TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics, Mechanics, and Engineering
Vol. 69, Issue Special I, February, 2026

# STRUCTURAL AND FUNCTIONAL OPTIMIZATION OF A CONTROL STATION HOUSING

**Henrik HALMOSI, Dan LEORDEAN, Tudor SUHAN, Mircea Aurelian RUSU**

***Abstract:*** *This paper presents the constructive-functional optimization of a control station housing, with the objective of improving its mechanical performance and cost-efficiency through modern simulation techniques. The enclosure is designed to ensure explosion resistance and prevent the ingress of dust and liquids, withstanding internal pressures up to 10 bar. The methodology involved an initial assessment of the existing model, definition of working conditions, and the development of 3D models followed by static structural analyses. Based on the simulation results, design modifications were progressively implemented, including the addition of central lugs and adjustments to wall thickness, joint radii, and rib geometry. After conducting 44 static analyses, the optimized solution (Variant 6) demonstrated a significant reduction in equivalent stress from 374 MPa to 209 MPa, below the material yield strength of 220 MPa. Additionally, the weight was reduced by approximately 36%, from 2921.72 g to 1867.03 g. The final design ensures structural integrity under operational conditions while offering material and cost savings.*

***Key words:*** *Finite element analysis (FEA), Structural optimization, Explosion resistance, ATEX, ANSYS, Aluminium Alloy*

## 1. INTRODUCTION

Explosion-proof enclosures play a critical role in safeguarding electrical equipment and preventing catastrophic failures in hazardous environments where flammable gases, vapors, or dust are present. According to Kurowski (2011), the structural integrity of such enclosures is vital to ensure that they withstand internal explosion pressures without deforming or propagating the explosion to the surrounding atmosphere. The design of these enclosures is therefore governed by strict safety requirements, particularly when operating under elevated pressure conditions, where even minimal structural weaknesses can compromise operational safety and reliability [1].

Finite Element Analysis (FEA) has emerged as one of the most efficient tools for evaluating and optimizing the structural and functional aspects of explosion-proof housing. As noted by Roylance (2001), FEA enables engineers to simulate real-world conditions during the early stages of design, reducing the need for costly prototypes and physical testing [2]. Recent research, such as the study by Kumar (2021), demonstrates that ANSYS static structural analysis enables the detailed construction of 3D models, precise meshing of complex geometries, accurate definition of material properties, and the application of realistic loads and supports, allowing researchers to examine stress distribution, predict deformation, and validate the structural safety of engineering components under different working conditions [8].

Recent studies have highlighted the importance of optimizing enclosure design to meet stringent safety and cost-effectiveness criteria. Bhardwaj (2018) demonstrated that targeted material reduction and geometric modifications, guided by FEA, can lead to significant weight savings without compromising strength or rigidity [4]. Similarly, Wadkar et al. (2015) emphasized that advanced computational simulations not only validate structural safety under high pressure but also ensure compliance with standards like ASME Section VIII for pressure vessels [9]. These findings underline the relevance of combining

FEA with modern optimization techniques to achieve robust, lightweight, and economically feasible designs for industrial applications.

The present research builds upon these findings, aiming to optimize the structural design of a control station housing to resist an internal explosion pressure of 10 bar. By leveraging FEA simulations in ANSYS, the study evaluates different design iterations, focusing on reducing material usage while maintaining sufficient safety margins against plastic deformation. This work contributes to ongoing efforts in the field of industrial safety engineering to improve the reliability, efficiency, and cost-effectiveness of explosion-proof enclosures [3][4][9].

