Aintelia Science Notes

ASNJ • BAUDER PRESS

Research Article

Distinguishing Physical Stress Concentration from Numerical Singularity in Large-Scale Structural FEM Analyses

Samet Dönerkaya[1,*], Samet Büken[2]

1, 2 BVS R&D Center, Ankara, Türkiye

Correspondence to:
Samet Dönerkaya – s.donerkaya@bvs.com.tr

How to Cite:
Actual Citation as taken from website

RECEIVED: 17/03/2026
ACCEPTED: 01/04/2026
PUBLISHED: 30/06/2026

ABSTRACT

In large-scale structural finite element analyses (FEA), extreme stress peaks frequently occur at geometric discontinuities, boundary conditions, and perfectly bonded interfaces. These values are often directly compared with the material yield strength, which may lead to overly conservative or misleading safety assessments. However, such stress peaks may originate from mathematical singularities inherent to linear elastic formulations rather than physically admissible stress states. This study investigates the distinction between physical stress concentration and numerical singularity in an industrial-scale structural steel model subjected to end loading. A three-dimensional finite element model containing approximately one million elements was analyzed under service loading conditions. The simulation results indicated that the maximum von Mises stresses reached nearly three times the yield strength of S355JR steel. Through theoretical evaluation of asymptotic stress behavior, plastic deformation theory, mesh characteristics, and contact modeling effects, it is demonstrated that these stress peaks represent numerical artifacts rather than actual structural failure conditions. A systematic engineering interpretation framework is proposed to prevent the misclassification of singular stresses in industrial FEM applications.

KEYWORDS

Finite Element Analysis, Numerical Singularity, Stress Concentration, Structural Analysis, Welded Steel Structures

1. Introduction

Finite element analysis (FEA) has become the dominant validation tool in modern structural engineering applications. In large-scale industrial systems such as crane structures, welded steel assemblies, heavy machinery components, and load-bearing frames, numerical simulations frequently involve millions of degrees of freedom and are expected to provide direct quantitative indicators of structural safety. In common industrial practice, the maximum equivalent (von Mises) stress is directly compared with the material yield strength, and structural adequacy is evaluated according to this comparison. Although this approach is simple and practical, it implicitly assumes that the calculated stress field is physically meaningful throughout the entire computational domain.

However, this assumption is not valid in all situations. Linear elastic finite element formulations naturally admit stress fields that exhibit asymptotic behaviour near geometric discontinuities, boundary conditions, bonded interfaces, and idealized contact transitions. In such regions, stresses may theoretically increase without bound as the distance to the discontinuity approaches zero. Nevertheless, real materials cannot sustain infinite stresses, and therefore such behaviour cannot correspond to a physically admissible stress state.

The fundamental distinction between physical stress concentration and mathematical stress singularity is often not sufficiently addressed in routine industrial FEM applications. Physical stress concentrations arise from the redistribution of loads around geometric features such as holes, chamfers, notches, or weld toes. These stress increases are finite and exhibit convergent behaviour under mesh refinement. In contrast, singular stresses are theoretically unbounded and do not converge as the element size decreases. These stresses are numerical artefacts resulting from idealized boundary conditions, perfectly sharp geometries, and linear material modelling assumptions.

Misinterpreting singular stresses as actual structural overstress in large-scale industrial models may lead to serious engineering consequences. Such misinterpretations may result in unnecessary structural reinforcements, over-conservative design decisions, increased production costs, or incorrect failure predictions. Conversely, ignoring stress peaks without proper classification may lead to unsafe conclusions. Therefore, it is essential to establish a systematic framework capable of distinguishing physically meaningful stress concentrations from numerical singularities.

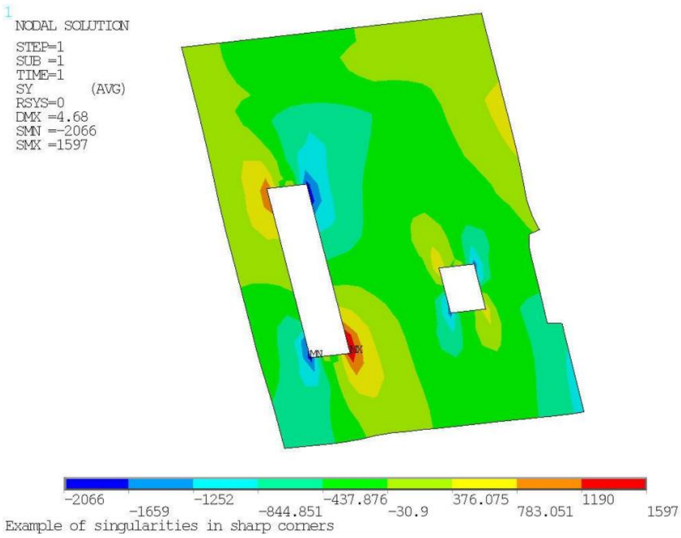

Figure 1: *Typical sources of stress singularities in finite element simulations, including sharp re-entrant corners and point loads.*

As illustrated in Figure 1, stress singularities may arise from several modelling idealizations frequently encountered in finite element simulations. Sharp geometric discontinuities, point load applications, and abrupt stiffness transitions introduced by boundary conditions may generate highly localized stress amplification. These modelling situations are particularly common in large industrial assemblies where complex connection regions and simplified boundary conditions are unavoidable. Therefore, distinguishing between physical stress concentration and mathematical stress singularity becomes a critical task in engineering interpretation.

Singularity problem becomes particularly critical in welded and contact-intensive steel assemblies. Perfectly bonded interfaces and idealized fixed supports introduce abrupt stiffness transitions that significantly amplify stress gradients. When combined with linear elastic material modelling, these conditions may produce calculated stress values that exceed the material yield strength even under service-level loading. Such results often raise serious concerns during industrial validation procedures; however, their physical relevance is not always critically examined.

Although stress singularities are well established in fracture mechanics and asymptotic elasticity theory, their interpretation within the context of large-scale industrial FEM analyses remains fragmented in the literature. Classical singularity theory is typically discussed in relation to crack tips and concave corners. In contrast, industrial analysts frequently encounter singular stress growth in crack-free structures, particularly near support edges and bonded contact interfaces. Bridging this gap between theoretical singularity concepts and practical large-scale simulation practice constitutes the primary motivation of this study.

In this work, a large-scale structural steel assembly subjected to end loading under linear elastic modelling is investigated. The numerical results show that the calculated von Mises stresses reach approximately three times the yield strength of S355JR steel. Instead of directly interpreting these stress peaks as indicators of structural failure, their numerical origin is analysed in terms of asymptotic stress behaviour, mesh dependency, material modelling limitations, and boundary condition idealizations. The objective of this study is not only to demonstrate the existence of singular stresses but also to develop a systematic engineering framework capable of distinguishing between finite physical stress concentration, mesh-dependent numerical stress growth, and actual structural overstress conditions. By clarifying this distinction, the reliability of structural interpretation in large-scale industrial FEM applications can be improved, thereby preventing both overly conservative and insufficient design decisions.

2. Literature Review

Stress singularities have been rigorously established within the framework of linear elasticity theory, particularly in the context of fracture mechanics and interface problems. The classical formulation developed by Rice (1968) demonstrated that the fields near crack tips in notched and cracked bodies can be characterized by path-independent integrals and asymptotically singular strain fields, thereby providing a physically meaningful measure of local stress concentration [1]. Subsequent asymptotic analyses expressed stress and displacement fields in terms of power-series expansions of the form r^λ, showing that the eigenvalues governing the order of singularity depend only on local geometry, boundary conditions, and material properties. In composite laminates and bimaterial interfaces, the order of the stress singularity is determined by the real part of the eigenvalue, and the interval $0 < \mathrm{Re}(\lambda) < 1$ represents physically admissible singular fields [2]. However, it must be emphasized that such singular solutions arise from the idealized assumptions of continuum elasticity. Real materials cannot sustain infinite stresses; therefore, singular behaviour must be interpreted as a mathematical consequence of the local boundary value problem. Nevertheless, asymptotically singular fields remain of central importance in fracture mechanics because they govern the fundamental quantities controlling crack initiation and propagation criteria. In this context, the characterization of singularity is physically meaningful and directly related to stress intensity factors.

