

BULETINUL INSTITUTULUI POLITEHNIC DIN IAȘI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iași
Volumul 72 (76), Numărul 3, 2026
Secția
CONSTRUCȚII DE MAȘINI
DOI: 10.2478/bipcm-2026-0026

STRUCTURAL RESPONSE OF A PROTECTIVE BRAKE SYSTEM SHIELD UNDER THERMAL LOAD

BY

EUGEN-CIPRIAN APRICOPOAIE and CARMEN BUJOREANU*

“Gheorghe Asachi” Technical University of Iași,
Faculty of Mechanical Engineering, Iași, Romania

Received: July 21, 2026

Accepted for publication: August 11, 2026

Abstract. Protective sheet-metal components located in the engine compartment are exposed to significant temperature variations generated by the powertrain and adjacent heat sources. These thermal loads can affect stress distribution, deformation behavior, and the performance of bolted joints. This study investigates the thermo-mechanical response of a steel protective shield mounted on a valve block support using an M6 class 8.8 bolted connection. A three-dimensional finite element model was developed to evaluate the assembly under an initial bolt preload of 8 kN and three thermal cycles between -40°C and $+120^{\circ}\text{C}$. The analysis focused on equivalent von Mises stresses, total deformation, contact pressure, and bolt clamp force evolution. The results indicate that thermal cycling causes periodic variations in preload and localized stress concentrations, particularly around the mounting hole and geometric discontinuities. The maximum bolt force remained below the ISO 898-1 proof load limit, confirming elastic behavior of the fastener. Furthermore, no significant preload relaxation was observed, demonstrating that the assembly maintains adequate clamping performance and structural integrity under the investigated thermal conditions.

Keywords: Finite element analysis, thermal cycling, bolted joint, clamp force, thermo-mechanical behavior.

*Corresponding author; *e-mail*: carmen.bujoreanu@academic.tuiasi.ro

© 2026 Eugen-Ciprian Apricopoaie and Carmen Bujoreanu

This is an open access article licensed under the Creative Commons Attribution-NonCommercial-NoDerivatives 4.0 International License (CC BY-NC-ND 4.0).

1. Introduction

The Protective sheet-metal components installed within the engine compartment are continuously exposed to significant temperature variations generated by the operation of the internal combustion engine and adjacent heat sources. During vehicle operation, these components experience repeated heating and cooling cycles, which can induce thermal stresses and deformations, especially when the parts are mechanically constrained through bolted connections. Over time, the interaction between mechanical preload and thermal loading may influence the structural integrity and durability of the assembly. Current development cycles necessitate the integration of finite element analysis (FEA) during the early design phases to predict these complex thermomechanical responses, as traditional prototyping fails to capture the intricate dynamics of restricted multi-layer shielding systems (Zika-Beyerlein *et al.*, 2005). By employing numerical simulation techniques, engineers can effectively model the transient thermal gradients and resultant structural deformations, providing critical insights into the potential for material fatigue (Belhocine and Abdullah, 2021; Machado and Ribeiro, 2025).

Bolted joints are extensively used in automotive structures due to their robustness, ease of assembly, and maintenance advantages. The performance of such joints depends primarily on the clamping force applied during assembly, which ensures proper load transfer between the connected components. However, exposure to severe thermal environments may alter the initial preload condition through differential thermal expansion of the assembly components, leading to a redistribution of stresses and contact pressures within the joint. Consequently, the combined effect of bolt preload and thermal loading must be carefully evaluated during the design stage. Furthermore, the use of coupled thermal-mechanical finite element analysis allows researchers to account for these environmental variations when predicting the long-term susceptibility of such assemblies to deformation and structural instability (Yang *et al.*, 2017). In particular, such computational frameworks must incorporate thermal contact resistance at the interfaces and heat flow through gaps to precisely determine the evolution of bolt clamping forces under fluctuating operating conditions (Fukuoka, 2005).

The component investigated in this study is a protective shield located in the engine compartment and attached to a rigid support through a bolted connection. Because of its proximity to heat-generating systems, the component is subjected to substantial temperature fluctuations throughout its service life. In addition to thermal loads, the assembly must maintain its structural functionality and fastening integrity under all operating conditions. Therefore, understanding its thermo-mechanical response is essential for ensuring long-term reliability. To address these requirements, this study implements a sequentially coupled thermo-mechanical framework to quantify how transient temperature gradients and evolving contact states influence the stiffness and stress distribution of the bolted

assembly (Guo *et al.*, 2025; Zhou *et al.*, 2015). This computational approach particularly focuses on the evolution of bolt force, which varies over time as the thermal load alters the clamping state at the interface (Fukuoka *et al.*, 2009; Sasikumar *et al.*, 2021).