## 2. METHODOLOGY

This study employs a systematic approach to analyze and optimize the structural design of an explosion-proof control station housing. The methodology is based on computational simulations using Finite Element Analysis (FEA) to predict the mechanical behavior of the enclosure under internal pressure conditions of 10 bar. The process begins with defining the initial model geometry, material properties, and boundary conditions, followed by meshing and applying load cases representative of operational stresses. Using ANSYS software, static structural analyses are performed to identify critical stress concentrations, deformation patterns, and potential failure zones. Based on these results, iterative design modifications are proposed and tested to achieve improved performance, reduced weight, and compliance with safety requirements. This methodological framework ensures that the optimized enclosure design meets both functional and safety standards efficiently and cost-effectively.

### 2.1 Description of the Analyzed Product

The analyzed product in this study is the housing of an explosion-proof control station used in hazardous environments, compliant with ATEX standards (see Fig.1). Its primary function is to protect internal electrical and control components from external agents such as dust, liquids, and explosive gases, ensuring safe operation even in the event of an internal explosion.

The control station consists of two main cast aluminum parts: a lower body and an upper cover, both manufactured from EN-AC 42100 – a high-casting-quality aluminum alloy known for its excellent flowability, good mechanical properties, and corrosion resistance. These two parts are joined together using four stainless steel M6 screws and washers, ensuring mechanical stability and chemical durability in corrosive environments. To prevent deformation or failure during an internal explosion, the enclosure must withstand an internal pressure of 10 bar, with a maximum equivalent stress below the yield strength of the material (220 MPa).

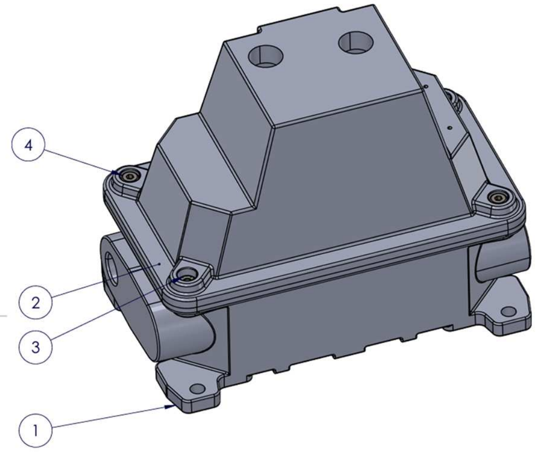

**Fig.1** Initial model: 1-lower housing, 2-upper housing, 3-washer, 4-screw

### 2.2 Initial Conditions and Assumptions

Before conducting the finite element analysis, a series of general initial conditions and engineering assumptions were established to simplify the simulation and reduce computational time.

It was assumed that the structure behaves as linearly elastic, with homogeneous and isotropic material properties. All components were considered as ideal solids, neglecting imperfections such as casting defects, microcracks, or assembly tolerances.

The model was tested under static loading conditions, without accounting for thermal effects, time-dependent behavior, or dynamic impacts. Contact between components was not simulated explicitly; instead, connections were assumed to be perfectly rigid or fixed, depending on the constraint configuration.

These simplifications are common practice in the early stages of design evaluation and allow for a fast and reasonably accurate prediction of stress distribution and deformation.

## 2.3 Modeling

The initial geometry of the housing was provided in STEP format by the collaborating company. This imported model served as the basis for all further design and analysis work. The model was fully rebuilt in SolidWorks to allow easier modifications and facilitate the optimization and simulation process.

## 2.4 FEA Setup

The finite element analysis was carried out using ANSYS Software, in a Static Structural environment. The goal of the simulation was to assess the stress and deformation behavior of the housing when subjected to internal pressure, and to guide the optimization process.

To carry out the simulation, the following setup steps and parameters were defined:

- Simulation type was linear static. The analysis focused on equivalent stress and total deformation to identify weak points and validate each design iteration.
- Fixed supports were applied to the surfaces corresponding to the bolt holes. These constraints replicated the real mounting configuration and prevented movement in all directions.
- Pressure loads of 1 MPa (equivalent to 10 bar) were applied uniformly to all internal surfaces of the housing. The pressure acted perpendicularly, simulating the effects of an internal explosion.
- Material properties were assigned based on EN-AC 42100 (AlSi7Mg0.3), including yield strength of 220 MPa. The material was considered homogeneous and isotropic.
- Meshing was performed using tetrahedral elements, providing a good balance between accuracy and computational efficiency. Local mesh refinement was applied in areas prone to stress concentration, such as sharp corners and fillets.

