XI SIINTEC
INTERNATIONAL SYMPOSIUM ON INNOVATION AND TECHNOLOGY
QUANTUM TECHNOLOGIES: The information revolution that will change the future
International Year of Quantum Science and Technology
SENAI CIMATEC UNIVERSIDADE

Finite Element Analysis of a Polymeric Support with metallic internal reinforcement for Carbon Steel Pipes

Gabriel Aelo Ribeiro[1], João Pedro Cheloni[1*], Adriel da Anunciação Santana[1], Lucas Lincoln Fonseca Soares[1], Francisco Ferreira de Carvalho Júnior[2]

[1] *SENAI CIMATEC, Development of Industrial Projects - DPI, Salvador, BA, Brazil*
[2]*Mauari, Plastic Components Industry, Betim, MG, Brazil*
**SENAI CIMATEC; Av. Orlando Gomes; joao.cheloni@fieb.org.br*

Abstract: The proper storage of carbon steel pipes is a critical concern in various industrial sectors where structural integrity and corrosion resistance are essential to operational safety and longevity. Traditionally, metallic supports have been used due to their high strength. However, these supports can accelerate corrosion through galvanic coupling when in contact with carbon steel pipes, posing a significant long-term risk. To address this issue, this study investigates the feasibility of using polymeric supports internally reinforced with metallic tubes as an alternative design that maintains mechanical performance while mitigating galvanic corrosion. A methodology of numerical modeling was adopted to analyze the proposed support structures using Finite Element Analysis (FEA). Eight polymeric support models were designed, incorporating a square cross-section steel tube reinforcement, and adjusted to sustain loads of up to 8 tons. To understand the mechanical behavior of these hybrid supports, simulations were conducted with varying steel tube wall thicknesses. The failure criteria were analyzed for both metallic and polymeric materials, considering von Mises stress and maximum allowable stress values. The simulation results revealed that a minimum steel tube thickness of 8 mm is required to prevent structural failure in both the polymeric support and the metallic reinforcement. Thinner configurations led to stress concentrations that exceeded permissible limits, compromising safety. These findings underscore the necessity of tailored engineering solutions when dealing with polymeric structural elements and hybrid systems. Furthermore, the study demonstrates how the integration of informed material selection and numerical simulation can lead to safer, more cost-effective, and corrosion-resistant storage solutions for carbon steel piping systems.

Keywords: Carbon steel pipes, Polymeric supports strength, Finite Element analysis, failure criteria

Abbreviations: FEA, Finite Element Analysis; FEM, Finite Element Methods.

1. Introduction

According to Gentil (1996) [1], a galvanic pile occurs when two metals are in contact and in the same environment. The use of traditional metallic supports for steel pipe, though robust, can accelerate pipeline corrosion due to galvanic effects, particularly in harsh environments. In this perspective, polymeric supports can be an alternative, offering advantages such as reduced weight, improved corrosion resistance, and the potential for tailored mechanical performance.

The use of polymers and composites for pipe corrosion prevention is largely investigated in scientific literature. Alabtah (2021) [2] demonstrated the effectiveness of fiberglass and Kevlar reinforced polymer coatings against corrosion using acids and corrosive solutions. Farias (2016) [3] introduced the using of PA 12 in three-layer system coating which has high impact resistance and toughness.

To predict the mechanical behavior of this type of structure, the Finite Element Method can be used. In this method, a large problem is subdivided into smaller, simpler parts called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. The finite element method then uses techniques based on the calculus of variations to approximate a solution (for the displacement field of the geometry) by minimizing an associated error function Madier (2020) [4].

This article presents a detailed analysis of the mechanical response of a polymeric support with internal metal reinforcement applied to carbon steel pipes, using three-dimensional modeling and computer simulations. The temperature which simulations have been performed was 25 °C (ambient temperature), but the influence of its variation was not investigated in this study. The objective is to validate the efficiency of the reinforcement, propose improvements in the design, and contribute to the advancement of hybrid solutions in pipeline engineering.