FEA provides an efficient numerical approach for evaluating the behavior of complex assemblies subjected to coupled mechanical and thermal loading conditions. Through detailed stress, deformation, and contact analyses, critical regions can be identified and potential failure mechanisms can be assessed before physical validation testing. Furthermore, numerical simulation offers valuable insights into the influence of thermal cycling on bolted joint performance and structural durability.

The objective of the present work is to investigate the thermo-mechanical behavior of an engine-compartment protective bracket subjected to an initial bolt preload of 8 kN followed by repeated thermal cycles between -40°C and $+120^{\circ}\text{C}$. Three complete thermal cycles were considered in the numerical analysis to reproduce severe environmental operating conditions. Particular attention was given to the evolution of von Mises stresses, total deformations, contact pressure distribution, and bolt clamping force throughout the loading sequence. The obtained results are intended to support the structural assessment and future optimization of bolted sheet-metal components operating in elevated thermal environments.

2. Geometry and Material properties

2.1 Geometry of the Assembly

The investigated assembly consists of a sheet metal bracket mechanically fastened to a rigid support using a bolted connection. The bracket includes a reinforced mounting area with a central hole designed for bolt installation. The support component provides the structural interface and constrains the bracket during operation.

The geometrical configuration was imported into the finite element environment and simplified by removing non-functional features that do not significantly influence the stress distribution. Particular attention was given to the mounting hole and the surrounding embossed region since these areas are expected to experience significant stress concentrations during mechanical preloading and thermal cycling.

Figure 1 presents the three-dimensional model of the analyzed assembly.

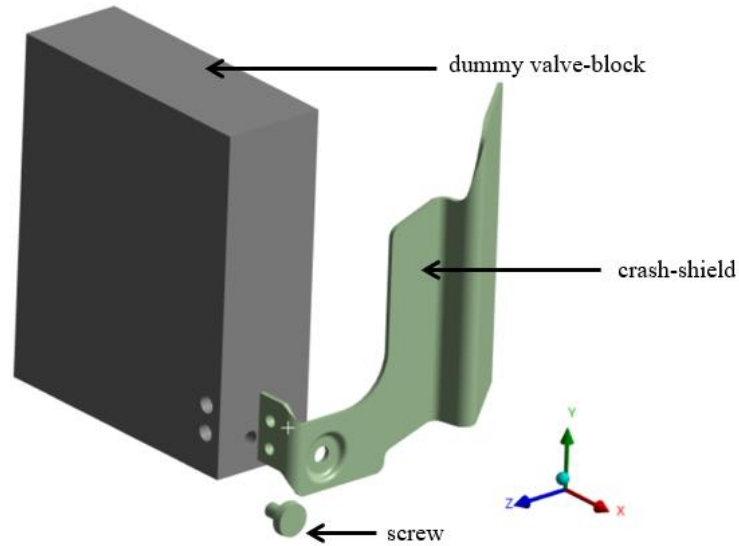


Fig.1 – Assembly.

2.2. Material Properties

The mechanical and thermal behavior of the assembly was modeled assuming homogeneous and isotropic materials. Bilinear material definition was considered throughout the analysis. The required material parameters include Young's modulus (E), Poisson's ratio (ν), Yield strength ($R_{p0.2}$), Tensile strength (R_m), and coefficient of thermal expansion.

Table 1 shows mechanical properties that were assigned to the components to accurately capture both the preload-induced stresses and the thermally induced deformations arising during the temperature cycles.

Table 1
Material properties

Body name	Poisson's ratio [-]	Young's Modulus [MPa]	Yield Strength [MPa]	Tensile Strength [MPa]	Coefficient of thermal expansion [1/°C]
valve-block	0.33	68900	240	350	2.5e-05
crash-shield	0.3	2.e+05	360	540	1.2e-05
screw	0.3	2.1e+05	640	800	1.1e-05

3. Finite Element Model

3.1. Mesh generation

A three-dimensional finite element mesh was generated for all assembly components. Mesh refinement was applied in areas expected to exhibit elevated stress gradients, particularly around the bolt hole, contact interfaces, and local geometric transitions.