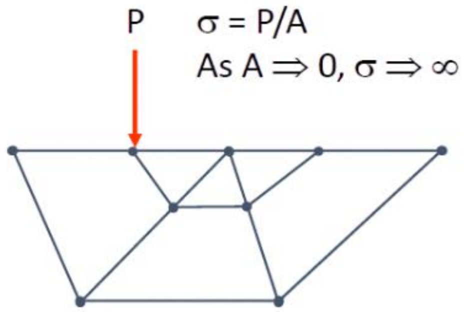

Figure 2: *Illustration of a stress singularity where the stress theoretically approaches infinity as the load application area tends toward zero.*

The mathematical origin of stress singularities can be understood through the relation between applied load and the effective load transfer area. As shown in Figure 2, when a force is applied over an infinitesimally small area, the theoretical stress tends to increase without bound as the area approaches zero. This behaviour represents a mathematical property of continuum mechanics formulations rather than a physically attainable stress state in real materials. Although singular solutions are analytically well understood, their numerical representation within classical finite element formulations involves significant challenges. It has long been recognized that quasi-uniform finite element meshes cannot accurately reproduce the stress field in the vicinity of a singular point [3]. To overcome this limitation, two principal approaches have been developed: local mesh refinement and basis function enrichment. Classical singular elements, such as quarter-point elements, enable accurate representation of stress fields of the form r^λ by embedding known asymptotic forms into the element interpolation. More recent developments combine the finite element method with the scaled boundary finite element method (SBFEM) through semi-analytical enrichment techniques, allowing the automatic treatment of singular behaviour without prior specification of asymptotic exponents [3–6]. These developments show that singularity handling is an active research area within computational mechanics. However, the majority of these studies focus on physically meaningful singular fields such as those found at crack tips. Comparatively few investigations have addressed singular stresses that arise unintentionally in large-scale industrial models because of geometric idealizations, discontinuous boundary conditions, or bonded interfaces.

In practical engineering applications, particularly in welded and large-scale steel assemblies, the interpretation of stress peaks is directly related to fatigue and strength assessment methodologies. Traditional finite element analyses generally use the maximum equivalent (von Mises) stress as a direct indicator of structural safety. However, such peak stresses are frequently mesh-dependent and may not represent a physically meaningful structural demand. Dong (2001) proposed a structural stress definition that separates stresses into membrane and bending components and remains insensitive to mesh size in the fatigue assessment of welded joints [7]. This approach clearly reveals the critical distinction between local stress concentrations and structural stress measures associated with damage mechanisms. In this context, it becomes evident that raw peak stresses extracted directly from finite element nodes near geometric discontinuities may be misleading. Similarly, stress singularities associated with traction discontinuities or line-supported stress distributions have been investigated using Airy stress function formulations [8]. These studies emphasize that stress intensifications observed along an interface or edge may be the consequence of idealized boundary conditions rather than actual structural overstress. More recent numerical studies involving singular source terms in elastic wave equations have likewise shown that the existence of mathematical singularities in the governing equations does not necessarily imply instability or physical collapse, provided that the global solution remains well defined and energetically bounded [9]. Classical fracture mechanics primarily focuses on square-root singularities of the form r^(−1/2). However, not all singular behaviour in elasticity takes the form of a pure power law. Under certain interface and boundary formulations, logarithmic stress singularities may arise. Sinclair and Epps (2002) showed that the use of displacement shape functions in submodelling boundary conditions can generate artificial logarithmic stress singularities because of derivative discontinuities in the prescribed displacement field [10]. Their analysis demonstrates that piecewise polynomial interpolation may produce spurious stress growth near boundary nodes even in otherwise regular elastic problems. This finding is particularly important for large-scale industrial finite element workflows in which global–local modelling or submodelling strategies are commonly employed. More recently, enriched crack-tip elements have been developed to explicitly represent logarithmic singular behaviour in interface fracture problems. Herrera-Garrido et al. (2026) proposed a special enriched element capable of reproducing logarithmic stress terms associated with spring-type interface models [11]. Their study confirms that logarithmic singularities can severely disrupt convergence behaviour in classical finite element formulations and that solution accuracy is significantly reduced if enrichment is not introduced. These studies clearly demonstrate that not all stress intensifications conform to classical crack-tip asymptotics. In industrial assemblies containing bonded interfaces, high-stiffness supports, or Robin-type boundary conditions, logarithmic-type stress growth may arise numerically even in the absence of a physical fracture process. Stress singularities are therefore not limited to cracks and V-notches.

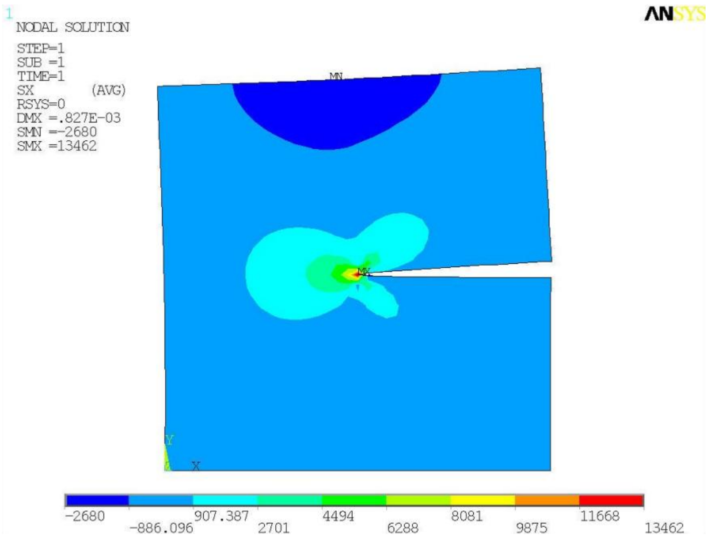

Figure 3: *Example of a classical stress singularity occurring at a crack tip predicted by linear elastic fracture mechanics.*

Contact mechanics problems also frequently exhibit singular stress behaviour, especially in stick–slip transition regions and at contact edges. Yue and Abdel Wahab (2014) investigated stress singularities arising in fretting contact under partial slip and gross sliding conditions [12]. Their finite element-based studies showed that singular stress signatures depend strongly on the fretting regime and the coefficient of friction. In particular, singular behaviour appears in high-friction partial-slip regimes, whereas it disappears under gross sliding conditions. This observation is of critical importance for industrial finite element applications. In large-scale steel assemblies, stress growth near bonded or frictional interfaces may represent regime-dependent singular fields rather than actual structural overstress. The eigenfunction expansion method has been widely used to determine the order of singular stresses in V-notched structures. Ping et al. (2008) developed a super-singular wedge-tip element containing numerical eigensolutions for anisotropic multi-material wedge tips [13]. This approach enabled rapid convergence in reproducing singular stress fields based on eigenvalue–eigenvector pairs. Similarly, Pageau et al. (1995) formulated a finite element approach for anisotropic materials exhibiting singular in-plane stress fields [14]. Their work extended eigenvalue analysis techniques to complex multimaterial junction regions and confirmed the dependence of singularity order on geometry and material properties. More recently, Pan et al. (2023) emphasized that the full stress field near a V-notch consists of both singular and higher-order non-singular terms [15]. These higher-order non-singular contributions were shown to have a significant influence on fracture prediction and stress intensity evaluation. The distinction between singular and non-singular contributions is therefore fundamental for interpreting peak stresses observed in numerical models. Collectively, these studies show that the order of singularity depends on local geometry and material mismatch, that higher-order non-singular terms influence practical stress assessment, and that accurate numerical extraction requires asymptotic consistency. However, these investigations mainly focus on the accurate solution of physically meaningful singular fields and do not provide a general framework for distinguishing numerical artefacts from physical stress concentrations in large-scale industrial assemblies.