2. Methodology

2.1. Materials and Geometry

In this study, eight polymeric supports internally reinforced with metallic tubes were developed. Two supports are combined to form a cradle, each designed to sustain a load of 8 tons attributed to placement of metallic pipes above the cradle. To prevent sliding during load application, metallic plates were installed at the ends of the cradle assembly.

Figure 1 (a) illustrates the intended application of the polymeric supports in conjunction with the pipe arrangement, while Figure 1 (b) details the configuration, highlighting the eight supports, the internal reinforcement, and the steel plates at the ends. The internal reinforcing steel tube has a square cross-section of 70 × 70 mm and a wall thickness of 3 mm.

The materials employed were Nylon (PA 6) for the polymeric support and structural carbon steel for the steel tube and plates. Table 1 provides their respective properties used in this study.

FIGURE 1

TABLE 1

2.2. Design strategy

The strategy adopted in the studies for design and simulate the components of followed the flow diagram in Figure 2, and the steps taken are described in more detail below.

FIGURE 2

- Physical understanding of the problem: Understanding the physics of the problem and assessing considerations regarding the study conducted, in order to control the computational cost of the analysis.
- Geometry treatment: The geometry submitted had to be slightly modified to improve the generation of mesh elements assigned to the finite element method used.
- Assigning material properties: Definition of properties such as Modulus of Elasticity and Poisson's Ratio.
- Mesh generation: Partitioning of the continuous medium into subdomains called elements.

• Definition of contact surfaces: The analysis performed presents two bodies in contact, which requires reptubeucing the effects generated by this contact in the transmission of the load.

• Boundary conditions: Application of external forces and constraints of the problem.

• Analysis solution: Solving the equations that govern the previously defined numerical model using the programming language implemented in the software used.

• Analysis of results: In this step, the results obtained by the numerical model are critically studied.

2.3. Numerical Simulation

Finite element (FE) models were developed using ANSYS Mechanical software, considering different thicknesses for the inner steel tube, to analyze the von Mises stresses in the parts. The loads and boundary conditions were analyzed as shown in. The load of 80 kN was applied over the upper surface of each support pair. Regarding movement constraints, displacement in the Y-direction was restricted on the bottom surface of the support simulating contact with a flat surface. Contact between the metal part and the polymer was modeled using a friction coefficient of 0.4. A mesh with 3D solid elements ranging from 1 mm to 20 mm was used to discretize the model.

FIGURE 3

2.4. Failure Criteria

The structural analyses developed are based on von Mises failure criterion for metallic materials, with stresses obtained using the Finite Element Method (FEM). A safety factor (SF) of 1.5 is adopted to account for uncertainties such as material impurities, assembly inaccuracies, and other potential defects that could lead to failure. Accordingly, the stress-based failure criterion is defined as follows:

$$\sigma_{adm} = \frac{\sigma_{esc}}{SF} \tag{1}$$

Where σ_{adm} is the allowable stress for the problem, σ_{esc} is the material's yield strength and *SF* is the safety factor. The structure is considered structurally safe when:

$$\sigma_{misses} < \sigma_{adm} \tag{2}$$

Considering yield strength for steel of 250 MPa and the specified safety factor of 1.5, the allowable stress is calculated to be 167 MPa. Therefore, for a geometry to be approved, the von Mises stress (σ_{misses}) obtained from the simulations must not exceed this value.

The failure criterion for the polymeric material is given by Equation 3 according Erhard (2006) [5]. This equation establishes the maximum permissible stress σ_{perm} given by:

$$\sigma_{perm} = \frac{K}{S} \cdot \frac{1}{A_T} \cdot \frac{1}{A_{st}} \cdot \frac{1}{A_{dyn}} \cdot \frac{1}{A_A} \cdot \frac{1}{A_W} \tag{3}$$

Where K (43,4 MPa) is characteristic strength, S (1,3) is safety factor for a 0.5% strain limit, A_T (1,0) is temperature factor, A_{st} (1,3) is static load factor, A_{dyn} (1,3) is dynamic load factor, A_A (1,0) aging factor e A_w (1,0) is water absorption factor. Applying these parameters to Equation 3 results in a maximum permissible stress for the polymer of 19.71 MPa.