## 3. RESULTS AND INTERPRETATION

The finite element simulation of the initial design provided valuable information regarding the structural performance of the housing when subjected to internal pressure (see Fig.2). This preliminary analysis served as a baseline for identifying critical weaknesses and guiding future optimization steps.

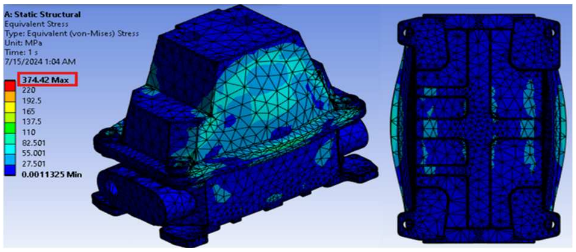

**Fig.2** Result of the initial model analysis

The key results obtained from the finite element analysis of the initial model are summarized below:

- The simulation revealed a peak stress of 374 MPa, significantly exceeding the material's yield strength of 220 MPa. This indicated that the structure would enter the plastic deformation range and fail under the specified pressure of 10 bar.
- The maximum total deformation reached approximately 0.48 mm, primarily located at the interface between the upper and lower parts of the enclosure. This suggested a risk of joint failure or loss of sealing integrity during operation.
- Elevated stress levels were identified at sharp internal corners and around mounting features, indicating the presence of stress concentrators. These localized areas of high stress are critical because they significantly increase the risk of crack initiation and structural failure, especially under repeated or extreme loading conditions.

The original design did not meet the safety requirements for explosion resistance. The excessive stress and localized deformation highlighted the need for structural reinforcement and redistribution of material.

## 4. PRODUCT OPTIMIZATION

The initial analysis of the control station housing revealed several critical structural deficiencies that prevented the enclosure from meeting the imposed safety requirements, particularly the ability to withstand an internal pressure of 10 bar without undergoing plastic deformation. Specifically, the initial equivalent stress of 374 MPa far exceeded the material's yield strength of 220 MPa, resulting in the risk of permanent deformation or structural failure. Moreover, significant asymmetrical stress distributions and localized deformation were observed, particularly at the interface between the upper and lower bodies of the housing, and around sharp geometric transitions acting as stress concentrators.

These issues highlighted the urgent need for comprehensive structural and functional optimization. The primary goals of the optimization process were:

- Reducing the maximum equivalent stress to below the yield point of the material (EN-AC 42100).
- Minimizing the total deformation of the enclosure under pressure loading.
- Evenly distributing internal stresses across the housing surfaces to avoid weak points.
- Eliminating stress concentrators caused by sharp corners or poor transitions.
- Reducing the overall weight of the housing while maintaining mechanical integrity.

### 4.1 Optimization Strategies & Design Variants Tested

To improve structural performance and reduce the overall weight of the enclosure, more than forty design variants were developed and analyzed using finite element simulations. From these, the six most significant design changes were selected for detailed evaluation. Each version introduced specific modifications based on the results of the previous analysis, with the goal of optimizing stress distribution, reducing deformation, and minimizing material usage. The following major design changes must be highlighted in the optimization process:

- Variant 1: In the first variant (see Fig.3), two additional lugs were added at the center of the enclosure to prevent the casing from opening under pressure. This change improved the stress distribution at the joint and reduced the deformation between the upper and lower bodies. The maximum equivalent stress decreased from 374 MPa to 200.2 MPa, falling safely below the yield limit. However, the added material increased the total weight from 2921.72 g to 2953.47 g, an increase of 31.75 g or +1.08%.