A classical example of such behaviour is observed in the vicinity of crack tips, where the stress field follows an asymptotic power-law distribution. As illustrated in Figure 3, the stress theoretically increases as the distance to the crack tip decreases. While this formulation provides a physically meaningful framework in fracture mechanics, similar singular stress growth may also appear unintentionally in numerical simulations of crack-free structures due to modelling idealizations. Fatigue assessment methods for welded and large steel structures have sought to reduce the influence of mesh-dependent peak stresses. The structural hot-spot stress method (Poutiainen et al., 2004) provides extrapolation-based procedures for obtaining mesh-independent structural stress parameters [16]. Likewise, the Peak Stress Method (Visentini et al., 2022) accounts for singular peak behaviour while estimating notch stress intensity factors from coarse meshes [17]. These approaches acknowledge a critical fact: raw peak stresses obtained directly from finite element nodes near sharp weld toes or rigid supports are not always physically meaningful. Instead, fatigue-relevant structural stresses are obtained through extrapolation, energy-based averaging, or decomposition into membrane and bending components. Thus, the literature implicitly recognizes the risk of misinterpreting singular stress peaks; however, it still lacks a general framework capable of systematically distinguishing asymptotic physical stress concentration, numerical singular growth, and modelling-induced artificial stress amplification in large-scale linear elastic industrial models. Singularity theory has also been explored outside structural mechanics. Lei and Sornette (2025) proposed log-periodic power-law finite-time singularities in the prediction of rockbursts [18]. Although developed in a geomechanical context, this study reinforces an important conceptual point: mathematical singular behaviour does not automatically imply physical collapse; its interpretation must be made within the framework of energy admissibility and constitutive constraints. In classical elasticity, stress singularities have been systematically characterized through asymptotic eigenfunction analysis. Williams (1952) showed that stress behaviour at angular corners depends critically on the combination of boundary conditions and that certain corner angles subjected to mixed boundary conditions, such as clamped–free edges, may produce unbounded stresses [19]. This fundamental study established the eigenvalue-based characterization of singularity strength in angular regions. Building on Williams' formulation, Sinclair's comprehensive studies clarified the necessity of recognizing singular behaviour as well as strategies for its removal, interpretation, and asymptotic identification [20,21]. These works classified singularities into power-law, logarithmic, log-squared, and oscillatory forms, and emphasized that singular stresses are not physically realizable stress states but mathematical consequences of boundary value problems. Sinclair also noted that the onset of plasticity does not eliminate the singularity in the mathematical sense, because an elastic response must first develop prior to yielding [21]. This distinction is particularly important for large-scale industrial finite element models in which linear elastic formulations are widely used even when local peak stresses exceed the yield strength. Subsequent analytical studies extended singularity characterization to various notch geometries and loading modes. Semi-analytical boundary element approaches developed for V-notches demonstrated that stress intensity factors and higher-order asymptotic coefficients can be extracted directly from expansion amplitudes [22]. These approaches confirm that the order of singularity is governed by eigenvalue solutions dependent on notch opening angle and material properties. Finite element analyses of more general configurations have shown that singular behaviour also persists under partial slip and high-friction contact conditions [12]. The strong dependence of singular stress signatures on stick–slip transitions in fretting regimes indicates that contact constraints may give rise to singular-like numerical growth even in the absence of a crack tip. Error-assessment strategies for singular elements have also been developed. Meshii and Watanabe proposed a stress intensity factor error index based on known analytical displacement fields [23]. Their findings showed that singular field extraction is highly sensitive to mesh size and loading configuration, and that convergence analysis is essential for the interpretation of peak stresses. In practical fatigue assessment, several local stress and energy concepts have been developed to reduce mesh dependency and singular amplification. The local strain energy density (SED) approach, comprehensively reviewed by Radaj [24], is based on the average energy density within a finite control volume around the notch tip rather than on pointwise peak stress evaluation. This approach explicitly acknowledges that even if stresses are asymptotically singular, the averaged energy density may remain finite and physically meaningful. Similarly, the Peak Stress Method (PSM) provides a practical engineering solution by relating coarse-mesh peak stresses in welded joints to notch stress intensity factors [25]. This method accepts the existence of singular peak behaviour but transforms it into fatigue-relevant equivalent stress measures. In this way, an important conceptual shift occurs: the focus moves from interpreting absolute peak stress values to extracting physically interpretable parameters such as SIF, NSIF, SED, and structural stress.

In the present study, the persistent industrial gap between computed peak stresses and physically admissible stress states is addressed, and a structured interpretation workflow is developed and demonstrated for singular stress growth arising in large-scale structural finite element models. This workflow is based on the comparative evaluation of an industrial-scale structural assembly made of S355JR steel, analysed under service loading with gravity and consisting of approximately 2,027,936 nodes and 990,617 elements, and two representative simulations in which the model predicts highly localized von Mises peaks (680.952 MPa and 825.487 MPa), while the global displacements remain within the service range (for example, 6.60265 mm under end loading). Building upon established singularity-management approaches, the study explicitly tests and documents the influence of modelling idealizations, including point loads, fixed supports, and bonded/contact transitions, and applies corrective strategies such as replacing point forces with equivalent pressure loads, adding radii to re-entrant corners, and substituting fixed supports with contact-based boundary representations. To ensure that the observed stress growth is not incorrectly attributed to discretization artefacts, the work also includes mesh-quality assessments using standard metrics such as aspect ratio, skewness, Jacobian ratio, and related mesh diagnostic measures. Finally, in order to relate numerical peaks to physical admissibility, the interpretation is framed in terms of the fundamental principles of rate-independent plasticity, including the yield criterion, flow rule, and hardening rule, which are regarded as the natural regularization mechanisms for unbounded elastic stresses. Taken together, these steps provide a repeatable engineering methodology for distinguishing among (a) finite physical stress concentration, (b) mesh-dependent numerical singularity, and (c) actual structural overstress in large-scale industrial finite element applications.

3. Methodology

3.1. Numerical Model Description

The structural model analyzed in this study corresponds to an industrially designed crane girder assembly. The geometry was obtained in CAD format and subsequently prepared for finite element analysis in the ANSYS Mechanical 2025 R1 environment. The model represents the typical geometric and connection characteristics of large-scale steel structural systems. Prior to the analysis, a geometry preparation stage was conducted in which the solid model was converted into a surface model in order to reduce solution time and improve mesh quality.

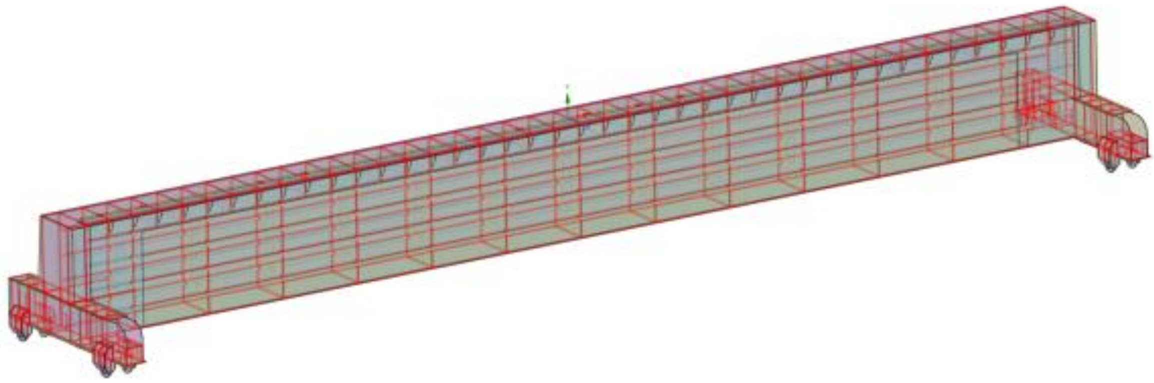

Figure 4: *Industrial crane girder assembly representing a typical welded structural system used in large-scale FEM simulations.*

The industrial crane girder assembly illustrated in Figure 4 was selected as the benchmark model for the present investigation. This structure contains several geometric discontinuities, connection interfaces, and load transfer regions typical of welded steel assemblies. Such characteristics make it a representative industrial case for examining the occurrence of stress singularities in large-scale finite element analyses. The selected crane girder assembly represents a realistic industrial-scale welded steel structure containing multiple geometric discontinuities, support interfaces and load transfer regions. Such configurations are known to produce strong stress gradients in finite element simulations and therefore constitute an appropriate benchmark for investigating numerical stress singularities. This transformation was performed in the ANSYS SpaceClaim environment using the midsurface command. Through the midsurface approach, thin-walled structural components were reduced to shell representations, allowing the generation of a more regular mesh structure while significantly reducing computational cost. This method increases solution efficiency while accurately representing the global stiffness behavior, particularly in long and thin-walled steel girder systems. During the model preparation phase, small geometric features that do not significantly influence the structural behavior, such as small radius, chamfers, and secondary holes, were removed from the geometry. This simplification process reduces unnecessary local mesh refinement, minimizes element distortion, and improves numerical stability. This step is particularly important for preventing artificial stress concentrations that may arise from sharp corners or small geometric details. In welded joint regions, a shared topology approach was applied in order to reduce the number of contact definitions and optimize computational performance. With this method, instead of defining separate contact interactions on adjacent surfaces, shared nodes are created between connected parts, thereby minimizing the need for bonded contact definitions. The use of shared topology improves the conditioning of the global stiffness matrix and significantly reduces solution time by eliminating unnecessary contact iterations. Across the model, eight fixed (clamped) boundary conditions were defined, and surface-based bonded contact interactions were used where necessary.

