ADVANCED STRESS ANALYSIS AND INTERPRETATION OF THE BULLDOZER BLADE USING ABAQUS AND PYTHON SCRIPTS

FLORIN VÎLCEANU[1], STELA DINESCU[2]

Abstract: This paper presents an advanced methodology for the analysis and interpretation of the stress state in a bulldozer blade, based on the finite element method and numerical post-processing of the results. The structural analysis is performed in the Abaqus environment, assuming linear elastic material behavior and loading conditions representative of the actual working process. In order to achieve an extended exploitation of the results obtained from FEM simulations, a dedicated set of Python scripts is developed for exporting and processing data from .odb files. These scripts enable the statistical analysis of equivalent Von Mises stresses, the determination of principal stresses, and the graphical representation of Mohr's circles for structurally critical points. The proposed approach allows a detailed interpretation of the blade's structural behavior and highlights the advantages of automated post-processing in safety assessment and structural optimization of earthmoving equipment.

Keywords: bulldozer blade, finite element method, Abaqus, numerical post-processing, Python, Von Mises stresses, Mohr's circle

1. INTRODUCTION

Bulldozer blades are essential working components of earthmoving machinery, being subjected to high mechanical loads generated by direct interaction with the soil during pushing, leveling, and excavation operations. In service conditions, these components experience significant variations in applied forces, determined by the nature of the processed material, operating conditions, and the geometry of the working tool. Consequently, an accurate evaluation of the stress state and structural behavior of the blade represents a critical aspect for ensuring operational safety and the durability of the equipment.

The finite element method (FEM) is widely used in the structural analysis of

[1] *Lecturer Ph.D. Eng., University of Petroşani, FlorinVilceanu@upet.ro*
[2] *Assoc. Prof. Ph.D. Eng., University of Petroşani, StelaDinescu@upet.ro*

heavy machinery components due to its capability to model complex geometries and to highlight stress and displacement distributions in structures subjected to realistic loading conditions [1–2]. Abaqus software is recognized for its robustness in performing advanced numerical analyses, offering precise definitions of loading conditions, contact interactions, and material behavior. However, the standard results provided by the integrated post-processing environment are often limited to general graphical representations, which do not always allow a detailed and quantitative assessment of the stress state in structurally critical regions [3–4].

In this context, advanced post-processing of FEM results becomes an essential stage in structural behavior evaluation. The use of the Python programming language, natively integrated into the Abaqus environment, enables the development of customized tools for extracting, organizing, and numerically analyzing data from .odb result files. Through these scripts, additional analyses can be performed, such as statistical distributions of stresses, determination of principal stresses, and evaluation of the stress state using Mohr's circle representations, which provide deeper insight into the structural loading mechanisms.

This paper aims to present an advanced methodology for the analysis and interpretation of stresses in a bulldozer blade, based on numerical simulations performed in Abaqus and automated post-processing of the results using Python scripts [5]. The study focuses on identifying critical structural zones, evaluating the stress level with respect to material limits, and highlighting the advantages offered by integrating numerical post-processing into the structural analysis workflow. The proposed approach can be extended and adapted to other components of earthmoving equipment, contributing to structural optimization and increased operational safety. The numerical analysis was carried out using Abaqus 2025 (Dassault Systèmes), provided under the academic license of the University of Petroșani.

2. NUMERICAL MODEL AND ANALYSIS CONDITIONS

The numerical analysis of the structural behavior of the bulldozer blade was carried out using the finite element method, with the aim of evaluating the stress state and structural response under representative operating conditions. The numerical model was developed in the Abaqus 2025 environment and configured to allow subsequent advanced post-processing of the results using Python scripts.

2.1 Geometric model

The three-dimensional geometric model of the bulldozer blade was created based on the real constructive configuration of the working tool, taking into account the overall shape, characteristic thicknesses, and stiffening elements.

The analyzed blade features a curved–segmented structural configuration, characteristic of modern bulldozers used in heavy earthmoving operations, snow removal, or bulk material handling. This blade consists of a curved central body and two inclined lateral extremities (side wings), connected by welded segments, ensuring structural continuity and overall rigidity of the assembly, as illustrated in Figure 1.

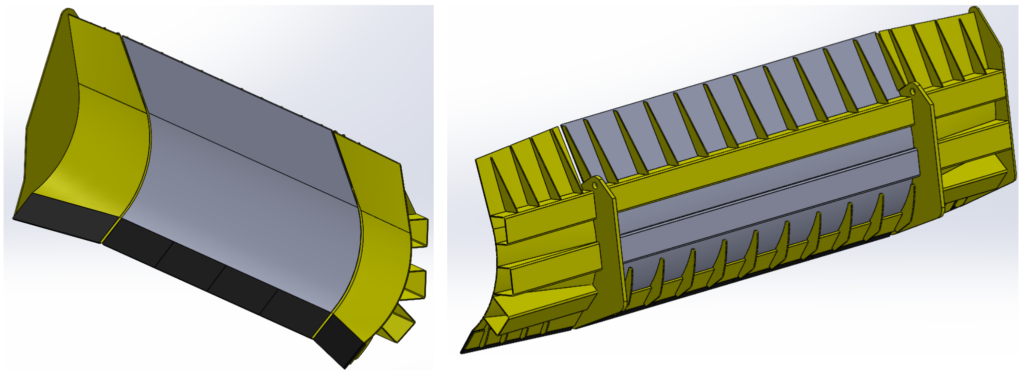


Fig. 1. Curved-type bulldozer blade

The curved frontal shape of the main blade panel is designed to reduce forward resistance and to ensure efficient rolling of the pushed material, minimizing friction between the soil and the blade surface. This curvature is optimized to channel the material mass toward the central area, thereby increasing stability during the pushing operation.