The adopted mesh density provided a suitable compromise between computational efficiency and solution accuracy. The resulting model contained sufficient elements to accurately capture local stress concentrations generated by bolt preloading and thermal expansion mismatch.

Figure 2 illustrates the finite element mesh used in the simulation.

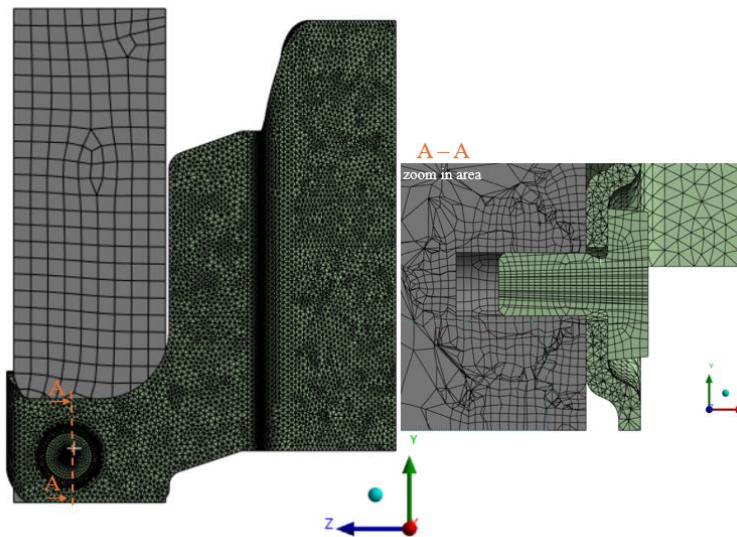


Fig. 2 – Mesh detail.

3.2. Contact definition

Multiple contact pairs were defined to represent the interaction between assembly components (Fig. 3). Contact interfaces were established between the bolt head and bracket, the bracket and support, and the bolt shank and mounting hole.

The contact formulation allows load transfer between the components and enables the evaluation of contact pressure evolution during thermal cycling. Frictional effects may be included depending on the simulation objectives and available experimental data.

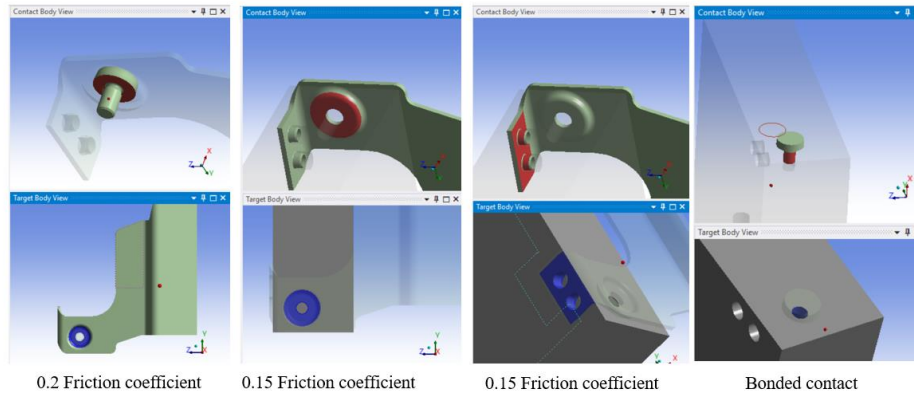


Fig. 3 – Body Contacts.

3.3. Boundary Conditions

The valve-block body was considered fixed, preventing translational and rotational motion. A bolt pretension load of 8 kN was applied to reproduce the assembly tightening process.

Following the application of pretension, thermal loads were introduced while maintaining the mechanical constraints. This loading sequence reproduces the actual assembly procedure encountered in industrial applications.

The numerical investigation have been performed using a sequential thermo-mechanical analysis. Initially, a bolt pretension force of 8 kN was applied to establish the clamping condition between the bracket and support.

Subsequently, the assembly was subjected to repeated thermal cycling ranging from -40°C to 120°C . A total of three complete thermal cycles were applied in order to evaluate the influence of repeated temperature variations on the stress state and structural response of the bolted joint.

The selected temperature limits are representative of severe environmental conditions frequently encountered in automotive and industrial applications.

Figure 4 presents the thermal loading profile adopted during the simulation.