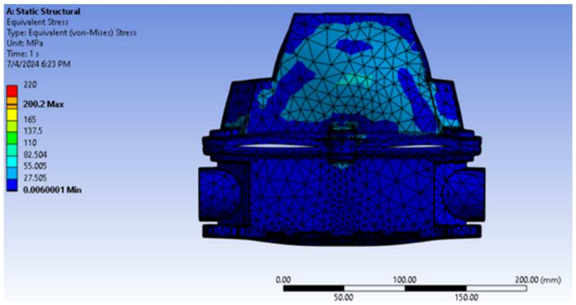

**Fig.3** Variant 1

- Variant 2 (Fig. 4) aimed to reduce stress concentrations by increasing the corner radii in the upper body, particularly in the regions adjacent to the ribs and internal geometric transitions. This approach effectively eliminated the previously identified internal stress hotspot, indicating an improved stress distribution in those areas. However, a new stress concentration developed near one of the added lugs, likely due to local stiffness changes and altered load paths. Consequently, the maximum equivalent (von Mises) stress slightly increased to 231.98 MPa, exceeding the material yield limit. The total deformation also increased compared to earlier variants, while the component weight remained essentially unchanged.

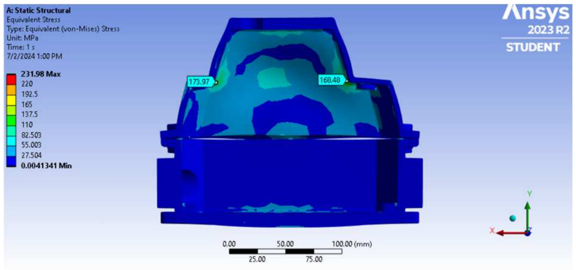

**Fig.4** Variant 2

- Variant 3: In the case of Variant 3 (see Fig.5), the focus was on reducing excessive weight and improving the load balance between the upper and lower components of the housing. To achieve this, the wall thickness of the lower body was significantly reduced. The side walls, base, and transition areas were thinned to the manufacturable minimums of 3–3.5 mm. As a result of these changes, the equivalent stress increased slightly to 239 MPa; however, the overall stress distribution between the two halves of the housing improved. This modification led to a substantial weight reduction, lowering the total mass from 2953.47 g to 2131.92 g, a decrease of 821.55 g, or 27.81%.

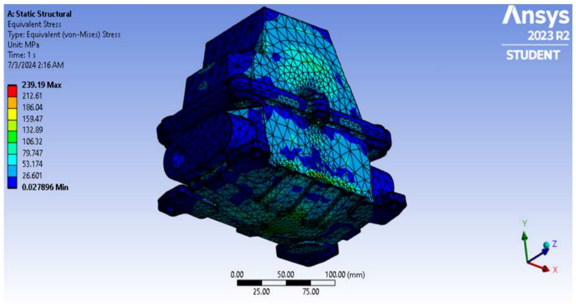

**Fig.5** Variant 3

- Variant 4: In the fourth variant (see Fig.6), the lower body was reinforced by redesigning the internal rib structure. The main longitudinal rib was widened by 4 mm, and additional horizontal ribs were added. These modifications decreased the equivalent stress to 231 MPa (slightly above the yield limit) and reduced total deformation from 0.475 mm to 0.29 mm. However, the weight increased slightly to 2204.18 g, a gain of +3.38% compared to Variant 3.