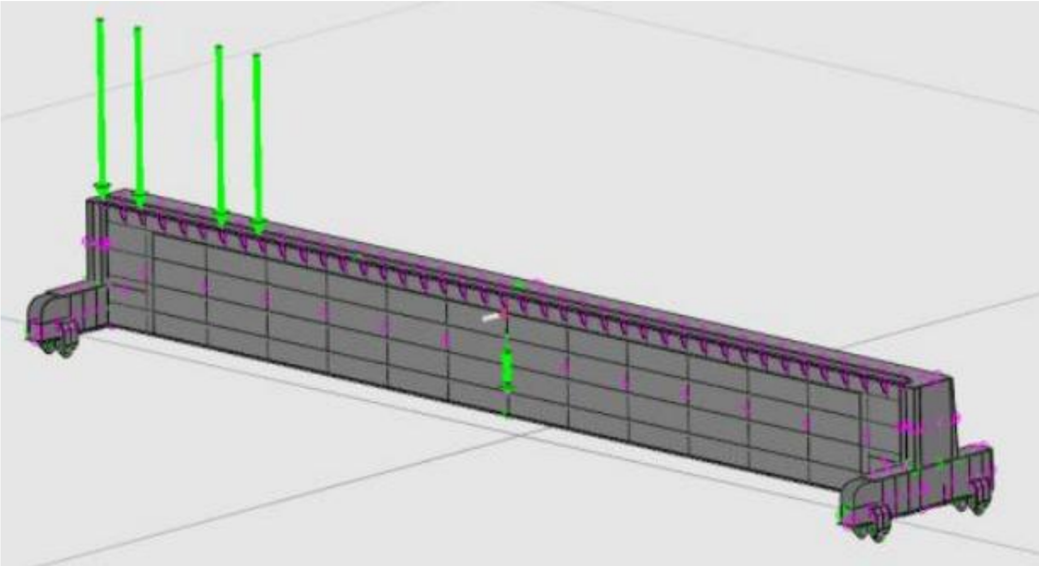

Figure 5: *Boundary conditions applied to the proposed CAD model.*

The boundary conditions applied to the model are shown in Figure 5. Although these constraints represent typical structural mounting conditions in industrial systems, they also introduce abrupt stiffness transitions within the numerical model. Under linear elastic assumptions, such transitions may significantly amplify local stress gradients, particularly near constrained edges and connection regions. These idealized boundary conditions were selected to represent the global load-bearing behavior of the structure. However, it is well known that fully clamped supports and bonded contact definitions may introduce abrupt stiffness transitions, which can lead to localized stress amplification under linear elastic analysis. Therefore, the modeling assumptions were selected deliberately with consideration for both computational efficiency and the interpretability of the resulting stress distribution. Consequently, the developed finite element model represents an industrial-scale steel assembly with a large number of degrees of freedom while preserving the global stiffness behavior of the system. This modelling framework provides a suitable basis for systematically investigating whether the stress peaks observed under linear elastic analysis originate from physically meaningful stress concentrations or from numerical singularities.

3.2. Material Model

In the analysis model, S355JR structural steel defined in the ANSYS material library was assigned to all structural components. The analyses performed within the scope of this study were conducted within the framework of a static linear structural solution, and the material behaviour was initially modelled as isotropic, homogeneous, and linear elastic.

Table 1: *Material card properties of S355JR steel.*

Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m^-3		
4	Isotropic Secant Coefficient of Thermal Expansion				
5	Coefficient of Thermal Expansion	1.2E-05	C^-1		
6	Isotropic Elasticity				
7	Derive from	Young's M...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0.3			
10	Bulk Modulus	1.6667E+11	Pa		
11	Shear Modulus	7.6923E+10	Pa		
12	Strain-Life Parameters				
13	Display Curve Type	Strain-Life			
14	Strength Coefficient	9.2E+08	Pa		
15	Strength Exponent	-0.106			
16	Ductility Coefficient	0.213			
17	Ductility Exponent	-0.47			
18	Cyclic Strength Coefficient	1E+09	Pa		
19	Cyclic Strain Hardening Exponent	0.2			
20	S-N Curve	Tabular			
21	Interpolation	Log-Log			
22	Scale	1			
23	Offset	0	Pa		
24	Tensile Yield Strength	2.5E+08	Pa		
25	Compressive Yield Strength	2.5E+08	Pa		
26	Tensile Ultimate Strength	4.6E+08	Pa		
27	Compressive Ultimate Strength	0	Pa		

In the linear elastic formulation, the material behavior is governed by Hooke's law:

$$\sigma = E\varepsilon \quad (1)$$

where σ denotes the normal stress, E represents the Young's modulus of the material, and ε is the normal strain. Within this approach, the stress–strain relationship is linear, and the deformation remains fully recoverable as long as the stress level stays below the yield strength. Although the elastic solution is sufficient for examining the global stiffness behaviour of the structure, it does not physically limit local stress growths. S355JR steel is a ductile material, and once the yield strength is exceeded, plastic deformation begins. Plastic deformation occurs due to atomic slip along crystallographic planes caused by deviatoric stresses (dislocation motion), resulting in permanent strains. This behaviour is characterized by the inability of the material to return to its original configuration after the removal of the load.

3.3. Boundary Conditions and Loading

The boundary conditions and loading definitions applied in the numerical model were established in order to represent realistic industrial operating conditions of the crane girder assembly. In large-scale structural finite element analyses, the definition of appropriate boundary conditions plays a critical role in ensuring that the numerical model accurately reflects the physical load transfer mechanisms of the system. For this reason, boundary conditions and loading scenarios were carefully selected to represent the structural constraints and operational loading environment of the assembly. All analyses were performed within the framework of a static linear structural solution in ANSYS Mechanical. In this formulation, loads were applied in a single load step under the assumption of small deformations and linear material behaviour. This approach allows the global structural response to be evaluated efficiently while maintaining numerical stability and reducing computational cost. The loading configuration was defined in a manner consistent with typical service-level loading conditions encountered in industrial crane structures.

3.3.1. Support Conditions

In order to represent the mechanical connection between the crane girder assembly and the supporting chassis structure, fixed support (clamp) boundary conditions were assigned to eight hole regions located on the mounting interfaces of the model.

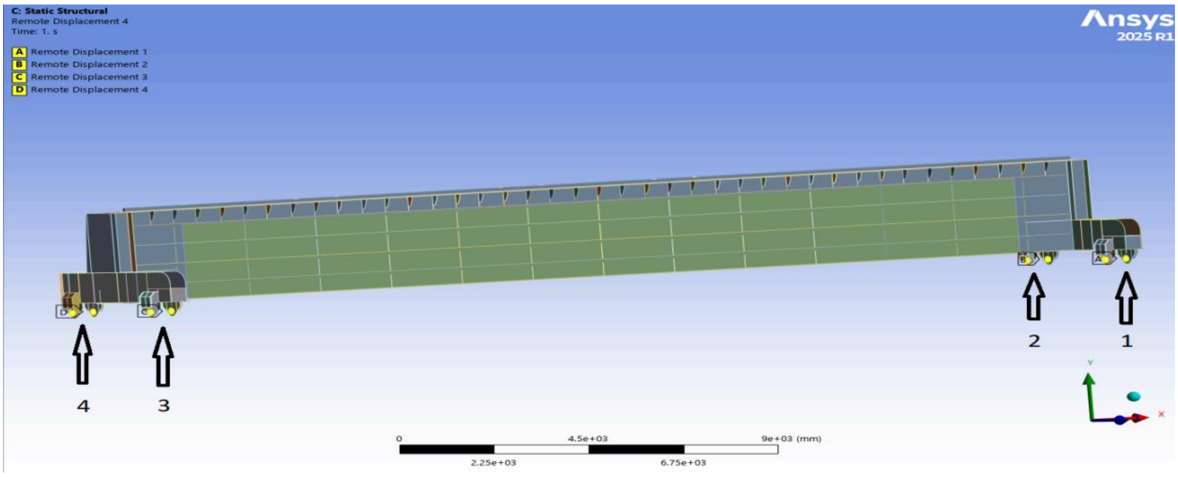

Figure 7: *Fixed support boundary conditions assigned to the hole regions of the CAD model.*

The fixed support locations illustrated in Figure 6 correspond to the mounting holes through which the structural assembly is attached to the supporting frame. While such boundary conditions are commonly used in structural simulations, they represent an idealization of the actual support stiffness and may therefore generate localized stress amplification in their vicinity. These hole regions correspond to the physical mounting points through which the structural assembly is connected to the supporting frame in the actual industrial system. By applying fixed support conditions at these locations, the structural constraints introduced by bolted or rigid mounting connections are represented in an idealized manner.

The fixed support definition constrains all translational displacement components at the associated nodes, such that

$$UX = UY = UZ = 0 \quad (2)$$

where UX, UY and UZ denote the translational displacement components in the global x, y and z directions, respectively. In other words, the nodes located on the support regions are prevented from undergoing any translational motion, effectively representing a rigid connection between the structure and its supporting base. This type of idealization is widely used in structural finite element modelling when the supporting structure is assumed to possess significantly higher stiffness than the analyzed component. Under such assumptions, the support structure can be considered rigid relative to the deformable structural assembly. However, it is also well established in the literature and in ANSYS technical documentation that fully fixed boundary conditions may introduce abrupt stiffness transitions within the numerical model. These transitions can result in steep stress gradients and local stress amplification, particularly in regions where constrained nodes interact with unconstrained structural domains. Such effects are especially pronounced in linear elastic analyses, where the absence of plastic deformation mechanisms prevents the redistribution of localized stresses. Consequently, high stress gradients and localized stress peaks are often observed near fixed support regions, particularly at geometric discontinuities or near support edges. For this reason, the interpretation of stress results in the vicinity of fixed supports must be performed with caution. In the context of the present study, these modelling assumptions were intentionally retained in order to allow the investigation of stress amplification behaviour and to distinguish between physically meaningful stress concentrations and numerical singularities arising from boundary condition idealizations.

