Article

Not peer-reviewed version

Finite Element Stress Analysis of the Developed Coupler Component of Freight Rail Car

Nutthapong Kunla and Anan Suebsomran *

Posted Date: 16 March 2026

doi: 10.20944/preprints202603.1138.v1

Keywords: type E coupler; finite element analysis (FEA); assembly-level analysis; stress concentration; load redistribution

doi:10.20944/preprints202603.1138.v1

Article

Finite Element Stress Analysis of the Developed Coupler Component of Freight Rail Car

Nutthapong Kunla [1] and Anan Suebsomran [2,*]

[1] Department of Mechanical Engineering, Faculty of Engineering, Bangkok Thonburi University, 16, 10 Thawi Watthana, Bangkok 10170, Thailand
[2] Department of Teacher Training in Mechanical Engineering, King Mongkut's University of Technology North Bangkok, 1518 Pracharat 1 Rd, Bangsue, Bangkok 10800, Thailand
* Correspondence: anan.s@fte.kmutnb.ac.th

Abstract

This study investigates the structural performance of a redesigned AAR Type E knuckle coupler using finite element analysis (FEA). The modified knuckle incorporates geometric reinforcement in critical load-bearing regions together with a hollow internal structure aimed at reducing component weight while maintaining structural integrity. Two numerical models were developed: a component-level model, in which the knuckle was analyzed independently, and an assembly-level model that integrates the knuckle with the coupler body to capture realistic load transfer through contact interactions. Both models were subjected to a tensile draft load of 650,000 lbs (2670 kN) in accordance with Association of American Railroads (AAR) standards. The component-level analysis predicted peak von Mises stresses of approximately 1050 MPa, primarily concentrated near the pivot pin hole and curved pulling face regions. When contact interactions between the knuckle and coupler body were included in the assembly model, the representative peak stress decreased to approximately 950 MPa, corresponding to a stress reduction of about 10% due to load redistribution across the assembly interfaces. Highly localized stress peaks at sharp geometric edges were identified as numerical stress singularities and were excluded from engineering interpretation. The results demonstrate that assembly-level finite element modeling provides a more realistic representation of load transfer mechanisms in railway coupler systems and is essential for accurately predicting stress distribution and identifying critical fatigue-prone regions. These findings provide valuable insights for improving the structural reliability and design optimization of freight rail coupler components.

Keywords: type E coupler; finite element analysis (FEA); assembly-level analysis; stress concentration; load redistribution

---

1. Introduction

Railway freight transportation plays a crucial role in modern logistics systems due to its high transport capacity, operational efficiency, and relatively low environmental impact compared with road transport. As global demand for sustainable freight transport continues to increase, railway systems have become increasingly important for supporting economic development while reducing greenhouse gas emissions associated with long-distance transportation [19]. Ensuring the structural reliability and safety of railway components is therefore essential for maintaining stable railway operations.

Among the mechanical components of freight rail vehicles, the coupler system is one of the most critical structural elements. Couplers mechanically connect adjacent railcars and transmit longitudinal draft and buff forces generated during acceleration, braking, and shunting operations. The Association of American Railroads (AAR) Type E coupler, commonly referred to as the Janney or knuckle coupler, remains the most widely adopted coupling system in freight railway applications due to its automatic coupling capability and ability to withstand substantial in-train forces. Despite its robust design, the knuckle component of the coupler assembly remains susceptible to fatigue damage and fracture under repeated service loading conditions. Since the coupler transmits large longitudinal forces between rail vehicles, failure of this component may lead to train separation and potentially hazardous operational conditions [1].

Due to its importance in railway safety, numerous studies have investigated the structural performance and failure mechanisms of coupler systems. Song et al.[2] performed finite element analyses of coupler knuckles and reported that maximum stresses typically occur near the knuckle corner region, which corresponds closely with crack initiation locations observed in service failures [3]. Similarly, Chunduru et al. [1] demonstrated that the pulling lug and pivot pin regions experience significant stress concentrations under tensile loading conditions, making them susceptible to fatigue crack initiation.