3. Results

The results from the initial analysis are available in Figure 4. With 3 mm thick tube for reinforcement of the polymeric part, the maximum stress is way beyond the admissible 167 MPa as mentioned before in section 2.4.

FIGURE 4

It was noted that the most part of the load is handled by the steel tube, the hypothesis that increasing the stiffness of the tube (i.e. increasing its thickness) would lead to a consequential diminishing of the polymeric part stresses was tested. Table 2 summarizes the results obtained from analysis performed from the variation of tube thickness.

TABLE 2

While the new analysis were being made, a necessity to refine the mesh had been felt for the analyses. To save computational time, an approach of study just a piece of the model had been carried on since, as can be seen from the results of Figure 4, the behavior of the entire component follows a pattern. Figure 5 shows the boundary conditions adoppted for the new studies.

FIGURE 5

In the last result from the Table 2, the aproval criteria were met and the stress and displacement plots can be seen in Figure 6 (for the internal reinforcement and Figure 7 (polymeric part).

FIGURE 6

FIGURE 7

4. Conclusions

At this work, the geometry of a polymeric support reinforced internally with a metallic tube made from steel was investigated in terms of its structural strength. Initially the 3 mm thick steel tube was not enough to handle the load causing failure in the pollymeric support too. Analysing the results globally, it was perceived that as the thickness of the tube was increased, the stresses in both parts decreased.

However, confronting the results of the maximum stresses in the component with 6,3 mm thick tube against the one with 4,75 mm a slightly increase of 3% occurred. This can be explained by the mesh refinement made in the studies from the 6,3 mm case forward, which converged the stresses and revealed some local effects not captured by the previous analysis.

As a final output from the analysis, it was possible to verify that the support can withstand the load applied if it is reinforced with 8 mm thick tube.

References

[1] Gentil, V., Corrosão, LTC – Livros Técnicos e Científicos Editora S.A., Rio de Janeiro, 1996.

[2] Alabtah, F.G. et al., Towards the Development of Novel Hybrid Composite Steel Pipes: Electrochemical Evaluation of Fiber-Reinforced Polymer Layered Steel against Corrosion, Qatar, 2021.

[3] Farias, C.A. et al., Polyamide 12 in Extruded Coating Systems for Steel Pipe Protection, ABRACO, Búzios, 2016.

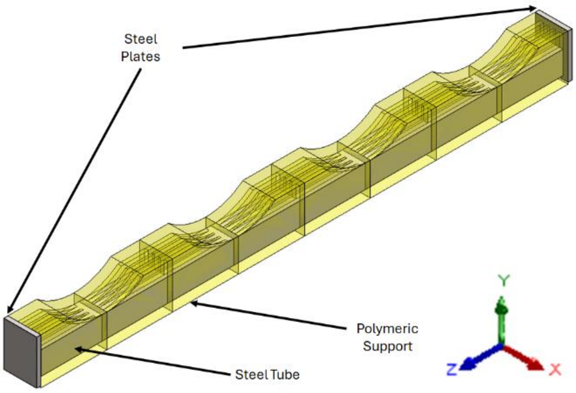
[4] Madier, D., Practical Finite Element Analysis for Mechanical Engineers, FEA Academy, Canada, 2020.

[5] Erhard, G., Designing with Plastics, Hanser Gardner Publications, Cincinati, 2006.

FIGURES

Figure 1: Analyzed geometry.

(a)



(b)

Figure 2: Methodology Applied in the work.