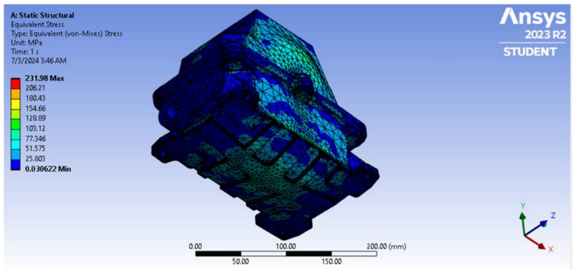

**Fig.6** Variant 4

- Variant 5: In this variant (see Fig.7), the optimization targeted the upper housing, specifically by reducing wall thickness in areas that were not significantly stressed. The longitudinal walls were thinned from 5 mm to 4 mm, the transverse walls from 5 mm to 3 mm, and the sloped surfaces from 5 mm to 3 mm. Additionally, the top surface thickness was reduced from 13.5 mm to 6 mm. Several fillet radii (1.5 mm, 3 mm, and 5 mm) were tested to maintain smooth transitions and minimize stress concentrations. These adjustments led to a further weight reduction, while the structure retained its mechanical integrity under 10 bar pressure. Although the stress level remained close to the yield limit, it was still acceptable, and deformation stayed within safe limits.

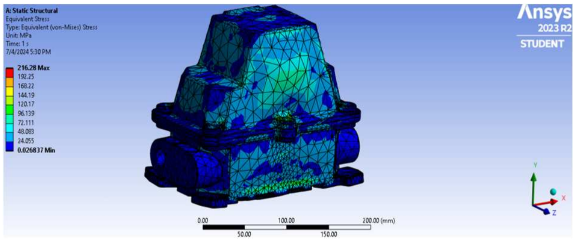

**Fig.7** Variant 5

- Variant 6: This variant (see Fig. 8) combined all previous design improvements into a single model. The mid-section lugs were retained for better fixation, and both upper and lower body wall thicknesses were reduced where feasible. Fillet radii were increased at critical transitions to minimize stress concentrations and improve durability. The rib structure was refined to enhance load distribution, including thicker and additional ribs on the lower housing.
- These cumulative changes resulted in an equivalent stress of 209 MPa, which remained below the material's yield strength, ensuring safe operation. The total weight of the enclosure decreased to 1867.03 g, representing a reduction of 1054.69 g compared to the original design, improving both efficiency and manufacturability.

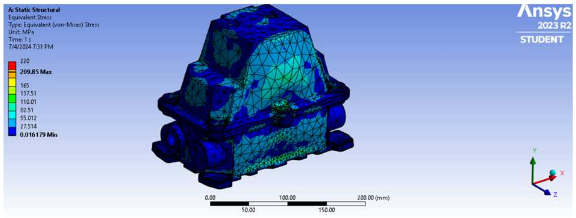

**Fig.8** Variant 6

## 4.2 Justification of the final solution

After a series of targeted design changes and simulations, a total of six different variants were developed and evaluated. Through this iterative optimization process, the design was gradually refined in terms of geometry, material distribution, and mechanical performance. This process ultimately led to the final version of the housing (see Fig.9).

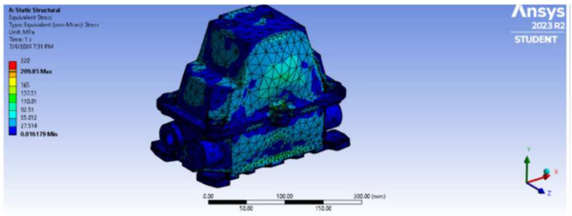

**Fig.9** Final model

- The maximum equivalent stress of 209 MPa is below the yield limit of 220 MPa. As long as this limit is not exceeded, the variant withstands the internal pressure of 10 bar (1 MPa) and is regarded as adequate.
- A significant 36% weight reduction, from 2921.72 g to 1867.03 g, was achieved, which translates to cost savings and material efficiency.
- Stress was evenly distributed, avoiding local concentration and potential failure zones.
- Added fixing lugs and improved rib geometry increased resistance to deformation during explosion-like loading.

## 5. CONCLUSIONS

This study focused on the constructive and functional optimization of an explosion-proof control station housing using Finite Element Analysis. Through extensive simulations and iterative design improvements, the research demonstrated how structural modifications can significantly enhance strength, reduce weight, and ensure safety under high internal pressure conditions. The following sections provide a summary of the main findings, discuss the practical relevance of the proposed solution, discuss the study limitations, and outline recommendations for future research directions.