3.3.2. Contact Definitions

In the numerical model, surface-based contact definitions were used for all interacting surfaces, and the contact type was assigned as bonded. Bonded contact prevents any relative displacement between contacting surfaces and enforces full displacement continuity at the corresponding nodes. This approach is commonly used for representing welded or rigid connections and provides a computationally efficient modelling strategy. However, bonded contact definitions may introduce abrupt deformation transitions, particularly between components with significantly different stiffness values. Under linear elastic analysis, such transitions may lead to localized stress amplification in the vicinity of the contact interfaces. In addition, through the application of the shared topology approach, common nodes were generated across many contact regions, thereby minimizing the number of explicit contact definitions required in the model. This modelling strategy reduces the number of contact solution iterations and improves computational efficiency. At the same time, it results in a mathematically fully connected structural system within the finite element model.

3.3.3. Loading Scenarios

In addition to the boundary conditions and contact idealizations described above, load definitions also represent important modelling parameters that may lead to stress singularities in linear elastic finite element models. In particular, point force applications or load definitions applied over vanishingly small areas may theoretically produce stress singularities under the continuum mechanics assumption. Two primary loading cases were considered in the numerical model. First, gravitational loading was applied to represent the self-weight of the structure. A gravitational acceleration of -9.81 m/s² was imposed, and the load was introduced into the model as a distributed body force acting throughout the entire structural volume. Secondly, a service load was applied in the form of a concentrated force in order to represent operational loading conditions. A point force with a magnitude of 1.93×10^6 N was applied in the Z direction on the designated surface of the girder. The force components were defined as

$$FX = 0, FY = 0, FZ = 1.93 \times 10^6 N \quad (3)$$

Point force applications are known to represent boundary conditions that may theoretically generate singular behaviour within the framework of continuum mechanics. When a force is applied over an infinitesimally small area, the stress value tends to increase according to the relation $\sigma = P/A$ as the effective area approaches zero. For this reason, the interpretation of stress peaks obtained near the load application region requires careful physical evaluation.

3.3.4. Linear Static Solution Assumptions

All simulations performed within the scope of this study were conducted under a linear static structural analysis framework. Within this formulation, the geometry was assumed to remain unchanged during deformation and the material behaviour was defined as linear elastic. Accordingly, geometric nonlinearities such as large deflection effects were neglected, plastic material behaviour and yielding were not activated in the material model, and contact nonlinearities were simplified through the use of bonded contact definitions. Consequently, the structural response of the system was evaluated entirely within the framework of linear elasticity theory. While this modelling approach provides an efficient method for evaluating the global stiffness behaviour and load transfer characteristics of the structure, it does not impose any physical limitation on the growth of local stresses. In linear elastic finite element formulations, stress values may increase significantly in regions where abrupt stiffness transitions, geometric discontinuities, or highly concentrated loads are present. Such modelling assumptions may therefore lead to the appearance of numerical stress singularities, particularly near constrained boundaries, sharp geometric transitions, or point load applications. In these regions, stresses may theoretically approach very large values due to the absence of plastic deformation or geometric redistribution mechanisms that would otherwise limit stress growth. For this reason, the stress peaks observed in the analysis must be interpreted carefully. Rather than representing a physically meaningful stress state, these peaks may reflect mathematical singular behaviour introduced by the modelling assumptions. Therefore, the interpretation of the results focuses primarily on the overall stress distribution patterns and the stress values in neighbouring elements, where the numerical solution is less affected by singularity behaviour.

3.4. Singularity Identification and Classification Framework

Distinguishing whether high stress peaks obtained in large-scale linear elastic finite element analyses correspond to physical stress concentrations or numerical singularities constitutes the primary objective of this study. In order to address this issue, a systematic identification and classification strategy was developed to bridge the gap between the asymptotic singularity definitions reported in the literature and the practical interpretation challenges encountered in industrial finite element modelling applications.

3.4.1. Definition of Singularity and Numerical Indicators

According to ANSYS technical documentation, a stress singularity is characterized by the absence of stress convergence under mesh refinement, where the calculated stress values continue to increase as the mesh becomes progressively finer. From a mathematical perspective, this behaviour can be described through an asymptotic stress field expressed as

$$\sigma(r) \sim r^{(-\lambda)} \quad (4)$$

where r represents the distance from the singularity point and $\lambda > 0$ denotes the singularity order. As the distance approaches zero ($r \to 0$), the stress theoretically tends toward infinity.

However, such behaviour cannot occur in real engineering materials. In physical structures, mechanisms such as plastic deformation, microstructural damage, or geometric rounding limit the magnitude of stresses and prevent the formation of infinite stress values. For this reason, the identification of singularities cannot rely solely on the maximum stress magnitude obtained from numerical analysis. Instead, the characteristic behaviour of the stress field must be carefully examined in order to determine whether the observed stress amplification represents a physically meaningful stress concentration or a numerical artefact associated with singularity behaviour.

3.4.2. Identification Criteria

In this study, four evaluation criteria were used to classify the observed stress peaks. First, a mesh dependency analysis was conducted. The behaviour of the maximum stress values was examined as the mesh density increased in the suspected singularity regions. If the maximum stress continues to increase with mesh refinement without convergence, it indicates a high probability of numerical singularity; whereas converging stress values suggest a physical stress concentration. The mesh refinement results in this study showed that the stress peaks increased as the element size decreased, indicating non-convergent behaviour.

Secondly, a location analysis was performed by comparing the spatial positions of the suspected singularity regions with typical singularity sources reported in the literature, such as point loads, sharp re-entrant corners, fully fixed boundaries, and bonded contact transitions. In the analysed model, the highest stresses were concentrated around the clamped hole regions, bonded contact interfaces, and load application surfaces, indicating that the stress peaks are largely related to modelling idealizations.

Thirdly, the results were evaluated in terms of global structural behaviour. In a real overstress condition, global stiffness reduction, excessive displacement, or local instability would be expected. However, the analysis results showed maximum displacements of approximately 6.60 mm and 16.24 mm, and the structure behaved consistently with an elastic beam response.

Finally, a plastic admissibility assessment was performed. In linear elastic analysis, stresses may exceed the yield strength because yielding is not considered in the material model. In reality, S355JR steel begins to plastically deform once the yield strength is reached, and the stress state becomes limited by the yield surface. Therefore, the calculated elastic stress peaks in the range of 680–825 MPa cannot represent the real structural stress state and are expected to be physically limited by plastic yielding.

3.4.3. Classification Framework

Based on the criteria described above, the observed stress peaks were classified into three categories according to their numerical behaviour and physical interpretation. The classification was performed by analysing mesh dependency, spatial correspondence with typical singularity sources, compatibility with the global structural response, and the plastic admissibility of the calculated stresses. This framework enabled a systematic distinction between numerical singularities, physically meaningful stress concentrations, and elastically overestimated stresses that would be constrained by material yielding under real structural conditions.

According to this classification framework, the stress peaks observed in the analysed model were identified as Type 2, corresponding to mesh-dependent numerical singularities.

Table 2: *Stress classification framework.*

Category	Definition	Characteristic
Type 1	Physical stress concentration	Converges with mesh refinement
Type 2	Mesh-dependent numerical singularity	Increases with mesh refinement
Type 3	Actual structural overstress	Accompanied by deterioration in global structural behaviour

3.4.4. Engineering Mitigation Strategies

According to ANSYS technical documentation, if a stress singularity occurs within the region of interest of the analysis, several corrective modelling actions can be applied. Typical approaches include replacing point loads with equivalent pressure distributions, introducing fillets at sharp geometric corners, replacing fully fixed boundary conditions with more realistic contact interfaces, and evaluating averaged stress measures over finite control volumes rather than single-node peak values. These strategies are consistent with the singularity-management practices proposed by Sinclair [20,21] and with the mesh-independent structural stress and Peak Stress Method approaches developed for welded joints [7,16,25]. In the present study, mitigation was applied by combining geometric regularization at re-entrant corners with the substitution of fixed supports by contact-based boundary representations, thereby allowing the interpretation to focus on physically admissible stress fields rather than on isolated numerical extrema.

4. Results and Discussion

4.1. Asymptotic Stress Field Behaviour

Within the framework of linear elasticity theory, the stress field in the vicinity of geometric discontinuities and stiffness transition regions is commonly described by an asymptotic expression of the form:

$$\sigma(r) = K\, r^{(-\lambda)} \quad (5)$$

where r represents the distance from the singularity point and λ denotes the singularity order. When $\lambda > 0$, the stress theoretically tends toward infinity as $r \to 0$. This behaviour is not physical but arises from modelling assumptions and from the mathematical nature of the linear elastic solution.