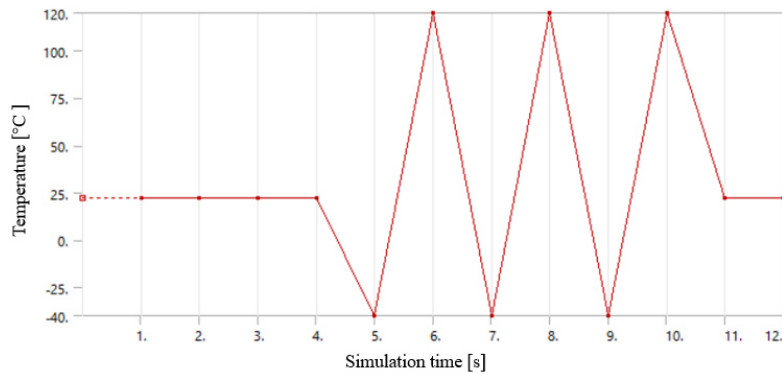


Fig. 4 – Thermal Condition.

4. Results and Discussion

4.1. Deformation and Stress Distribution after Bolt Pretension

The application of the 8 kN preload results in localized deformation of the bracket in the immediate vicinity of the bolted connection. The deformation pattern reflects the clamping effect generated by the bolt and the load transfer toward the support structure.

The maximum displacement occurs in the unsupported regions of the bracket, while the constrained areas exhibit limited movement. The deformation levels remain sufficiently small, confirming adequate structural stiffness of the assembly.

Figure 5 illustrates the deformation field generated by bolt tightening.

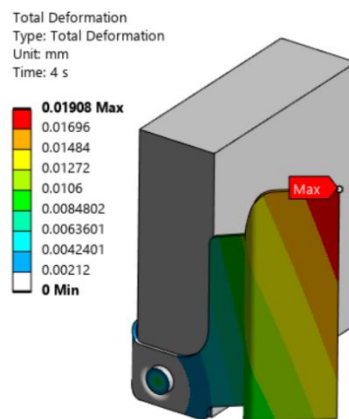


Fig. 5 – Total deformation.

The first stage of the analysis focused on the stresses generated exclusively by the bolt pretension load. The von Mises stress distribution indicates that the highest stress levels occur in the vicinity of the mounting hole and beneath the bolt head, where the load transfer between components is concentrated.

The obtained stress field confirms the presence of local stress concentrations associated with the geometric discontinuities of the bracket. Nevertheless, the stress values remain below the material yield strength, indicating elastic behavior of the assembly under the applied preload.

Figure 6 presents the von-Mises Stress distribution after tightening.

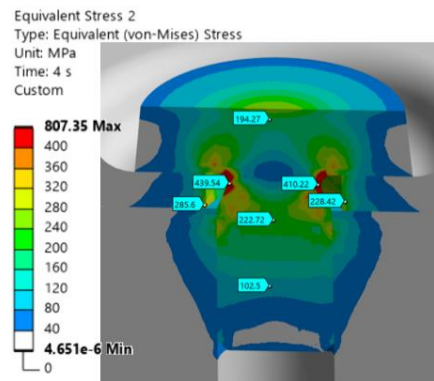


Fig. 6 – Equivalent Stress.

4.2. Thermal Stress Distribution at -40 and 120°C

The cooling stage produces thermal contraction of all components. Due to the geometric constraints and the interaction between connected parts, thermal stresses develop throughout the assembly.

At -40°C , elevated stress levels are observed around the bolt hole and in areas characterized by geometry transitions. Cooling the parts generates a reduction in their size, contraction leads to a decrease in the equivalent stress levels compared to the initial assembly condition.

During heating, all components experience thermal expansion. The resulting interaction between the constrained parts generates a redistribution of the contact pressure and internal stresses.

The analysis indicates that specific regions of the bracket are subjected to increased stress levels because thermal expansion is restrained by the bolted connection. The obtained stress field allows the identification of the most critical operating condition and potential failure initiation sites.

Figure 7 shows the stress distribution at a minimum and the maximum cycle temperature.