Several investigations have also examined design improvements and fatigue behavior of coupler components. Steed and Kimpton [4] proposed an improved heavy-duty coupler knuckle configuration capable of achieving higher strength and fatigue resistance compared with conventional designs. Gonzales et al. [5] further investigated stress distributions in coupler systems through combined experimental measurements and numerical simulations using strain gauges. Their results confirmed that finite element simulations can accurately capture stress concentrations in critical regions of coupler components. Wang et al. [6] studied the structural behavior of forged coupler knuckles under high draft loading conditions and identified the key structural regions controlling failure initiation. Li and Sun [7] later proposed fatigue life prediction models for heavy-haul coupler knuckles based on local stress–strain approaches.

In addition to structural strength studies, several researchers have investigated the dynamic behavior of coupler systems at the train level. Early analytical models of longitudinal train dynamics were developed to analyze the transmission of in-train forces between rail vehicles [8,9]. These models were later extended to incorporate more detailed coupler mechanics and vehicle interaction effects. Wu et al. [10] emphasized the importance of accurate coupler modeling in predicting longitudinal force transmission and train dynamics. Similarly, Yadav and Vyas [11] demonstrated that coupler systems are subjected to significant longitudinal and lateral forces due to intermittent impacts and slack action between railcars. Yuan et al. [12] further investigated the stability of heavy-haul couplers with arc-surface contact and demonstrated that coupler geometry significantly influences load transmission and operational stability. Similarly, simulation studies on coupling performance have shown that vehicle offset and coupler swing angles can significantly influence coupling safety and operational reliability under curved track conditions [13]

Recent research has also explored advanced monitoring and diagnostic techniques for railway coupler systems. Vision-based monitoring approaches have been proposed to estimate coupler displacement and alignment during freight wagon operation, enabling improved condition monitoring and maintenance planning for railway systems [14] .

Beyond coupler systems, fatigue failures have also been extensively studied in other critical railway components. Railway axles, for example, are subjected to severe cyclic bending loads during long service periods and may develop fatigue cracks that propagate until catastrophic failure occurs. Numerical investigations have demonstrated that fracture mechanics-based simulations can effectively predict crack propagation behavior and residual service life of railway axles under operational loading conditions [15].

With the advancement of computational mechanics, numerical simulation techniques have become powerful tools for analyzing structural behavior and predicting failure mechanisms in complex engineering systems. Finite element methods and fracture mechanics approaches have been widely applied to simulate crack propagation and fatigue behavior in engineering structures [16–20]. In railway applications, these numerical frameworks have been successfully used to evaluate crack growth behavior and estimate residual life of structural components subjected to cyclic loading [21,22].

Despite the considerable progress in coupler research, most previous investigations have primarily focused on failure diagnosis, fatigue behavior, or operational performance of existing coupler designs. Comparatively limited attention has been given to the structural behavior of newly developed coupler geometries incorporating geometric modifications intended to improve durability while reducing component weight. Furthermore, many finite element analyses have examined coupler components independently without explicitly considering the mechanical interactions between the knuckle and the coupler body. In practical railway operations, however, these components function as part of an integrated mechanical assembly in which contact interactions and load transfer mechanisms significantly influence stress distribution and structural response.

To address these limitations, the present study investigates the structural behavior of a redesigned AAR Type E coupler knuckle using finite element analysis. The redesigned knuckle incorporates geometric reinforcement in critical load-bearing regions together with a hollow internal structure aimed at reducing component mass while maintaining structural integrity. Two numerical models were developed: a component-level model evaluating the knuckle independently and an assembly-level model integrating the knuckle with the coupler body to simulate realistic load transfer through contact interactions.

Both models were subjected to a tensile draft load of 650,000 lbs (2670 kN) in accordance with AAR design standards. By comparing the stress distributions predicted by the component-level and assembly-level analyses, this study provides new insights into how assembly interactions influence stress redistribution within the coupler system.

The main contributions of this study can be summarized as follows:

1. Finite element stress analysis of a redesigned AAR Type E coupler knuckle developed to improve structural durability while reducing weight.
2. Comparative evaluation between component-level and assembly-level finite element models of the knuckle–coupler system.
3. Assessment of stress distribution under realistic AAR draft loading conditions.
4. Identification of critical structural regions affecting the reliability and fatigue resistance of freight rail coupler assemblies.