The lower part of the blade is equipped with a segmented, bolted wear plate (cutting edge), manufactured from high-hardness steel, serving both a protective function and allowing easy replacement in case of wear. The mechanical bolted connection ensures simplified maintenance and adaptability to varying ground conditions.

The lateral end caps feature a closed geometry that prevents material spillage toward the sides, increasing the effective volume of material displaced during each pass. In addition, these elements contribute to structural stiffening in the corner regions, which are known to be critical zones under mechanical loading.

The bulldozer blade is manufactured from structural steel grade S355, a material commonly used for earthmoving equipment working tools due to its favorable balance between strength, ductility, and weldability. In the numerical analysis, the material behavior was modeled as linear elastic isotropic, characterized by the Young's modulus and Poisson's ratio corresponding to this steel grade. The assumption of elastic behavior is justified by the analyzed stress levels, which remain below the yield limit of the material.

2.2 Boundary conditions and loading

The first stage of the analysis involved the development of a simplified computational model, such that the employed geometry accurately reflects the essential constructive form of the bulldozer blade, while avoiding unnecessary details that could increase the computational cost. To this end, the three-dimensional model was imported and adapted to ensure compatibility with finite element simulation methods, as illustrated in Figure 2.

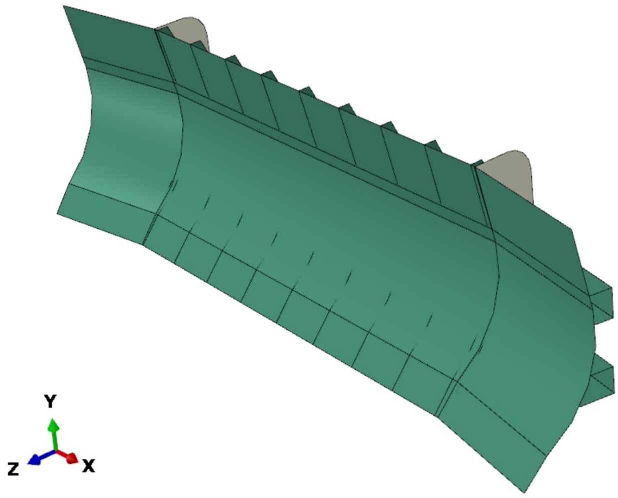


Fig. 2. Bulldozer blade modeled using shell surfaces

The structural geometry was represented using shell-type entities (surfaces), which allow an efficient description of thin components such as the blade plate and the stiffening ribs. The selection of this element type was based on the relatively small thickness of the components compared to their other geometric dimensions, enabling an accurate representation of the mechanical behavior while maintaining a reduced computational cost.

The pressure applied on the active face of the blade was introduced as a distributed load acting over the entire curved surface, thereby simulating the actual loading conditions occurring during the soil pushing process.

The magnitude of the applied pressure was determined as a function of the type of displaced material (clay, marl, sandy soil), based on a detailed analytical calculation.

The calculated pressure values were obtained by considering the working surface of the blade, defined as the contact area between the blade and the displaced material.

Accordingly, for the three analyzed soil types, the corresponding pressure values distributed over the blade surface were determined and are presented in Table 1 [6].

Table 1. **Operating pressure values corresponding to the working regimes**

Soil type	**Applied pressure *p* [MPa]**
Clay	0.0118
Marl	0.0099
Soil (sandy/loamy)	0.0079

To simulate the connections between the different components of the assembly (stiffening ribs, support bars, and reinforcements), *tied contact* interaction conditions were defined. This assumption allows the assembly to be modeled as a welded structure, preventing any relative sliding or separation between the connected parts. In the rear region of the blade, the contact interactions were configured to ensure proper force transmission between the reinforcing elements and the main plate, accurately reflecting the actual mechanical fastening or welded assembly conditions.

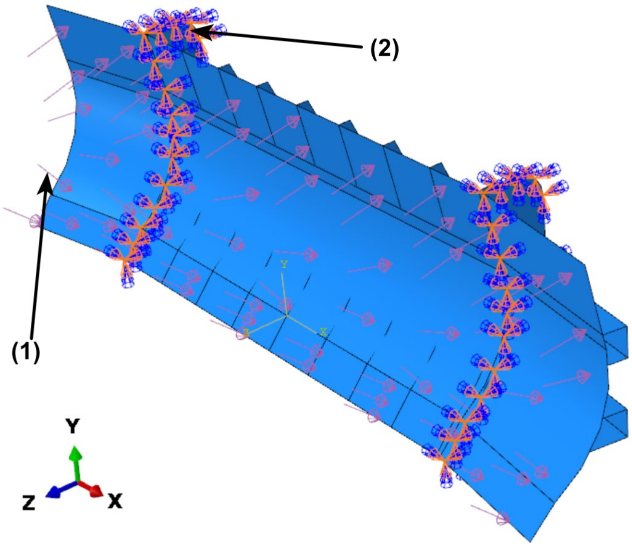


Fig. 3. Loading and boundary conditions: 1 – pressure loading applied on the working surface of the blade; 2 – support conditions of the bulldozer blade

The support conditions were imposed at the blade attachment regions, corresponding to the articulation points with the bulldozer frame (2). These boundary conditions were defined in such a manner as to allow rotation or limited displacement, in accordance with the actual degrees of freedom of the structure during operation, as illustrated in Fig. 3.

Through this modeling approach, the finite element model provides a clear representation of the stress distribution and maximum deformations within the structure, enabling both the verification of global structural strength and the identification of critical zones that may require constructive optimization.