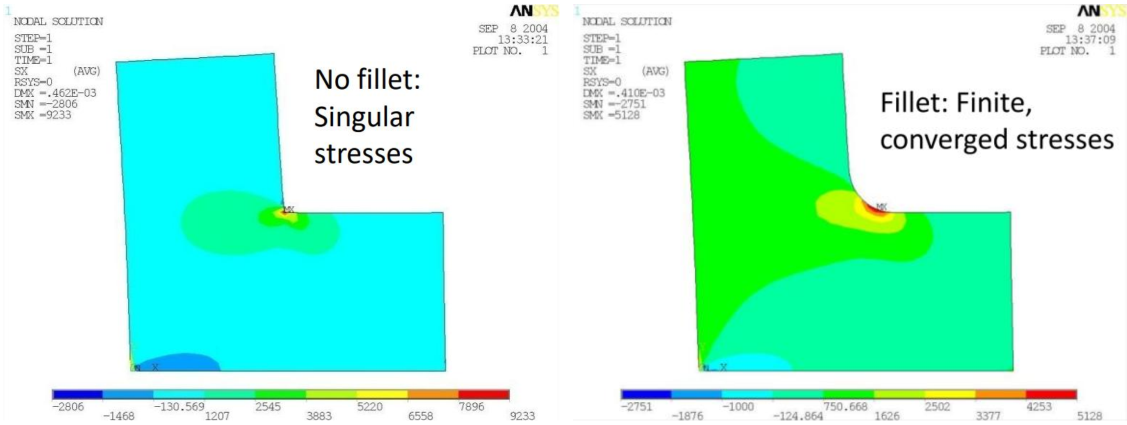

Figure 7: *Effect of geometric regularization on stress fields. Introducing a finite radius eliminates the stress singularity and produces convergent stresses.*

As demonstrated in Figure 7, geometric regularization plays a crucial role in the numerical behaviour of stress fields. Introducing a finite radius at sharp geometric transitions eliminates the mathematical singularity and results in convergent stress values. This observation highlights the fundamental difference between numerical singularities and physically meaningful stress concentrations. In the analyzed crane model, the maximum stress values were observed to concentrate in the hole regions where clamped boundary conditions were applied, at bonded contact transitions, and on the load application surfaces. All of these regions correspond to idealized modelling locations where abrupt stiffness variations occur. When mesh refinement was performed, the stress peaks did not converge and instead showed an increasing trend. This behaviour is consistent with the asymptotic growth predicted by classical singularity theory.

4.2. Energy-Based Interpretation

From a physical interpretation standpoint, the decisive factor in evaluating singularities is not only the magnitude of the stress but also the energy behaviour of the system. The elastic strain energy density is defined as

$$W = \tfrac{1}{2}\,\sigma_{ij}\,\varepsilon_{ij} \qquad (6)$$

When the asymptotic stress expression is substituted into this formulation, the energy density varies approximately as $W(r) \sim r^{(-2\lambda)}$. The total elastic energy of the system can be expressed as the volume integral:

$$U = \int_V W\, dV \qquad (7)$$

In cylindrical coordinates, this integral can be reduced to:

$$U \sim \int_0^{r_0} r^{(1-2\lambda)}\, dr \qquad (8)$$

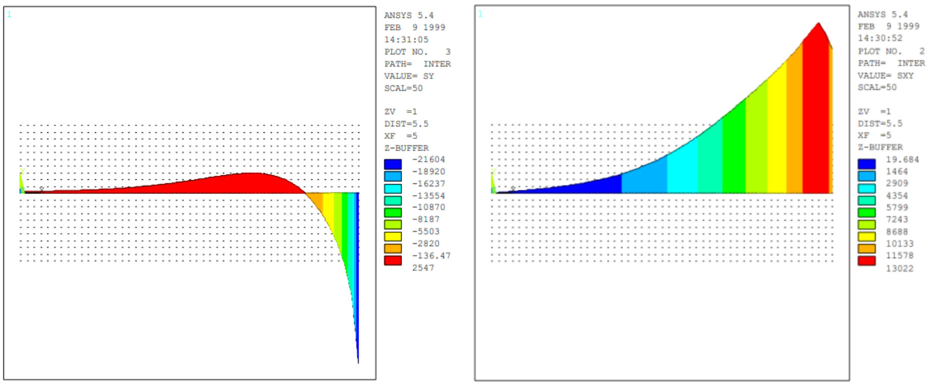

Figure 8: *Example of non-converging stress distributions near a singular region in finite element analysis.*

The stress distributions presented in Figure 8 clearly illustrate the non-converging behaviour typical of singular stress fields. As the evaluation point approaches the singular location, the stress magnitude increases rapidly rather than stabilizing at a finite value. Such behaviour indicates that the observed stress peak originates from asymptotic mathematical growth rather than from a physically distributed load transfer mechanism. For this integral to converge, the condition $\lambda < 1$ must be satisfied. In most structural singularities caused by boundary conditions, this condition is fulfilled. Under such circumstances, although the stress theoretically approaches infinity, the total strain energy remains finite. In the present analysis, the maximum displacement values were limited to 6.60 mm and 16.24 mm, indicating that the global energy balance of the system is preserved and that no loss of structural stiffness has occurred. This observation supports the mathematical nature of the singularity. While the stress field grows locally, the global structural response remains within physically acceptable limits.

4.3. Limits of the Elastic Solution and Plastic Regularization

In a linear elastic solution, the stress–strain relationship is defined by $\sigma = E\varepsilon$ (Eq. 1) and therefore does not include any yield limitation. As a result, the numerical model is mathematically capable of producing stress values exceeding the yield strength of the material. In real ductile steels, however, once the equivalent stress reaches the yield strength, plastic deformation begins and the stress field becomes limited by the yield surface. Plastic deformation acts as a natural regularization mechanism that prevents stresses from increasing indefinitely. For this reason, the stress peaks calculated in the linear elastic analysis, which lie in the range of 680–825 MPa, do not represent physically attainable stress states. These values arise from the mathematical growth permitted by the elastic formulation when plasticity is not included in the constitutive model.

4.4. Separation of Local and Global Structural Behaviour

In a real structural overstress condition, a reduction in global stiffness, uncontrolled growth in displacement, or a loss of structural stability would typically be expected. However, in the present study the global deformation behaviour of the system remained within serviceable limits. The energy integral remained finite and the overall load-carrying mechanism of the structure was preserved. Therefore, the observed stress peaks do not indicate actual structural failure but rather correspond to local asymptotic stress growth caused by modelling idealizations. The analyzed stress distribution can therefore be classified as a boundary-condition-induced numerical singularity rather than a physical stress concentration.

4.5. Implications for Industrial FEM Interpretation

These findings demonstrate that the maximum stress value alone cannot be used as a reliable safety indicator in large-scale linear elastic finite element analyses. In order to properly interpret stress peaks, the asymptotic behaviour of the stress field, the convergence characteristics of the energy response, and the plastic admissibility of the material must be evaluated together. Otherwise, purely mathematical singularities may be misinterpreted as structural failure, potentially leading to unnecessary design modifications and overly conservative engineering decisions.

5. Conclusions

5.1. Model Scale and Numerical Characteristics

The structure analyzed in this study corresponds to a large-scale industrial finite element model consisting of approximately one million elements. The model contains a total of 2,027,936 nodes and 990,617 second-order tetrahedral (TE10) elements. This numerical scale indicates that the solution possesses sufficient spatial resolution to represent the global structural behaviour rather than local numerical fluctuations. An evaluation of the mesh quality criteria shows that the majority of the elements fall within acceptable limits. Approximately 87.49% of the elements satisfy the aspect ratio criterion within the "good" quality range, with an average aspect ratio value of 3.056. The distributions of maximum and minimum element angles fall within acceptable limits for 96.68% and 95.18% of the elements, respectively. The average skewness value is 0.371, and 95.34% of the elements belong to the good-quality category. These mesh quality indicators demonstrate that the observed stress peaks are not caused by element distortion, excessive element deformation, or numerical instability. Therefore, the high stress values obtained in the analysis can be attributed to the nature of the boundary conditions and geometric discontinuities defined in the model rather than to poor mesh quality or numerical errors. Considering both the scale of the model and the adequacy of the mesh quality, the analysis results represent the behaviour of a realistic industrial structural system rather than that of simplified academic examples. This allows the stress singularity discussion presented in this study to be evaluated within a structural interpretation framework rather than as a consequence of numerical inadequacy.