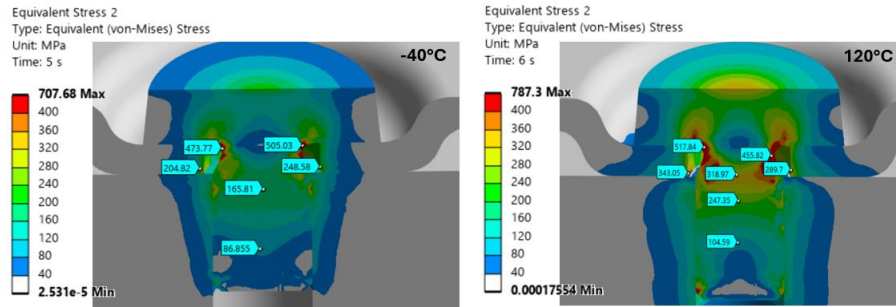


Fig. 7 – Equivalent Stress.

4.3. Evolution of Stresses during Thermal Cycling

The evolution of the maximum von Mises stress was monitored throughout the three thermal cycles. The results reveal cyclic variations in stress magnitude corresponding to the imposed temperature changes.

The stress response exhibits a periodic pattern associated with the heating and cooling stages. The repeatability of the stress levels from one cycle to another indicates a stable elastic response of the assembly and the absence of progressive degradation within the analyzed temperature range.

Figure 8 shows the stress evolution during the complete thermal cycling procedure.

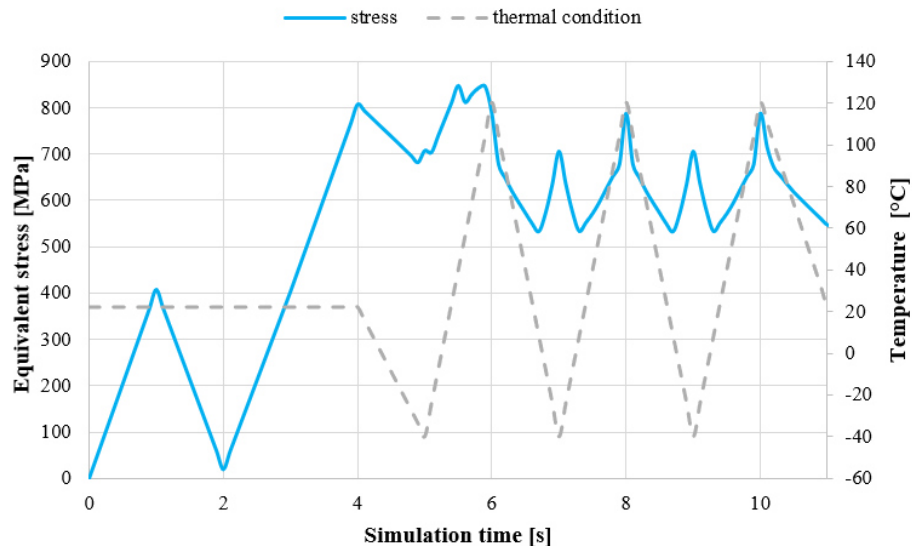


Fig. 8 – Equivalent Stress evolution.

4.4. Variation of Bolt Clamp Force

To assess the structural safety of the fastening system, the computed bolt loads can be compared with the proof load specified for metric fasteners. The proof load represents the maximum tensile load that a bolt can sustain without experiencing permanent plastic deformation and is defined in ISO 898-1: Mechanical properties of fasteners made of carbon steel and alloy steel. The proof load is derived from the proof stress of the fastener material and the tensile stress area of the thread.

The proof load can be calculated using Eq. (1):

$$F_p = S_p \cdot A_s \quad (1)$$

where: F_p - proof load [N], S_p - proof stress [MPa], A_s - tensile stress area of the threaded section [mm²].

The evolution of the bolt clamp force during thermal cycling is presented in Fig. 9. An initial preload of 8 kN was applied to the M6 property class 8.8 bolt prior to the thermal analysis. During the temperature excursions between -40°C and $+120^{\circ}\text{C}$, the axial bolt force exhibited periodic variations caused by the constrained thermal expansion and contraction of the assembled components.

The minimum clamp force recorded during the simulation was approximately 6.7 kN, whereas the maximum value reached 9.98 kN. Consequently, the thermal loading induced a preload variation of approximately 3.3 kN throughout the three thermal cycles.