2.3 Model discretization

The discretization of the geometric model was performed using finite elements of shell type S4, i.e., linear quadrilateral elements with four nodes, which are well suited for the analysis of three-dimensional structures with complex geometries and for accurately capturing stress gradients, as shown in Fig. 4.

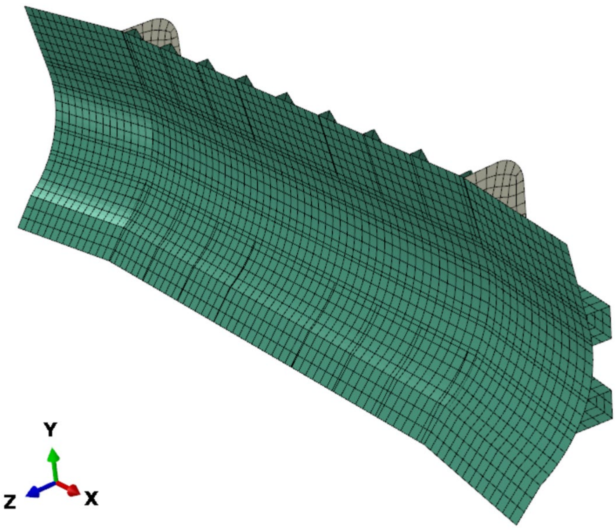


Fig. 4. Finite element discretization of the bulldozer blade using shell elements

The finite element mesh was adapted to the geometry of the blade, with local refinement applied in areas considered potentially critical from the standpoint of stress concentration, such as the cutting edge, corners, and structural joints. The selection of the element size aimed to achieve an optimal compromise between result accuracy and computational cost.

2.4 Type of analysis

The numerical analysis was performed under a linear static regime, an approach that is appropriate for evaluating the stress state of the bulldozer blade under quasi-static conditions, characteristic of soil pushing and displacement processes. Static analysis allows the determination of stress distributions and structural displacements under the assumption of elastic material behavior and slowly varying applied loads.

In order to highlight the influence of the displaced material properties on the structural behavior of the blade, the analysis was carried out for three distinct soil types, representative of real operating conditions: clayey soil, marl, and sandy soil. In each case, the loading was defined as a distributed pressure applied over the entire curved active surface of the blade, simulating the actual stresses generated during the soil pushing process.

The magnitude of the applied pressure was determined based on a detailed analytical calculation, as a function of the type of displaced material and the contact surface area between the blade and the soil, as presented in Table 1.

3. METHODOLOGY FOR FEM RESULTS POST-PROCESSING USING PYTHON

The results obtained from finite element analyses contain a large volume of information which, in standard post-processing workflows, is usually limited to global graphical representations and extreme values of the analyzed quantities. For an in-depth assessment of the structural behavior of the bulldozer blade, additional data processing is required in order to enable a detailed quantitative analysis of the stress state in the critical regions of the structure.

For this purpose, an advanced post-processing methodology was developed in this study, based on the use of the Python programming language, which is natively integrated within the Abaqus software environment. This approach enables the automation of the extraction, organization, and analysis of results stored in *.odb* files, providing significantly greater flexibility compared to standard post-processing tools [5].

In order to ensure a coherent and detailed interpretation of the results obtained through the finite element method, the proposed Python-based post-processing methodology extends the standard capabilities of the Abaqus post-processing environment by enabling numerical extraction and processing of stress fields generated by FEM simulations. The methodology is designed to address both the global evaluation of the stress state and the local analysis of critical points, thus providing a unified framework for advanced interpretation of the numerical results.

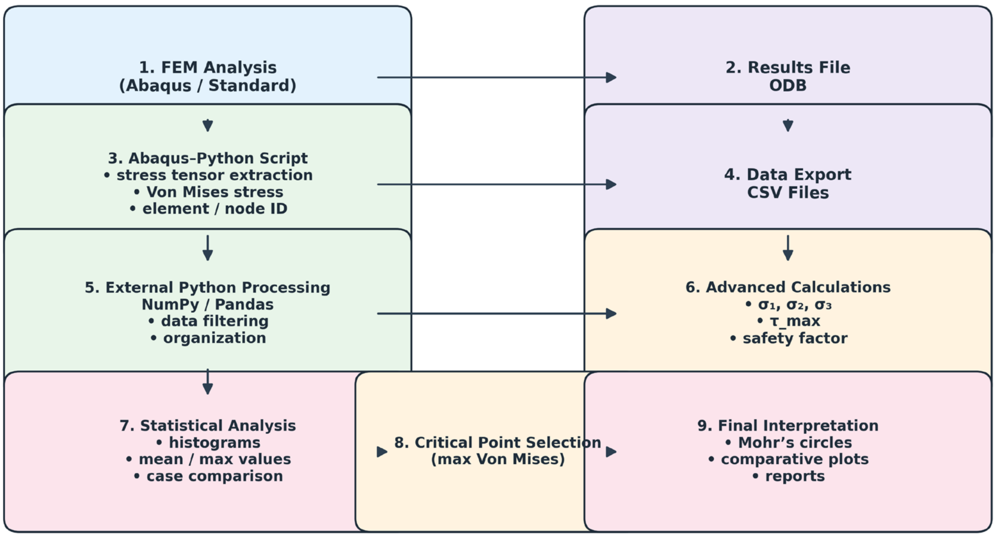


Fig. 5. Methodology for FEM results post-processing using Python scripts

The logical flowchart presented in Fig. 5 highlights the sequence of post-processing stages, starting from the structural analysis performed in the Abaqus environment and continuing with the extraction of results from *.odb* files. By means of

Python scripts integrated within Abaqus, the relevant data are exported and subsequently processed in an external Python environment to perform advanced calculations, including statistical analysis of Von Mises equivalent stresses, determination of principal stresses, computation of the maximum shear stress, and representation of Mohr's circles. This methodology enables an in-depth evaluation of the stress state in the bulldozer blade and provides the necessary support for identifying critical zones and formulating conclusions regarding the structural behavior under real operating conditions.