5.2. Maximum Displacement and Global Structural Behaviour

As a result of the linear elastic analysis performed under service loading conditions, the maximum displacement of the structure was calculated as 6.60265 mm. The minimum displacement value is zero, which corresponds to the expected physical behaviour at the fixed support regions. Examination of the displacement distribution shows that the deformation exhibits a smooth and continuous bending profile along the girder. The maximum displacement occurs near the mid-span region, while the displacement magnitude gradually decreases toward the support regions. This distribution is consistent with classical bending theory for a beam subjected to end loading and indicates that the global stiffness behaviour of the structure is physically consistent.

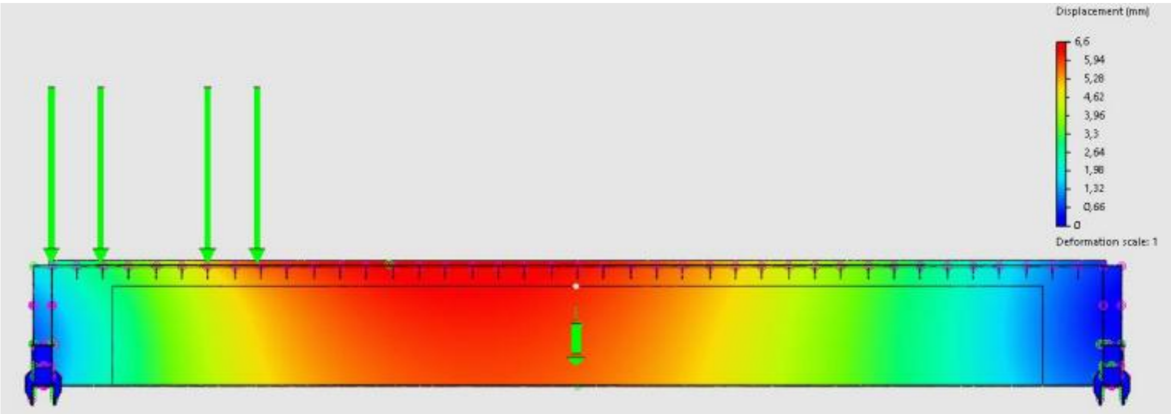

Figure 12: *Displacement result of the crane beam under service loading conditions.*

Despite the presence of extremely high localized stress values, the global displacement field shown in Figure 9 exhibits a smooth and continuous deformation pattern. This behaviour is consistent with classical beam bending theory and indicates that the structural system retains its global stiffness under the applied loading conditions. This result plays a critical role in supporting the central argument of the present study. Although the maximum von Mises stress obtained in the same analysis reaches approximately three times the nominal yield strength of the material, the global deformation response of the structure remains limited and physically reasonable under service-level loading conditions. If the calculated high stress values represented a true structural overload condition, a significant loss of stiffness or a deviation from the linear elastic response would be expected in the global displacement behaviour of the system. However, no such anomaly is observed in the displacement field.

The continuity and smooth gradient of the displacement distribution also indicate that the numerical solution is stable. Despite the presence of very high stress values in the singularity regions, no discontinuities or irregular oscillations are observed in the displacement field. This suggests that the extreme stress values are concentrated within a very limited local region and do not control the global structural behaviour. In this context, the maximum displacement result serves as an important global performance indicator supporting the conclusion that the excessive stress values obtained in the analysis cannot be directly interpreted as indicators of structural failure. Since the deformation behaviour remains physically consistent, stable, and compatible with the expected bending mechanics under service loading, the local extreme stress values in the stress field should be evaluated within the framework of mathematical stress singularities.

5.3. Maximum von Mises Stress and Comparison with Yield Strength

The maximum equivalent (Von Mises) stress obtained from the analysis was calculated as 825.487 MPa. The material used in the model is S355JR structural steel, which has a 0.2% yield strength of 275 MPa. Accordingly, the calculated maximum stress value corresponds to approximately three times the yield strength:

$$\sigma_{max} / \sigma_y \approx 825 / 275 \approx 3 \qquad (9)$$

At first glance, this ratio may suggest that the structure is subjected to a severe overstress condition if the results of the linear elastic analysis are interpreted directly in terms of structural safety. However, a detailed examination of the stress distribution shows that this high value is concentrated within an extremely localized region. The majority of the global stress field remains within the range of 80–250 MPa, which lies within the yield strength limits of the material. The maximum stress values occur primarily in regions where rigid connections are modeled using bonded contact definitions and in areas containing sharp geometric discontinuities. In these regions, the stress field exhibits a very steep gradient, and the peak stress value is concentrated within a very small volume. Stress contour plots clearly indicate that the red high-stress region remains limited to the scale of individual elements, while the stress magnitude decreases rapidly in the surrounding elements. This behavior is consistent with the mathematical singularity characteristics described in classical elasticity theory. In the presence of sharp geometric corners or perfectly rigid boundary conditions, the stress field may increase asymptotically as the distance approaches zero. In contrast, if a real yielding condition were present, the stress increase would be expected to spread over a finite region through plastic deformation and would produce a measurable effect on the global stiffness of the structure. However, no such stiffness reduction is observed in the displacement field. Furthermore, since a linear elastic material model is used in the present analysis, the stress redistribution mechanism that occurs after yielding is not activated. In real material behavior, once plastic deformation begins, the stress field becomes limited and cannot increase indefinitely. Therefore, the 825 MPa value obtained in the elastic analysis does not represent a physically sustainable stress state but rather a numerical amplification resulting from the boundary conditions and geometric idealizations defined in the model. Consequently, the maximum von Mises stress value alone should not be interpreted as an indicator of structural inadequacy. When the localization of the stress field is evaluated together with the global deformation response, it becomes evident that the extreme stress values correspond to a mathematical singularity rather than to a physically meaningful overstress condition.

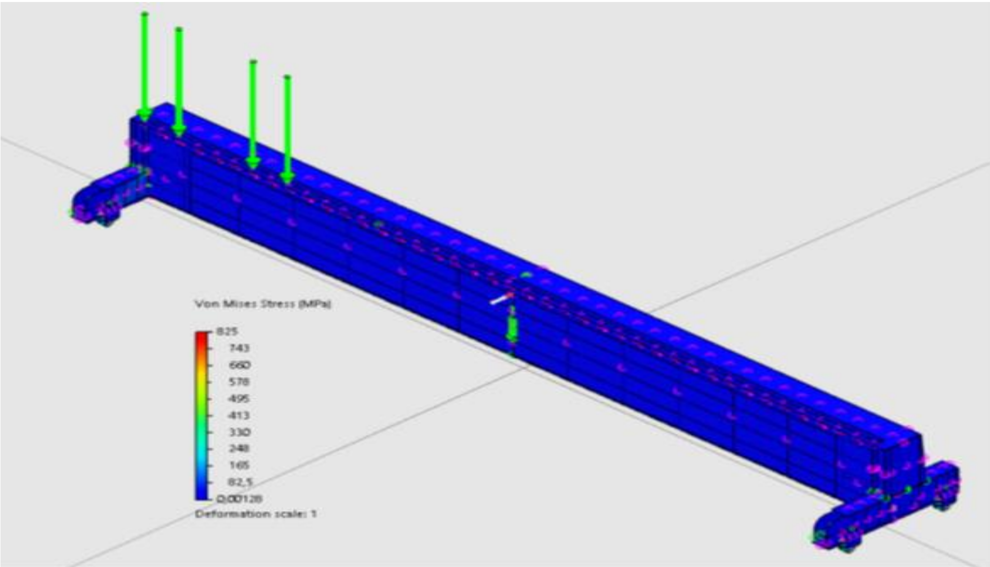

Figure 13: *Von Mises stress distribution of the crane beam.*

5.4. Mesh Quality and Numerical Reliability Assessment

The von Mises stress distribution illustrated in Figure 10 reveals that the extreme stress values are confined to a very limited region near geometric discontinuities and boundary interfaces. Outside these localized zones, the majority of the structure experiences stress levels well within the elastic range of the material. In order to technically support the singularity interpretation, it must first be demonstrated that the high stress values obtained in the analysis are not caused by poor mesh quality. For this reason, the mesh characteristics of the model were evaluated in detail. The finite element model used in the analysis consists of a total of 990,617 second-order tetrahedral (TE10) elements, and the element quality statistics were comprehensively examined. An evaluation of the mesh quality criteria shows that the majority of the elements fall within acceptable engineering limits. Approximately 87.49% of the elements satisfy the aspect ratio criterion within the "good" quality range, with an average aspect ratio value of 3.056. The maximum angle criterion falls within the acceptable range for 96.68% of the elements, with an average value of 105.802°. The minimum angle criterion is acceptable for 95.18% of the elements, with an average value of 29.439°. The average skewness value is 0.371, and 95.34% of the elements belong to the good-quality category. In terms of the stretch criterion, 87.81% of the elements also satisfy the good-quality range. These indicators demonstrate that the model does not contain element distortion, excessive element stretching, deteriorated angle distributions, or numerical instability related to element shape quality. In particular, the high percentage of elements classified as "good" in terms of skewness and minimum angle distribution indicates that the stress calculations are not significantly influenced by geometric discretization errors. An examination of the mesh structure in the regions where the stress peaks occur shows that the elements are not distorted or excessively elongated; instead, a homogeneous and numerically stable discretization is achieved. Therefore, the stress peaks reaching 825 MPa cannot be attributed to deficiencies in mesh quality but must instead be interpreted as physical stress concentrations or mathematical singular behaviour caused by boundary condition idealizations.