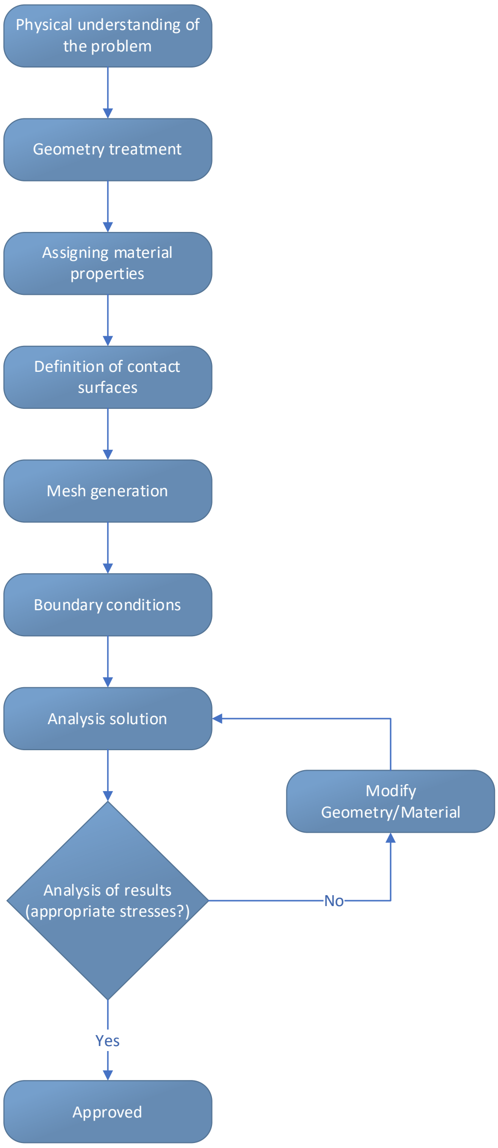


Figure 3: Loads and boundary conditions

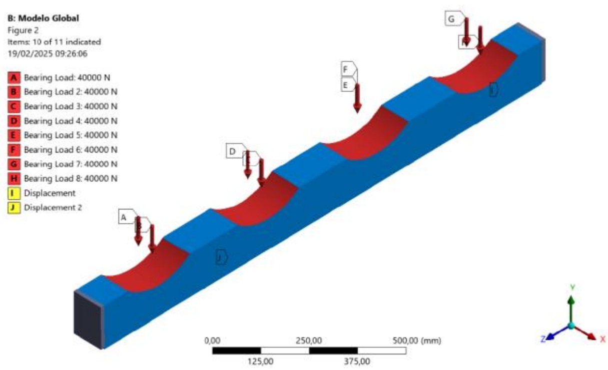
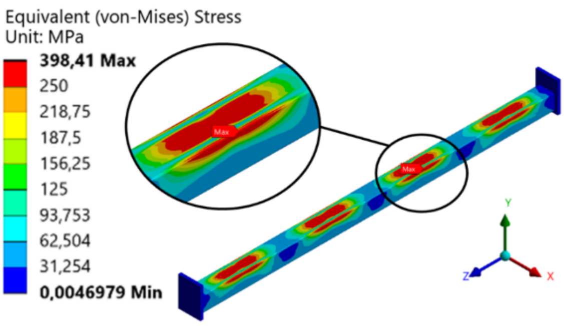
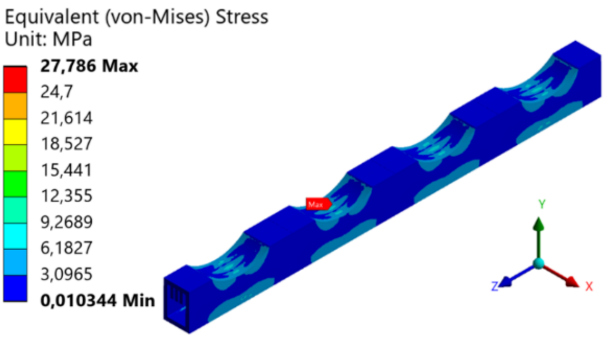


Figure 4: Results from analysis of the assembly - 3 mm thick steel tube reinforcement: (a) Stresses in the tube reinforcement and (b) Stresses in the polymeric part.



(a)



(b)

Figure 5: New model for mesh refinement.