3.1 Data extraction from *.odb* files

The first step of the methodology consists of accessing the result files generated by Abaqus and extracting the information relevant to the structural analysis. Using the *odbAccess* module, stress values at the finite element level were read, corresponding to the last analysis increment, which was considered representative of the maximum loading condition.

The developed Python scripts enable:

- ■ access to the stress field output;
- ■ selection of the domain of interest;
- ■ extraction of the stress tensor components;
- ■ computation of the equivalent Von Mises stress values.

The extracted data are organized and exported into CSV files, a structure that facilitates subsequent numerical processing and the performance of statistical analyses.

3.2 Structure of data files and numerical processing

The generated CSV files contain, for each analyzed integration point or node, information related to:

- • element identification;
- • stress tensor components;
- • equivalent Von Mises stress value.

This data structure allows the execution of customized analyses independent of the Abaqus environment, using Python libraries dedicated to numerical computation and graphical representation. Through these tools, the raw data are filtered, sorted, and analyzed to identify critical points and characteristic distributions of the mechanical quantities.

3.3 Export of numerical data from Abaqus

After extracting the relevant information from the *.odb* files, the numerical data were organized and exported into an open format compatible with further processing in external computational environments. The data export was performed using Python scripts executed within Abaqus, which provide direct access to the result fields and allow their storage in CSV files.

For each analyzed point, the data files include information regarding element identification, stress tensor components, and the equivalent Von Mises stress value. This data structure ensures traceability of the FEM results and allows subsequent correlation of numerical values with their spatial location in the discretized model.

The use of the CSV format provides independence from the Abaqus environment and enables easy import of the data into specialized Python libraries for numerical computation and statistical analysis. Through this stage, the transition is achieved from the raw results generated by the FEM solver to a structured dataset prepared for advanced quantitative evaluations and comparative analyses between different loading cases.

The automated export of numerical data contributes to increased reproducibility of the analysis, reduces errors associated with manual intervention, and enables the extension of the proposed methodology toward additional studies, such as durability assessment, probabilistic analysis, or parametric and comparative investigations.

4. ADVANCED ANALYSIS OF THE STRESS STATE THROUGH NUMERICAL POST-PROCESSING

For an in-depth evaluation of the mechanical behavior of the bulldozer blade, the results obtained from the finite element analysis were subjected to an advanced post-processing stage performed using custom-developed Python scripts. This stage extends the classical analysis provided by the Abaqus environment by enabling the detailed extraction and numerical processing of the stress tensor at each analyzed point.

Within the post-processing procedure, equivalent Von Mises stresses were evaluated for each finite element, as this criterion is commonly used to assess the stress state relative to the material yield limit. Based on their distribution, statistical analyses were carried out, including maximum, minimum, and mean values, as well as frequency histograms, providing a global overview of the structural loading level and the uniformity of the stress field.

Furthermore, starting from the stress tensor components, the principal stresses σ_1, σ_2, and σ_3, as well as the maximum shear stress τ_max, were determined. These parameters are essential for evaluating potential failure mechanisms and for identifying critical regions subjected to high stress concentrations. The computation of principal stresses enables the assessment of the actual stress state at each structural point, independently of the chosen coordinate system.

For an intuitive interpretation of the stress state, graphical representations in the form of Mohr's circles were generated for the points exhibiting the highest equivalent stress values. These representations provide a clear visualization of the relationship between normal and shear stresses and allow a rapid assessment of the maximum shear stress levels that may lead to the initiation of failure mechanisms.

By integrating these analysis methods—Von Mises stress statistics, determination of principal stresses, and Mohr's circle representation—a comprehensive characterization of the stress state in the bulldozer blade is achieved, going beyond conventional FEM analysis. This approach enables the identification of critical zones, the evaluation of structural safety margins, and provides solid support for result interpretation and for formulating conclusions regarding the mechanical behavior of the blade under real operating conditions.

The proposed post-processing methodology offers several advantages compared to classical approaches, among which the following can be highlighted:

- automation of the FEM result analysis process;
- capability to perform advanced and customized analyses;
- independence from the standard graphical post-processing environment;
- increased level of detail and accuracy in stress state interpretation.

By integrating numerical post-processing using Python, the structural analysis of the bulldozer blade acquires an extended character, focused not only on obtaining results but also on their in-depth interpretation, with direct applications in safety assessment and constructive optimization.

4.1 Finite Element Analysis Results

Following the numerical simulations performed using the finite element method, stress and deformation distributions were obtained for all analyzed configurations corresponding to different types of material pushed by the blade (clay, marl, and sandy soil). The analysis was carried out under linear static conditions, assuming ideal elastic behavior of the blade material.

The evaluation of the stress state was based on the Von Mises criterion, commonly employed for assessing the strength of ductile materials. Accordingly, for each loading case, a distribution map of equivalent Von Mises stresses was generated, highlighting the regions subjected to the highest stress levels.

The results are presented in both graphical and numerical forms, enabling rapid identification of stress concentrations and assessment of the influence of the applied pressure on the structural behavior. The maximum Von Mises stress values were compared with the admissible limit of the material in order to verify compliance with operational safety requirements. The blade material is S355 structural steel, widely used in the construction of earthmoving equipment blades.

4.1.1. Classical FEM Analysis Method – Stress and Displacement Maps Obtained in Abaqus

Within the classical finite element analysis framework, the structural behavior of the bulldozer blade was evaluated using the standard post-processing tools available in the Abaqus environment. This approach allows the direct generation of equivalent Von Mises stress distribution maps and displacement fields, providing a global overview of the structural stress state for each analyzed loading case.