By integrating advanced numerical modeling with practical design modifications, this research contributes to improving the safety, reliability, and cost-effectiveness of heavy-haul freight railway operations.

The remainder of this paper is organized as follows. Section 2 describes the research methodology, including geometry acquisition, finite element modeling, mesh convergence analysis, and boundary condition definitions. Section 3 presents the redesigned knuckle model and its structural modifications. Section 4 reviews relevant AAR specifications. Section 5 summarizes material properties, while Section 6 details the finite element analysis procedures. Section 7 discusses and compares simulation results, followed by conclusions in Section 8 and future research directions in Section 9.

2. Methodology

This study employed finite element analysis (FEA) to evaluate the structural performance and internal stress distribution of an AAR Type E knuckle coupler. Two numerical models were developed: a **component-level model**, in which the knuckle and coupler were analyzed separately, and an **assembly-level model**, in which both components were integrated to capture realistic load transfer and contact interactions. The overall simulation workflow is illustrated in Figure 1.

doi:10.20944/preprints202603.1138.v1

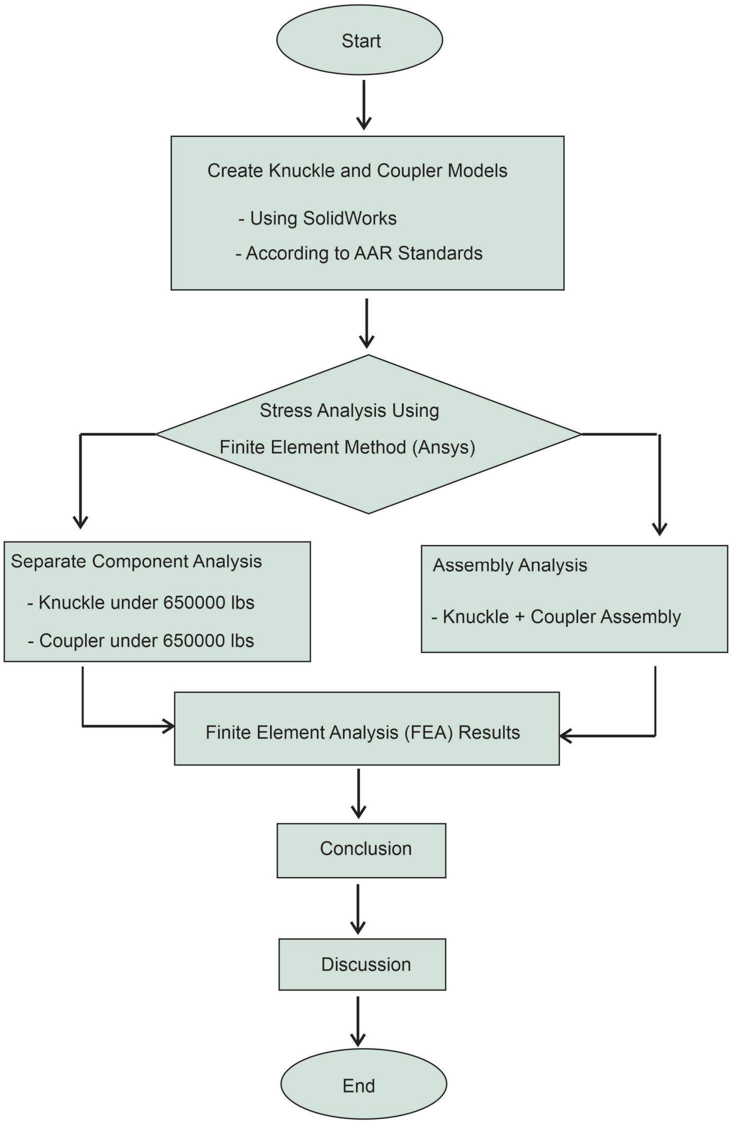


Figure 1. Diagram of the simulation procedure.

The analysis followed the procedures summarized below.