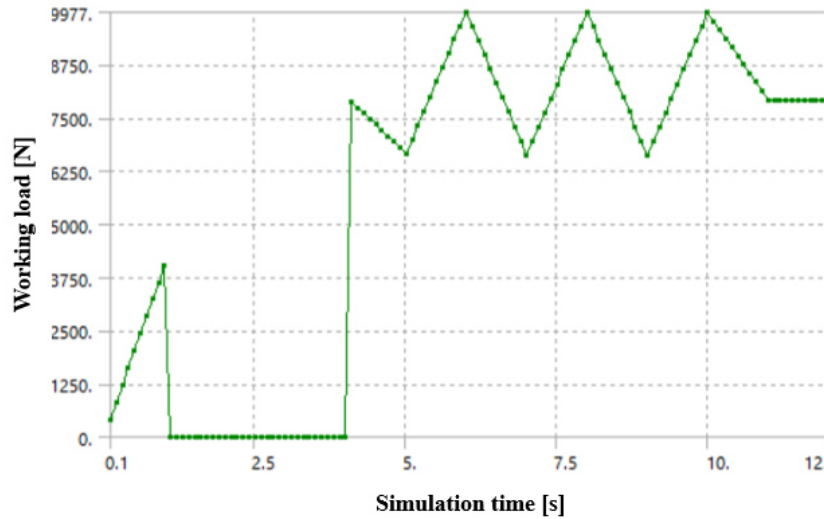


Fig. 9 – Working load during simulation.

To assess the structural integrity of the fastener, the predicted bolt load was compared with the proof load defined by ISO 898-1 for an M6 coarse-pitch bolt of property class 8.8 (Table 2).

Table 2
*Proof loads – ISO metric coarse pitch
thread – EN ISO 898-1:2013*

Thread <i>d</i>	Nominal stress area, <i>A_s [mm²]</i>	Property class 8.8 Proof load, <i>F_p [N]</i>
M6	20.1	11600

According to the standard, the corresponding proof load is 11.6 kN. The maximum bolt force obtained from the numerical simulation therefore represents approximately 86% of the proof load:

$$\frac{F_{max}}{F_p} = \frac{9977}{11600} = 0.86 \quad (2)$$

In Eq. (2) the resulting safety factor with respect to proof loading is approximately 1.16. Since the proof load was not exceeded at any stage of the thermal cycling sequence, the bolt remained within the elastic operating range and no permanent deformation is expected.

4.5. Contact Pressure Analysis

The contact pressure distribution between the bracket and valve-support was evaluated during all loading stages. The highest contact pressures occur directly beneath the bolt head and around the fastening region.

Temperature variations produce noticeable changes in contact pressure magnitude due to the thermal deformation of the assembled components. Monitoring these variations provides useful information regarding load transfer mechanisms and potential wear initiation areas.

Figure 10 illustrates the contact pressure distribution corresponding to the most critical loading condition.

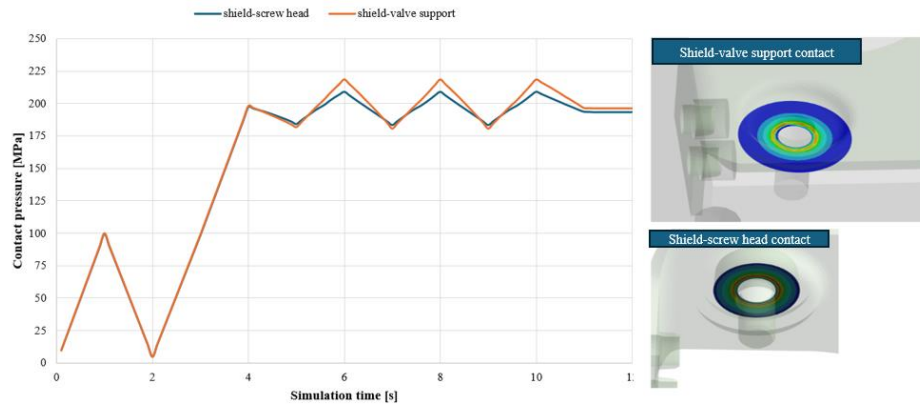


Fig. 10 – Contact pressure distribution.

4.6. Identification of Critical Regions

Based on the stress and deformation results, several critical regions were identified within the assembly. These locations include the mounting hole perimeter, the embossed reinforcement around the fastening area, and the geometric transitions characterized by abrupt curvature changes.

These regions exhibit the highest stress concentrations and therefore represent potential sites for crack initiation during long-term service. Future optimization efforts should focus on reducing stress concentration effects through local geometry modifications.