The resulting graphical representations highlight regions with high stress concentrations as well as areas characterized by maximum displacements, facilitating the identification of sensitive points within the blade structure. The classical FEM approach is widely used in design and verification stages due to the simplicity of visual interpretation and the possibility of direct correlation with the discretized model geometry.

However, this approach is mainly limited to the analysis of extreme values and qualitative interpretation of distributions, without providing direct access to detailed numerical processing of stress fields at the level of individual elements or integration points.

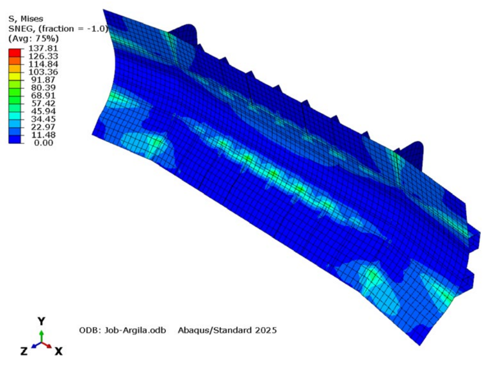
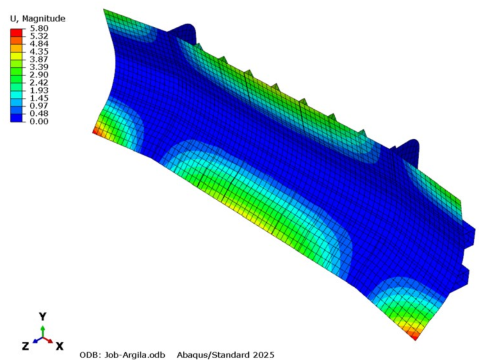
The distribution maps of equivalent Von Mises stresses and maximum displacements obtained through classical post-processing in Abaqus enable a global assessment of the structural behavior of the bulldozer blade for the three analyzed materials: clay, marl, and sandy soil (Figures 6–8).

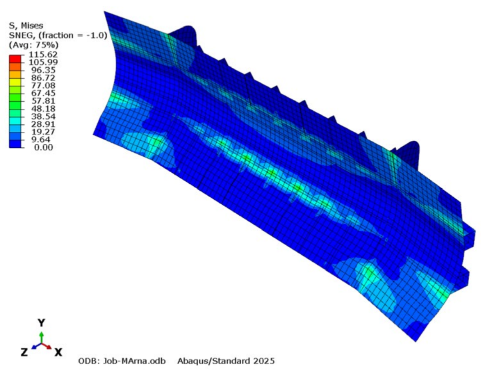
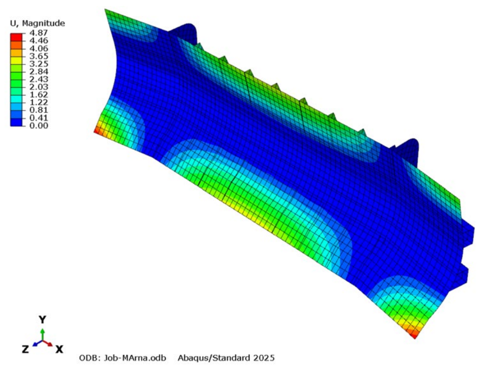
In all three loading cases, the Von Mises stress distribution reveals the occurrence of maximum values in critical structural regions of the blade, namely near the cutting edge, at the junctions between the main plate and the longitudinal stiffeners, and in areas of geometric transition. These stress concentrations are characteristic of welded structures subjected to bending and contact loads, resulting from local stiffness variations and the transfer of forces toward the stiffening elements.

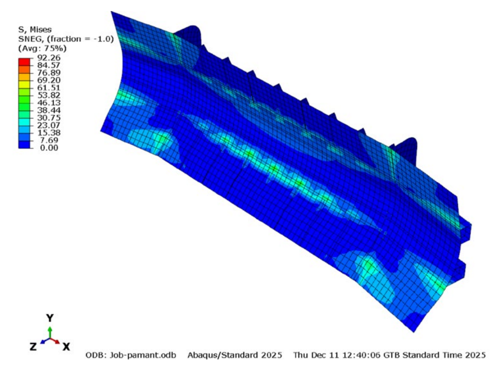
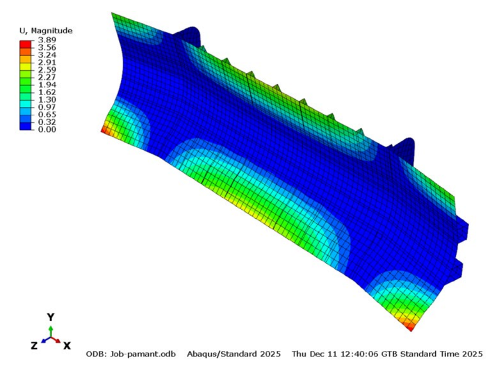
➥ For the clay loading case (Fig. 6), the maximum equivalent Von Mises stress reaches a value of $\sigma\_{VM,max} \approx 91.87$ N/mm², indicating a relatively high stress level generated by the pressure applied on the active surface of the blade. Even under these conditions, the obtained stress values remain below the yield limit of S355 structural steel, confirming that the blade operates entirely within the elastic regime.

➥ In the case of marl loading (Fig. 7), the maximum equivalent stress decreases to $\sigma\_{VM,max} \approx 77.08$ N/mm², reflecting a lower stress level compared to the clay case, as a consequence of the reduced distributed pressure applied on the blade surface.