1. Geometry acquisition:

The knuckle geometry was obtained from manufactured components using physical measurements and three-dimensional laser scanning. The resulting point-cloud data were processed to generate high-fidelity CAD models in SolidWorks.

2. Finite element modeling:

The CAD geometry was exported in IGES format and imported into ANSYS Workbench. Three-dimensional quadratic tetrahedral elements (SOLID187) were used to accurately represent curved surfaces and geometric discontinuities.

3. Mesh convergence:

A mesh convergence study was conducted to ensure numerical accuracy. A global element size of 4 mm was selected as an optimal compromise between solution accuracy and computational efficiency.

4. Boundary conditions and loading:

A static tensile load of 650,000 lbs (2670 kN), in accordance with AAR specifications, was applied at the pulling face in the component-level model, with appropriate backstop and pinhole constraints. In the assembly-level model, surface-to-surface contact definitions were applied at critical interfaces, including the pinhole, pulling lug, and backstop, to simulate realistic load transmission between the knuckle and the coupler body.

5. Solution and post-processing:

Linear elastic static analyses were performed. von Mises and principal stress distributions were evaluated to identify critical stress regions and potential failure mechanisms.

Figure 1 summarizes the complete simulation procedure, from geometry acquisition to result interpretation. This framework enables a direct comparison between localized stress behavior and system-level load redistribution in coupler assemblies.

3. Knuckle Model

The knuckle investigated in this study conforms to the standard AAR Type E design widely used in North American freight service. A modified prototype was developed to address critical failure locations reported in previous studies, particularly at the pulling face and pivot pin regions, where high stress concentrations and fatigue cracking are commonly observed [1,23].

Two primary design modifications were introduced:

1. **Localized geometric reinforcement** at the pulling face to reduce stress concentration and improve load-carrying capacity.
2. **A hollow interior** in non-load-bearing regions to reduce mass while preserving structural integrity.

The redesigned knuckle has a mass of 40.36 kg, representing a reduction of approximately 4.09 kg compared with the original 44.45 kg design. A uniform wall thickness of 12 mm was maintained based on manufacturability and structural considerations. The external geometry and internal cross-section of the modified knuckle are shown in Figures 2 and 3.

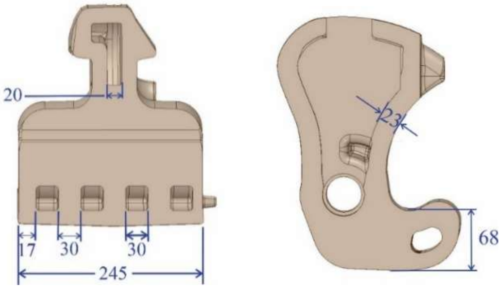


Figure 2. Image of a Knuckle and its dimensions.

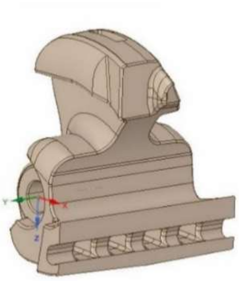


Figure 3. Cross-section image of a knuckle.

doi:10.20944/preprints202603.1138.v1

The geometry was generated using combined laser scanning and CAD modeling to ensure consistency with actual components manufactured in Thailand. Although all scanned knuckles originated from the same production batch, measurable geometric variations were observed, particularly in the internal core geometry, reflecting inherent variability in the casting process. The finalized digital model was subsequently used for finite element evaluation under the prescribed loading conditions.

4. AAR Specifications for Standard Knuckle Coupler Type E

The Association of American Railroads (AAR) defines strict requirements for the geometry, material properties, and mechanical performance of Type E knuckle couplers to ensure interoperability and safety in heavy-haul service. Figures 4(a) and 4(b) illustrate the essential geometric features of a standard Type E knuckle, including the pulling face, pivot pin hole, lug, and tail backstop.

