution in contact with the plate was determined, the stress intensity was determined. Figure 7 shows the result obtained from this analysis.

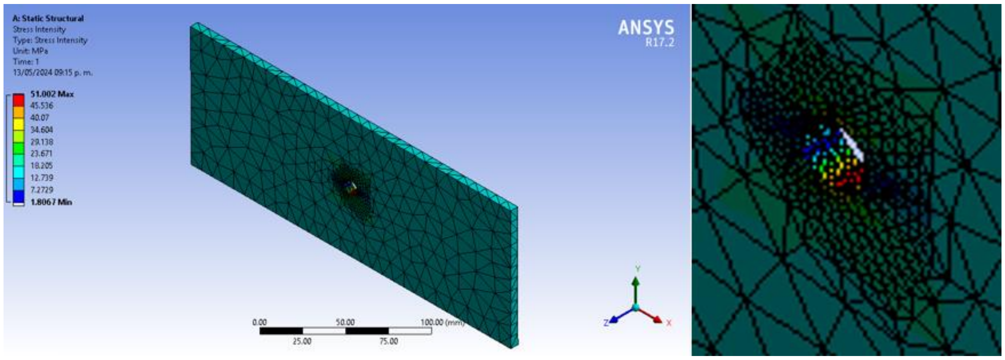

Figure 7. a) Stress intensity in the model. b) Maximum intensity zone.

It is observed that stress intensity is located in the hole (Figure 7b), with a result of 51.002 MPa, although the distribution of this stress is irregular due to two factors contributing to the stress state in that zone: plate stretching under the axial load and contact zone bending. The stress magnitude under only the axial load was determined using the stress concentrator proposed by Pilkey and Pilkey (2020) with a correction factor proposed by Dragoni (1997), assuming it behaves like a periodic notch in plate.

Conclusions

The stresses originated in a plate made of AISI 1045 structural steel were determined by using the finite element method.

It was determined that the stresses in the plate can be influenced by various factors such as geometry, the applied load, and the properties of the material used in its fabrication.

The Von Mises maximum equivalent stress to which the plate is exposed is 50.295 MPa, indicating that the component is subjected to significant loads and may be at risk of failure by fracture.

Bibliographic References

Baran, N. (1989). *Finite Analysis on Microcomputers*. McGraw-Hill book Company.

Benito Muñoz, J.J., Salete Casino, E., Álvarez Cabal, R., Ureña Prieto, F., & Flores Escribano, J. (2023). *Introducción al método de los elementos finitos*. Editorial UNED.

Bezerra, A., Rade, D., & Scotti, A. (2006). Simulation of a TIG weld using finite element method: Part 1 Thermal Analysis. *Soldagem & Insecaop*, *11*(1), 12-22.

Cangelosi, C.G., & Sanzi, H.C. (2022). Análisis termomecánico de un reactor mediante el método de diferencias finitas y su verificación a partir del método de los elementos finitos. *Revista Iberoamericana de Ingeniería Mecánica, 26*(1), 25-41. https://revista.uned.es/universidad/dam/facultades/industriales/RIBIM/V26N1Abril2022/26N1Ab20200-A04.pdf

Carnicero, A. (2001). *Introducción al método de los elementos finitos.* Escuela Técnica Superior de Ingeniería ICAI. Universidad Pontificia Comillas, Madrid.

Castillo, J.C. (2025). Ecuaciones Diferenciales: Fundamentos. Método de Solución y Aplicaciones en Sistémicas Dinámicos y Teoría del caos. *Ibero ciencias. Revista Científica Académica*, 4(2), 1-21. https://revistaiberociencias.org/index.php/multidisciplinar/aticle/view/35

Dragoni, E. (1997). Effect of thread pitch on the fatigue strength of steel bolts. Proceeding of the institution of mechanical engineers. Part C. *Journal of Mechanical Engineering Science*, *211*(8). https://doi.org/10.1243/0954406981521970

Estrada-Garibay, J.A., Vázquez-Martínez, B., Castillo-González, D.J. & Ochoa-Almazán, D.E. (2024). Prototipo de dispensador automático para la optimización de ventas en diferentes sectores. *Revista De Ciencias Tecnológicas,* 7(3). https://doi.org/10.37636/recit.v7n3e358

Felix, D.H., Nogueira, S.F., & Maíz, S. (2023). Análisis del Efecto Poisson en las Vibraciones Naturales de Microvigas Timoshenko Mediante el Método de Elementos Finitos. *Mecánica Computacional*, *40*(5), 131-140. https://amcaonline.org.ar/ojs/index.php/mc/article/view/6533

Ferro, P., Bonollo, F. & Tiziani, A. (2010). Metodologías y validaciones experimentales de la simulación numérica del proceso de soldadura. *Revista Internacional de Ciencia Computacional de Materiales e Ingeniería de superficies, 3*(2-3), 114-132. https://doi.org/10.1504/IJCMSSE.2010.033148

García, Y. & Bejar, A. (Noviembre 2003). *Obtención de los campos térmicos mediante el Método de los Elementos Finitos en las reparaciones por soldadura en equipo o estructuras metálicas de gran importancia*. XXV Congreso Internacional de Metalurgia y Materiales. Instituto Tecnológico de Saltillo, México.