➥ For sandy soil loading (Fig. 8), the lowest maximum stress value is obtained, $\sigma\_{VM,max} \approx 61.51$ N/mm², corresponding to the least severe loading condition among the analyzed cases. This result clearly confirms the direct influence of the mechanical properties of the displaced material on the stress state developed within the blade structure.

<table>
<tr><th colspan="2">Soil type: Clay</th></tr>
<tr><td>Mises stress reaches,
$\sigma_{(VM,max)} = 91.87$ [N/mm^2]</td><td>Maximum displacement , mm
Umax = 3.87 [mm]</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">**Fig. 6**. Distribution of equivalent Von Mises stresses and displacement field for the bulldozer blade under clay loading conditions.</td></tr>
</table>

<table>
<tr><th colspan="2">Soil type: Marl</th></tr>
<tr><td>Mises stress reaches,
$\sigma_{(VM,max)} = 77.08$ [N/mm^2]</td><td>Maximum displacement , mm
Umax = 3.25 [mm]</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">**Fig. 7.** Distribution of equivalent Von Mises stresses and displacement field for the bulldozer blade under marl loading conditions</td></tr>
</table>

<table>
<tr><th colspan="2">Soil type: Sandy soil.</th></tr>
<tr><td>Mises stress reaches,
$\sigma_{(VM,max)} = 61.51$ [N/mm^2]</td><td>Maximum displacement, mm
Umax = 3.89 [mm]</td></tr>
<tr><td>
</td><td>
</td></tr>
<tr><td colspan="2">Fig. 8. Distribution of equivalent Von Mises stresses and displacement field for the bulldozer blade under sandy soil loading conditions</td></tr>
</table>

The displacement maps indicate a global elastic behavior of the blade structure, with maximum values localized in the frontal area, away from the attachment points. This deformation pattern is characteristic of cantilever-type structures, where the load is applied at the free edge and stiffness is ensured by internal structural reinforcements.

➥ For the clay loading case, the maximum displacement reaches U_max ≈ 3.87 mm, highlighting a local flexibility of the blade under high pressure, without leading to excessive deformations that could affect structural stability or functional performance.

➥ In the case of marl, the maximum displacement decreases to U_max ≈ 3.25 mm, confirming the direct correlation between the applied load level and the structural deformability.

➥ For sandy soil, the maximum displacement is U_max ≈ 3.89 mm, a value comparable to that obtained for clay. This result suggests that, in addition to the magnitude of the applied pressure, its distribution over the blade surface and the curved geometry of the structure significantly influence the displacement field.

4.1.2. Advanced analysis method through numerical post-processing using Python scripts

To overcome the limitations of classical FEM analysis, an advanced numerical post-processing methodology was developed in this study, implemented through dedicated Python scripts. The purpose of this approach is to transform the raw finite element results into an engineering evaluation tool applicable to design, operation, and maintenance.

By automatically exporting the stress tensor from the Abaqus environment and organizing it in tabular format, the proposed methodology enables detailed statistical and comparative analyses of Von Mises equivalent stresses as well as principal stresses. This stage significantly extends the interpretation capabilities beyond purely visual inspection.

The comparative analysis of the stress state for different soil types (clay, marl, and sandy soil) provides direct insight into the influence of operating conditions on the structural loading of the blade. By comparing stress distributions and characteristic values, conclusions can be drawn regarding optimal operating regimes, namely conditions under which the blade is uniformly and efficiently loaded. In this context, the resulting comparative plots can serve as technical support for the development of an operating manual, indicating soil types and working conditions that lead to reduced stresses and safer equipment operation.

The determination of principal stresses (σ_1, σ_2, σ_3), maximum shear stress τ_max, and their representation using Mohr's circles allows an in-depth evaluation of stress mechanisms and material strength reserves. Correlating these results with the yield limit of the S355 steel used for the blade provides an objective demonstration of the load-bearing capacity of the structure, constituting a solid argument for structural conformity verification and structural strength certification of the design.

Furthermore, the analysis of cumulative distributions, stress paths, and parameters derived from Mohr's circles enables the identification of zones and regimes where stresses are cyclic or approach critical values. These findings are particularly relevant for long-term operation, as they indicate areas susceptible to wear, fatigue, or crack initiation. Consequently, the obtained plots can be used as a basis for developing a maintenance guideline, indicating when and where blade inspections are required, with emphasis on the most mechanically stressed zones [7–8].

4.2. Graphs for comparative analysis and engineering interpretation of results

4.2.1 Hazard map – identification of critical blade zones

To identify areas with high damage potential and to assess the severity of local stresses, a structural hazard map was defined based on the analysis of maximum values and upper percentiles of the Von Mises equivalent stress. This representation highlights the differences between local extreme stresses and the overall stress level of the structure, providing a clear perspective on stress concentrations that may lead to crack initiation or premature structural degradation.