5. Conclusions

The thermo-mechanical behavior of a bolted bracket assembly subjected to an initial preload of 8 kN and repeated thermal cycling between -40°C and 120°C was investigated using finite element analysis. The results demonstrated that the highest stress levels occur in the proximity of the mounting hole and bolt contact regions. Bolt pretension generates significant local stresses while maintaining the assembly in the elastic domain.

Thermal loading produces additional stresses due to constrained thermal expansion and contraction. The maximum stress values are strongly dependent on the temperature level.

Contact pressure and clamp force vary throughout the thermal cycles. The identified critical regions represent potential locations for fatigue crack initiation during extended service conditions. The performed numerical methodology can be used for future design optimization and durability assessments of similar bolted structures.

REFERENCES

- Belhocine A., Abdullah O.I., *A thermomechanical model for the analysis of disc brake using the finite element method in frictional contact*, Journal of Engineering Research, 10(4), 128-141 (2021).
- Fukuoka T., *Finite Element Analysis of the Thermal and Mechanical Behaviors of a Bolted Joint*, Journal of Pressure Vessel Technology, 127(4), 402-407 (2005).
- Fukuoka T., Nomura M., Shino K., *Analysis of Heat Flow Around Bolted Joints and Variations of Axial Bolt Force*, Journal of Pressure Vessel Technology, 131(6), (2009).
- Guo N., Yun W., Zhang S., Du H., Xu C., *Finite Element Study on the Stiffness Variation Mechanisms of Radially Bolted Cylindrical-Cylindrical Shell Joints Under Transient Thermo-Mechanical Loading*, Aerospace, 13(1), 49-49 (2025).
- Machado J.P.F., Ribeiro R.E., *Thermo-Structural Analysis of a Brake Disc Project Through Finite Element Method*, SAE Technical Papers on CD-ROM/SAE Technical Paper Series, 1, 2025.
- Sasikumar A., Guerrero J.M., Quintanas-Corominas A., Costa J., *Numerical study to understand thermo-mechanical effects on a composite-aluminium hybrid bolted joint*, Composite Structures, 275, 114396-114396 (2021).
- Yang L., Li Y., Ding P., Zamankhan P., Cherng J.G., *Integrated Brake Squeal with Induced Thermal Stress Analysis*, SAE Technical Papers on CD-ROM/SAE Technical Paper Series, 1, 2017.
- Zhou W., Ai S., Chen M., Zhang R., He R., Pei Y., Fang D., *Preparation and thermodynamic analysis of the porous ZrO₂/(ZrO₂ + Ni) functionally graded bolted joint*, Composites Part B Engineering, 82, 13-22 (2015).
- Zika-Beyerlein B., Köstle S., Friedow T., *FE Analysis of Multi Layer Composite Shielding Systems*, SAE Technical Papers on CD-ROM/SAE Technical Paper Series, 1, 2005.

RĂSPUNSUL STRUCTURAL AL UNUI SCUT DE PROTECȚIE AL
SISTEMULUI DE FRÂNARE SUB SARCINĂ TERMICĂ

(Rezumat)

Lucrarea prezintă o analiză termică prin metoda elementelor finite (FEA) a unei protecții din tablă de oțel amplasate în compartimentul motor și fixate pe un bloc de valve prin intermediul unui șurub M6 clasa 8.8. Analiza a urmărit evaluarea comportamentului structural al ansamblului în condiții de solicitare termică specifice mediului de funcționare al vehiculului.

Modelul numeric a inclus aplicarea unei forțe de pretensionare de 8 kN în șurub, urmată de trei cicluri termice între -40°C și +120°C. Pe parcursul simulării au fost investigate distribuția tensiunilor echivalente Von Mises, deformațiile totale, presiunile de contact și variația forței de strângere din șurub.

Rezultatele au evidențiat apariția unor concentrații locale de tensiuni în zona găurii de fixare și în regiunile cu discontinuități geometrice. De asemenea, ciclurile

termice au determinat variații ale forței de strângere, însă valoarea maximă obținută a rămas sub limita de proof load prevăzută de standardul ISO 898-1 pentru șurubul analizat. În urma celor trei cicluri termice nu s-au observat pierderi semnificative ale pretensionării, ceea ce demonstrează că ansamblul își menține integritatea structurală și funcționalitatea în condițiile de exploatare investigate.