### 5.1 Summary of Main Findings

The study successfully achieved its primary goal: the constructive-functional optimization of a control station housing capable of withstanding 10 bar internal pressure. Throughout the study, a total of 44 design variants were developed and analyzed using Finite Element Analysis (FEA). Each version addressed specific structural limitations such as stress concentration, excessive deformation, and unnecessary mass.

The final optimized version reduced the maximum equivalent stress from 374 MPa to 209 MPa, which is below the material's yield strength of 220 MPa. At the same time, one of the most impactful outcomes was the substantial reduction in total weight, from 2921.72 g in the original version to 1867.03 g in the final design. This represents a total reduction of 1054.69 g, or approximately 36%, making the product more efficient in terms of both material usage and manufacturing cost. Additionally, deformation was lowered by 39%, and the stress distribution was significantly improved across the structure.

### 5.2 Practical Relevance of the Proposed Solution

The optimized control station housing design holds substantial practical relevance for industrial applications, particularly in hazardous ATEX (explosive atmosphere) environments such as oil refineries and chemical plants. Its enhanced impact resistance directly addresses critical safety concerns in these high-risk settings.

By ensuring the structural integrity of the cover under significant impact forces, the risk of internal cracking is effectively minimized. Such cracking could otherwise lead to spark propagation, which poses a severe threat in ATEX environments as it could trigger catastrophic explosions. The robust design thus serves as a crucial safeguard against potentially devastating incidents, significantly enhancing the safety of personnel and infrastructure.

Furthermore, the achieved weight reduction, approximately 36% compared to the initial design, translates into tangible economic benefits. This reduction directly contributes to lower material consumption during manufacturing and decreased production costs. Therefore, this optimized solution not only elevates safety and operational reliability in high-risk industrial environments but also offers economic advantages through efficient resource utilization. The successful integration of improved impact resistance and reduced material usage makes this design a valuable contribution to the field of industrial safety and engineering.

### 5.3 Study limitations

One important limitation of this study is that only a static structural analysis was performed using finite element methods. The simulation evaluated the behavior of the housing under constant internal pressure, without considering dynamic effects such as impact loads, vibrations, or pressure fluctuations over time. These dynamic factors could influence real-world performance, especially in explosion scenarios or during transportation and handling. As a result, further investigation using transient or modal analysis is recommended to fully validate the enclosure's structural integrity under a wider range of operating conditions.

### 5.4 Suggestions for Future Research

To further improve the structural performance and reliability of the enclosure, future research could focus on several key directions:

- While the current work focused exclusively on static structural analysis, future research could explore the behavior of the enclosure under dynamic loads, such as impact, shock, or vibration. These simulations would provide a better understanding of how the housing performs in real-life operating environments, especially in applications where sudden mechanical shocks or explosions may occur.
- In this study, the initial model was improved and optimized based on existing geometry. A valuable future step would be to design the housing from scratch, using only the functional and safety requirements provided by the company. This would allow greater design freedom and potentially lead to a more compact, efficient, and cost-effective solution tailored to the production and certification constraints of the manufacturer.
- Another promising research direction would be to replace the current aluminum alloy (EN-AC 42100) with alternative materials that offer higher mechanical strength, improved thermal resistance, or better fatigue performance. Running the same FEA simulations on different materials would allow a comparative evaluation, helping identify the most suitable material for performance-critical applications in ATEX environments.

## 6. REFERENCES

[1] Kurowski, P. M. *Finite Element Analysis for Design Engineers. Third Edition*, SAE International, books.google.com, 2022.

[2] Roylance, D., *Finite Element Analysis*, Department of Materials Science and Engineering, MIT, 2001.

[3] Neelakandan, V., Ganesan, T., & Rao, P. C. *Weight Optimization of Housing Bracket for Electrical Starter Motor using FEA*, Procedia Structural Integrity, 2019.