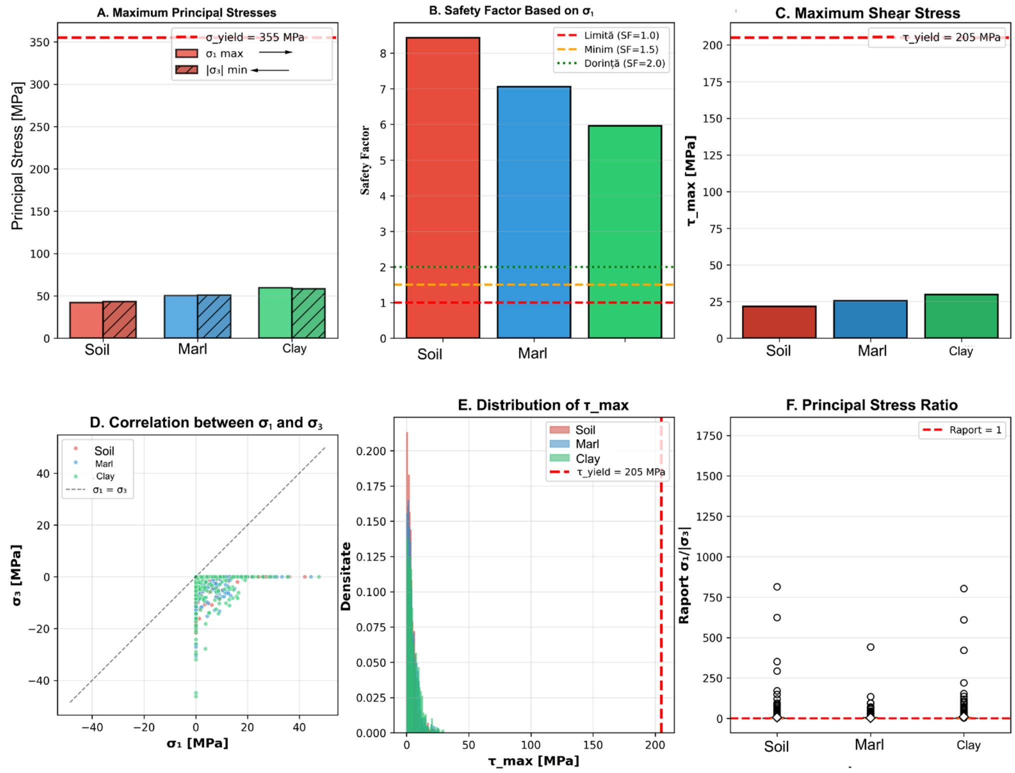


Fig. 9. Structural hazard map of the bulldozer blade based on advanced stress state analysis

By correlating the principal stresses, maximum shear stresses, and safety factors, the critical stress zones were clearly identified, thus defining a structural hazard map. The statistical representations and the σ_1–σ_3 correlations highlight the multiaxial nature of the stress state and the significant influence of soil type on the stress level. The distributions of τ_max and the ratios between principal stresses provide essential information regarding the risk of wear, crack initiation, and local failure. Overall, the adopted methodology supports both the validation of the blade's structural strength and the development of concrete recommendations for operation and maintenance, contributing to increased safety and durability of the equipment.

4.2.2 Structural strength certificate – Verification of the load-bearing capacity of the design

To validate the proposed structural solution, the maximum values of the Von Mises equivalent stresses were compared with the yield strength of the material used for the blade (S355 structural steel). This comparison allows the evaluation of the material utilization level and the verification of compliance with the safety requirements imposed during operation.

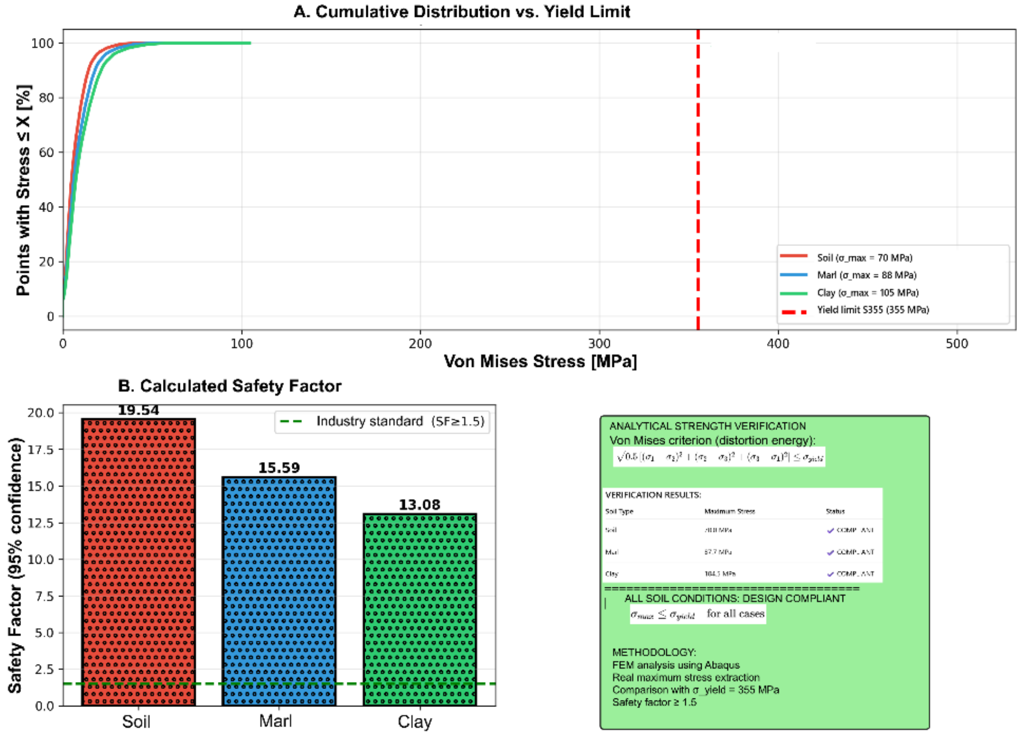


Fig. 10. Structural strength certificate of the bulldozer blade – analytical verification based on the Von Mises criterion and safety factors

The utilization ratio values remain below the critical threshold equal to unity for all analyzed cases, which confirms that the blade structure operates within the elastic domain and satisfies the imposed strength criteria. This type of graphical representation can be regarded as a numerical strength certificate, demonstrating that the design solution is mechanically adequate and provides sufficient safety margins for normal operating conditions.