5.5. Overall Interpretation

Taken together, the model scale, mesh quality, global displacement behaviour, and localized peak stress magnitude collectively demonstrate that the high stress values observed in the analysis do not represent a real structural overstress condition but instead exhibit the characteristics of mathematical singularities. The study demonstrates, through a large-scale industrial model, the distinction between physical stress concentrations and numerical singularities, highlighting the importance of careful interpretation of finite element results in engineering practice. The proposed evaluation approach reduces the risk of unnecessary overdesign while also preventing incorrect safety assessments. In large-scale structural analyses, not only the magnitude of the stress field but also its behavioural characteristics and physical plausibility must be considered.

STATEMENTS AND DECLARATIONS

Supplementary Material

No supplementary material accompanies this article.

Acknowledgements

The authors gratefully acknowledge BVS R&D Center for providing the industrial CAD model, licensed software resources, and computational infrastructure used in this study.

Author Contributions

S. Dönerkaya: Conceptualization, Methodology, Formal Analysis, Investigation, Writing – Original Draft. S. Büken: Methodology, Software, Validation, Writing – Review & Editing.

Conflicts of Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Data, Code & Protocol Availability

The data supporting the findings of this study are available from the corresponding author upon reasonable request. Model files remain the intellectual property of BVS R&D Center and are subject to the company's data-sharing policy.

Funding Received

This research did not receive any specific grant from funding agencies in the public, commercial, or not-for-profit sectors.

ORCID IDs

Samet Dönerkaya https://orcid.org/0000-0001-7577-5124
Samet Büken https://orcid.org/0009-0004-8824-5199

REFERENCES

1. J. R. Rice, "A path independent integral and the approximate analysis of strain concentration by notches and cracks," J. Appl. Mech., vol. 35, no. 2, pp. 379–386, 1968.
2. A. Carpinteri and M. Paggi, "Asymptotic analysis of stress singularities in composite laminates," Compos. Struct., vol. 71, pp. 210–219, 2005.
3. J. Bulling, H. Gravenkamp, and C. Birk, "A high-order finite element technique with automatic treatment of stress singularities by semi-analytical enrichment," Comput. Methods Appl. Mech. Eng., vol. 355, pp. 135–156, 2019, doi: 10.1016/j.cma.2019.06.013.
4. S. H. Ebrahimi, S. Mohammadi, and I. Mahmoudzadeh Kani, "A local PUFEM modeling of stress singularity in sliding contact with minimal enrichment for direct evaluation of generalized stress intensity factors," Eng. Fract. Mech., vol. 105, pp. 16–40, 2013.
5. Y. Qian and B. Teng, "A finite element–scaled boundary finite element coupled method for corner singularities in 2D second-order radiation problems," Ocean Eng., vol. 281, art. 114910, 2023, doi: 10.1016/j.oceaneng.2023.114910.
6. C. Wang, X. Ping, and X. Wang, "An adaptive finite element method for crack propagation based on a multifunctional super singular element," Int. J. Mech. Sci., vol. 247, art. 108191, 2023, doi: 10.1016/j.ijmecsci.2023.108191.
7. P. Dong, "A structural stress definition and numerical implementation for fatigue analysis of welded joints," Int. J. Fatigue, vol. 23, no. 10, pp. 865–876, 2001, doi: 10.1016/S0142-1123(01)00055-X.
8. A. Montanino and A. Iannuzzo, "A quadrilateral plate-type finite element to model stress singularities in no-tension materials," Comput. Methods Appl. Mech. Eng., vol. 432, art. 117433, 2024, doi: 10.1016/j.cma.2024.117433.
9. L. Jing, M. Feng, Y. Li, F. Teng, and Z. Luo, "A reduced-dimension method for unknown Crank–Nicolson finite element solution coefficient vectors of elastic wave equation with singular source term," J. Math. Anal. Appl., vol. 540, art. 128629, 2024, doi: 10.1016/j.jmaa.2024.128629.
10. G. B. Sinclair and B. P. Epps, "On the logarithmic stress singularities induced by the use of displacement shape functions in boundary conditions in submodelling," Commun. Numer. Methods Eng., vol. 18, pp. 121–130, 2002, doi: 10.1002/cnm.473.
11. M. A. Herrera-Garrido, L. Távara, M. Muñoz-Reja, A. Vázquez-Sánchez, and V. Mantič, "Enriched crack-tip element for logarithmic stress singularities of Mode I and II cracks in spring interfaces," Eng. Fract. Mech., vol. 336, art. 111941, 2026.
12. T. Yue and M. Abdel Wahab, "Finite element analysis of stress singularity in partial slip and gross sliding regimes in fretting wear," Wear, vol. 321, pp. 53–63, 2014, doi: 10.1016/j.wear.2014.09.008.
13. X.-C. Ping, M.-C. Chen, and J.-L. Xie, "Singular stress analyses of V-notched anisotropic plates based on a novel finite element method," Eng. Fract. Mech., vol. 75, pp. 3819–3838, 2008, doi: 10.1016/j.engfracmech.2008.01.003.
14. S. S. Pageau, P. F. Joseph, and S. B. Biggers Jr., "Finite element analysis of anisotropic materials with singular in-plane stress fields," Int. J. Solids Struct., vol. 32, no. 5, pp. 571–591, 1995, doi: 10.1016/0020-7683(94)00152-9.
15. W. Pan, C. Cheng, F. Wang, Z. Hu, and J. Li, "Determination of singular and higher order non-singular stress for angularly heterogeneous material notch," Eng. Fract. Mech., vol. 292, art. 109592, 2023, doi: 10.1016/j.engfracmech.2023.109592.
16. I. Poutiainen, P. Tanskanen, and G. Marquis, "Finite element methods for structural hot spot stress determination — a comparison of procedures," Int. J. Fatigue, vol. 26, pp. 1147–1157, 2004, doi: 10.1016/j.ijfatigue.2004.04.004.
17. A. Visentini, A. Campagnolo, V. Babini, and G. Meneghetti, "Automated implementation of the Peak Stress Method for the fatigue assessment of complex welded structures," Forces Mech., vol. 6, art. 100072, 2022, doi: 10.1016/j.finmec.2021.100072.
18. Q. Lei and D. Sornette, "Oscillatory finite-time singularities in rockbursts," Int. J. Rock Mech. Min. Sci., vol. 192, art. 106156, 2025, doi: 10.1016/j.ijrmms.2024.106156.
19. M. L. Williams, "Stress singularities resulting from various boundary conditions in angular corners of plates in extension," J. Appl. Mech., vol. 19, pp. 526–528, 1952, doi: 10.1115/1.4010550.
20. G. B. Sinclair, "Stress singularities in classical elasticity—I: Removal, interpretation, and analysis," Int. J. Numer. Methods Eng., vol. 60, no. 5, pp. 993–1018, 2004, doi: 10.1002/nme.1002.
21. G. B. Sinclair, "Stress singularities in classical elasticity—II: Asymptotic identification," Int. J. Numer. Methods Eng., vol. 60, no. 6, pp. 1019–1045, 2004, doi: 10.1002/nme.1003.

22. Y. Huang, C. Cheng, Z. Hu, D. Kondo, and R. Das, "Stress singularity analysis for the V-notch with a novel semi-analytical boundary element," Eng. Anal. Bound. Elem., vol. 163, pp. 84–93, 2024, doi: 10.1016/j.enganabound.2024.03.006.
23. T. Meshii and K. Watanabe, "Stress intensity factor error index for finite element analysis with singular elements," Eng. Fract. Mech., vol. 70, pp. 657–669, 2003, doi: 10.1016/S0013-7944(02)00083-9.
24. D. Radaj, "State-of-the-art review on the local strain energy density concept and its relation to the J-integral and peak stress method," Fatigue Fract. Eng. Mater. Struct., vol. 37, no. 10, pp. 1091–1112, 2014, doi: 10.1111/ffe.12231.
25. A. Visentini, A. Campagnolo, V. Babini, and G. Meneghetti, "Automated implementation of the Peak Stress Method for the fatigue assessment of complex welded structures," Forces Mech., vol. 6, art. 100072, 2022, doi: 10.1016/j.finmec.2021.100072.