[4] Bhardwaj, S., Ashok, B., Lath, U., Agarwal, A., *Design and Optimization of Steering Upright to Reduce the Weight Using FEA*, SAE International, International Conference on Advances in Design, Materials, Manufacturing and Surface Engineering for Mobility, 2018.

[5] Patel, B. P., Prajapati, J. M., Parmar, P. S. *Review on FEA and Genetic Algorithm Approach for Structural Optimization*, Global Journal of Mechanical Engineering and Computational Science, 2011.

[6] Albadrani, M. A., Almutairi, A. D., *Study on the Mechanical Compression Properties of a New Carbon-Free Plaster Using ANSYS Software*, Buildings, 2023.

[7] Callister, W. D., & Rethwisch, D. G., *Materials science and engineering: An introduction*, Philadelphia University,

Faculty of Engineering, Department of Mechanical Engineering, 1999.

[8] Kumar, A. C. R., Raffic, N. M., Babu, K. G., Selvakumar, S. *Static Structural Analysis of Spur Gear Using ANSYS 15.0 and Material Selection by COPRAS, MOORA Techniques*, Materials Today: Proceedings, Vol. 47, 2021.

[9] Wadkar, V. V., Malgave, S. S., Patil, D. D., Bhore, H. S., Gavade, P. P., *Design and Analysis of Pressure Vessel Using Ansys*, J. Mech. Eng. Technol., 2015.

[10] Directive 2014/34/EU of the European Parliament and of the Council on equipment and protective systems intended for use in potentially explosive atmospheres (ATEX).

[11] IEC 60079 series, Explosive atmospheres – Standards for equipment protection.

[12] EN 62262:2002, Degrees of protection provided by enclosures for electrical equipment against external mechanical impacts (IK code).

## Optimizarea structurală și funcțională a carcasei unei stații de control

Rezumat: Această lucrare prezintă optimizarea structurală și funcțională a carcasei unei stații de control utilizate în medii periculoase, unde există risc de explozie. Obiectivul principal a fost îmbunătățirea performanțelor mecanice și reducerea masei carcasei prin analiza cu element finit, utilizând software-ul ANSYS. Studiul începe cu o versiune inițială care nu îndeplinește criteriul de siguranță impus, urmată de mai multe iterații de proiectare ce implică modificări succesive ale geometriei, grosimii pereților și nervurilor de rigidizare, precum și introducerea unor urechi de prindere. Rezultatul final constă într-o carcasă cu tensiunea echivalentă redusă de la 374 MPa la 209 MPa și o scădere a greutății cu 36%. Studiul demonstrează eficiența analizei cu element finit în obținerea unor îmbunătățiri semnificative atât în ceea ce privește siguranța, cât și eficiența, și subliniază importanța acesteia în contextul certificării ATEX.

**Henrik HALMOSI,** Engineer, Graduate student, Faculty of Industrial Engineering, Robotics and Production Management, Technical University of Cluj-Napoca, henrikhalmosih@gmail.com, Foieni, Romania, +40747500189.

**Dan LEORDEAN,** PhD, Associate Professor, Faculty of Industrial Engineering, Robotics and Production Management, Technical University of Cluj-Napoca, Department of Manufacturing Engineering, dan.leordean@tcm.utcluj.ro, +40264415653, +40744810091, Cluj-Napoca, Romania.

**Tudor SUHAN,** Engineer, Engineering Director Europe, Emerson - Appleton Group, tudor.suhan@emerson.com, +40749154515, Balastierei St., No. 1, 407310 Gilău, Cluj, Romania.

**Mircea Aurelian RUSU**, PhD Dr. Eng., Teaching assistant, Faculty of Industrial Engineering, Robotics and Production Management, Technical University of Cluj-Napoca, Department of Manufacturing Engineering, mircea.rusu@tcm.utcluj.ro, +40744559188, Cluj-Napoca, Romania.