4.2.3 Triaxial analysis of the stress state using Mohr's circles

To evaluate the stress state under complex loading conditions, a triaxial stress analysis was performed using Mohr's circles. This approach allows a comprehensive representation of the normal and shear stress components acting on material planes, providing direct insight into the stress distribution and shear resistance mechanisms. The analysis was carried out for three representative ground conditions—soil, marl, and clay—based on principal stresses extracted from finite element simulations. In addition, the stress states were projected into the p–q stress space to enable a comparative assessment of deviatoric and mean stress components.

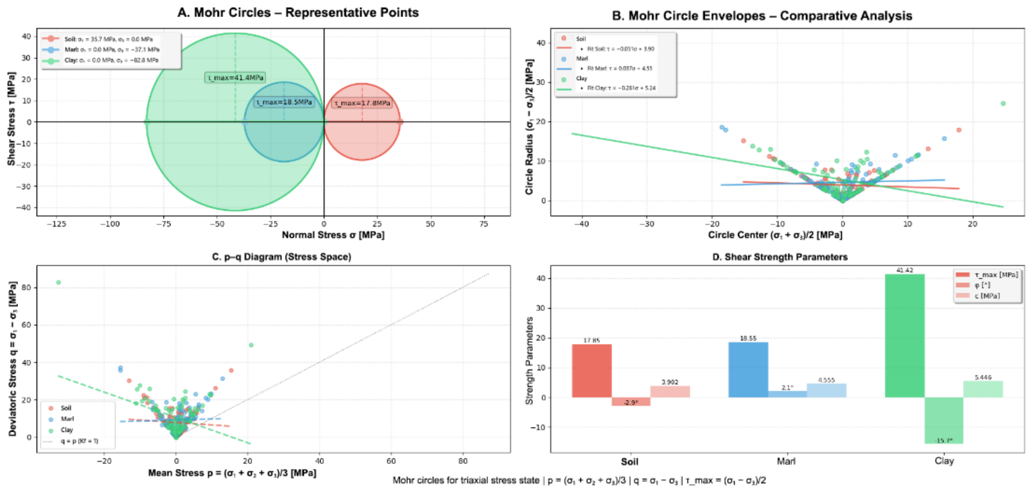


Fig. 11. Mohr's circle and p–q diagram for triaxial stress analysis of soil, marl, and clay

The Mohr circle representation highlights significant differences in the stress response of the analyzed materials. Clay exhibits the largest Mohr circle radius, indicating the highest maximum shear stress, which reflects a pronounced sensitivity to deviatoric loading. Marl and soil present smaller circle radii, corresponding to lower shear stress levels and a more moderate stress redistribution. The fitted envelopes show distinct slopes and intercepts, revealing variations in shear strength parameters among the materials. The p–q diagram confirms these observations, with clay displaying higher deviatoric stress concentrations for comparable mean stress values. Overall, the triaxial analysis demonstrates that material heterogeneity strongly influences the stress path and shear resistance, emphasizing the necessity of material-specific characterization in structural and geotechnical assessments.

5. CONCLUSIONS

In this paper, an advanced structural analysis of a bulldozer blade was carried out, based on the finite element method and complemented by an extended numerical post-processing stage using Python scripts. This approach enabled not only the classical evaluation of stresses and displacements, but also an in-depth interpretation of the mechanical stress state, directly correlated with real operating conditions.

The FEM analysis revealed that the distribution of stresses and deformations is strongly influenced by the type of displaced soil, with the highest stress levels recorded for clay, followed by marl and sandy soil. The identified critical zones are predominantly located in the cutting edge region and at structural joints, where stress concentrations arise due to geometric discontinuities and contact conditions.

By employing advanced numerical post-processing, detailed comparative analyses were performed, including statistical distributions, safety factors, principal stresses, and triaxial analyses based on Mohr's circles. The results confirm that, for all analyzed cases, the maximum stress values remain well below the yield limit of the S355 material, ensuring comfortable safety factors and validating the structural adequacy of the design.

The integration of numerical results into dedicated graphical representations enabled the transformation of the engineering analysis into practical tools, such as the structural hazard map, operating manual, strength certificate, and preventive maintenance guide. Consequently, the study demonstrates the direct applicability of numerical analysis not only in the design phase, but also in operation and maintenance.

In conclusion, the proposed methodology provides a comprehensive and reproducible framework for evaluating the mechanical behavior of active components of earthmoving equipment, contributing to increased operational safety, optimization of operating parameters, and extension of the service life of the analyzed structure.

REFERENCES

[1]. Bathe, K. J., Finite Element Procedures, Prentice Hall, New Jersey, SUA, 1996.

[2]. Cook, R. D., Malkus, D. S., Plesha, M. E., Witt, R. J., Concepts and Applications of Finite Element Analysis, 4th Edition, John Wiley & Sons, New York, SUA, 2002.

[3]. Dassault Systèmes, Abaqus 2025 Documentation – Theory Guide, Dassault Systèmes Simulia Corp., Providence, Rhode Island, SUA, 2025.

[4]. Dassault Systèmes, Abaqus 2025 Documentation – Scripting User's Guide, Dassault Systèmes Simulia Corp., Providence, Rhode Island, SUA, 2025.

[5]. Gautam Puri,Python Scripts for ABAQUS, ISBN: 978-0-615-52050-6 , 2011

[6]. McKyes, E., Soil Cutting and Tillage, Elsevier, Amsterdam, Olanda, 1985.

[7]. Boresi, A. P., Schmidt, R. J., Sidebottom, O. M. Advanced Mechanics of Materials, 6th Edition, Wiley, 2003.

[8]. Chen, W. F., Han, D. J. Plasticity for Structural Engineers, Springer, 1988.