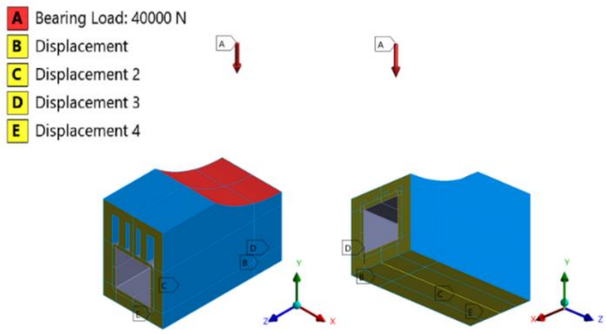
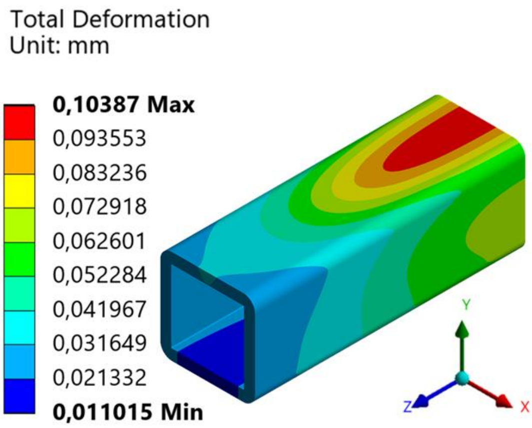
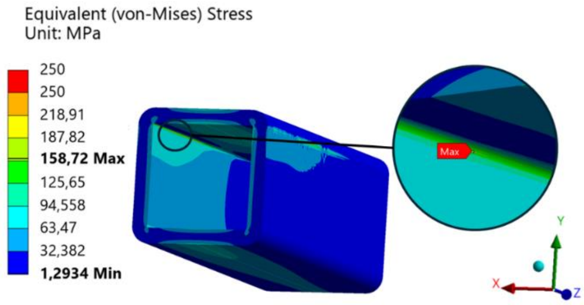


Figure 6: Analysis results for the internal reinforcement (with 8 mm thickness): (a) Total Displacement and (b) Von Mises Stresses

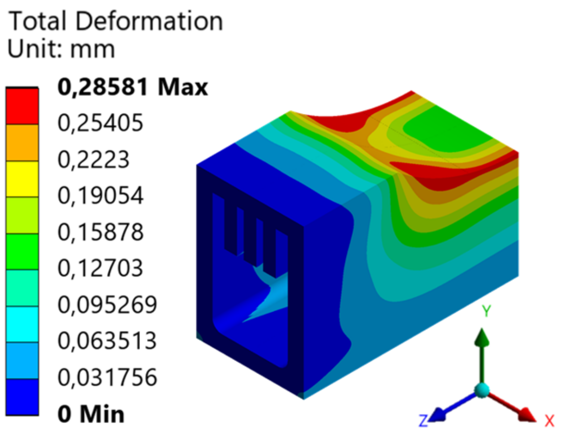


(a)

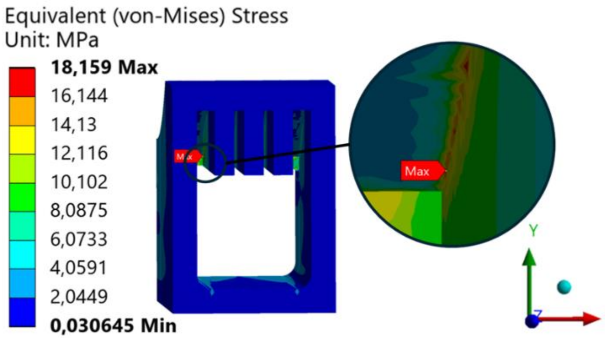


(b)

Figure 7: Analysis results for the polymeric part: (a) Total Displacement and (b) Von Mises Stresses



(a)



(b)

TABLES

Table 1: Mechanical properties of the materials used in the analysis.

Component	Material	Elastic Modulus (GPa)	Poisson's Ratio	Yield Strength (MPa)
Polymeric Part	PA 6	0.12	0.35	43.13
Tube	Structural steel	200.00	0.30	250.00

Table 2: Analysis results from the variation of tube thickness.

Tube Thickness (mm)	Max. stresses in polymeric parts (MPa)	Max stresses in tube (MPa)
3	27,786	398,41
4,75	23,267	217,17
6,3	19,501	223,7
8	18,159	158,72