Goldak, J. & Moashi, G. (1999). *Numerical Aspect of modelling Weld*. Carleton University. ASM Handbook, Vol 6 Welding, Brazing and Soldering.

Hibbitt, H. & Marcal, P. (1973). A numerical thermo-mechanical model of the welding and subsequent loading of a fabricated structure. *Computadoras y Estructuras*, *3*(5), 1145-1174. https://doi.org/10.1016/0045-7949(73)90043-6

Lorza, R.L., Martínez, A.P., Macedo, S.Í., Fraile, C.S., Gomez, F.S., Bobadilla, M.C.& González, S.R. (2023). *Caracterización mecánica y térmica de Bronce-PLA fabricado mediante modelado por deposición fundida MDF*. XXIV Congreso Nacional Ingeniería Mecánica (CNIM 2023). Libro de Resúmenes.

Lu, X. & Hassan, T. (2001). Residual Stresses in Butt and Socket Welded Joints. *SMiRT*, 16. https://repository.lib.ncsu.edu/server/api/core/bitstreams/034c9c28-0775-4171-ae54-3309d7f9fea8/content

Mausa, S.P. (2024). *Solución numérica para las ecuaciones de Nernst-Planck-Poisson usando un Método de Elementos Finitos.* (Doctoral Thesis, University of Córdoba, Colombia). https://repositorio.unicordoba.edu.co/server/api/core/bitstreams/eda9be7a-f969-4e0b-97ac-1231e58057e6/content

Mercado Bautista, J.D. (2020). Evolución de los softwares de simulación para el Diseño y Construcción en la Industria. *Polo del Conocimiento, 5*(8), 1332-1343. https://dialnet.unirioja.es/servlet/articulo?codigo=7554321

Moreira, D.H., Lopes, W.C., & Sanches, R.A. (2023). Stable high order space-time finite element formulation for large displace-ment elastodynamics. *XLIV Ibero-Latin American Congress on Computational Methods in Engineering*, *5*(05). https://publicacoes.softaliza.com.br/cilamce2023/article/download/5095/4173

Otero, P., Peñafiel, S., Vallejo, D., Granda, N. & Ramírez, J. (2021). Simulación de Mallas de Puesta a Tierra utilizando el Método de los Elementos Finitos en Tres Dimensiones. *Revista Técnica Energía, 17*(2), 113-123. https://doi.org/10.37116/revistaenergia.v17.n2.2021.437

Pilkey, W.D., Pilkey, D.F., & Bi, Z. (2020). *Factores de concentración de tensiones de Peterson*. John Wiley & Sons.

Quirón Avendaño, L.A., & Cruz Zuñiga, N. (2024). BIM 6D methodology as a tool to evaluate thermal comfort in Costa Rican schools. *Ingeniería, 35*(1), 24-37. http://dx.doi.org/10.15517/ri.v35i1.60089

Ramos, F., Duffus, A., Rodríguez, M., Diaz, E,. & Pozo, J. (2009). Influencia de la termodependencia de las propiedades físicas del acero en la simulación por elementos finitos del proceso de soldadura. *Revista de la Facultad de Ingeniería de Antioquia*, *48*(4), 87-96. http://pascal-francis.inist.fr/vibad/index.php?action=getRecordDetail&idt=21411409

Sánchez, J.A., Sánchez, T.A., Gasca, A., Martínez, L.S., Núñez, K.N., Ruelas, M., & Sánchez, F. (2022). Simulación del esfuerzo mecánico en una pieza modelada en 3D por el método de los elementos finitos. *Jóvenes en la Ciencia*, 16, 1-10. https://www.jovenesenlaciencia.ugto.mx/index.php/jovenesenlaciencia/article/view/3567

Siddique, M., Abid, M., Junejo, H. & Mufti, A. (2005). 3-D finite element simulation of welding residual stresses in pipe-flange joints: effect of welding parameters. *Materials Science Forum*, (100), 79-84. https://doi.org/10.4028/www.scientific.net/MSF.490-491.79