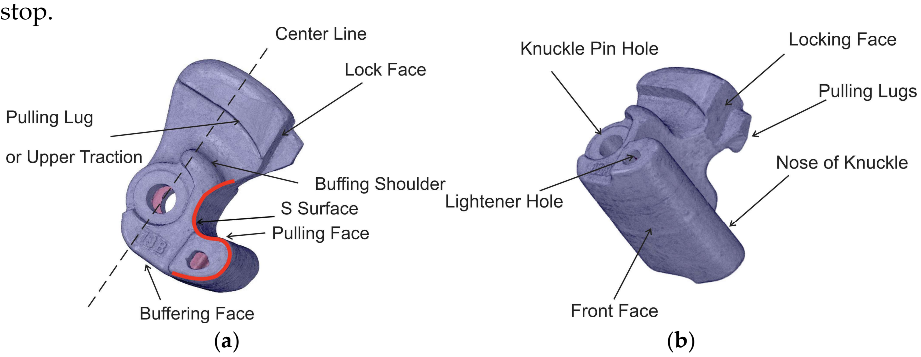


Figure 4. (a) Side aspect showing essential longitudinal characteristics and section thickness (b) Front perspective emphasizing the pulling lug, curvature profile, and contact interface shape.

Figures 4(a) and 4(b) show the essential geometric characteristics of a conventional Type E knuckle, viewed from the lateral and front perspectives, respectively. The key features include the pulling face, pivot pin hole, lug, and tail backstop, all of which are critical for force transfer and structural integrity. Their locations correspond directly to the dimensions and mechanical specifications listed in Tables 1–3.

Table 1. Dimensions of Knuckle Components According to AAR Standards.

Item	**Unit**	**Dimension**
Length	mm	279.4
Width	mm	117.8
Hole Diameter	mm	38.1
Pulling Lug Thickness	mm	22–27.9

Table 2. Test Load Requirements for Coupler and Knuckle According to AAR M -216.

Item	Max. Permanent Set (inches) at 400,000 lbs	Max. Permanent Set (inches) at 700,000 lbs	Min. Ultimate Load (lbs)
Knuckle	0.03	-	650,000
Coupler Body	-	0.03	900,000

Table 3. Coupler and Knuckle Material Composition According to AAR M-211.

Material	C (%)	Si (%)	Mn (%)	P (%)	S (%)	Ni (%)	Cr (%)	Mo (%)
AAR	< 0.32	< 1.5	< 1.85	< 0.04	< 0.04	-	-	-

doi:10.20944/preprints202603.1138.v1

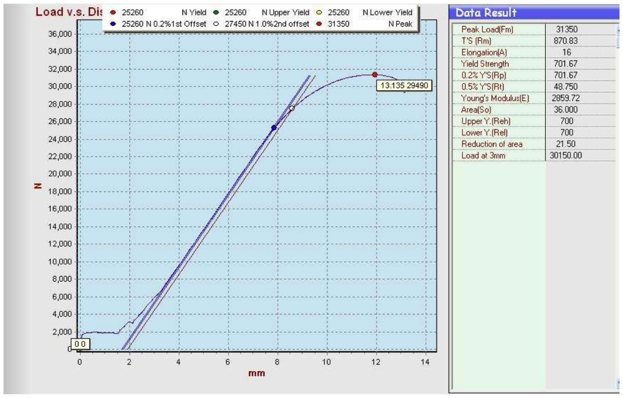
Original Material	0.297	0.983	1.534	0.025	0.015	-	-	-
Improved Material	0.28 – 0.33	0.4 – 1.0	0.8 – 1.5	-	-	> 0.4	> 0.4	> 0.2

Table 1 summarizes the nominal geometric dimensions specified by the AAR 10-A contour, including overall length, width, pivot pin hole diameter, and pulling lug thickness. Table 2 presents the mechanical load requirements according to AAR M-216,[24] which specify a minimum ultimate tensile load of 650,000 lbs for the knuckle. Table 3 lists the allowable chemical composition ranges for AAR M-211[25], ensuring adequate casting quality and mechanical strength for Grade E cast steel.

The material composition and geometric dimensions used in this study conform to the relevant AAR standards. These specifications were directly incorporated into the finite element models, boundary conditions, and loading definitions described in the subsequent sections.

5. Physical Properties of the Knuckle

Tensile stress–strain tests were performed on the coupler knuckle material in accordance with ASTM E8/E8M, the standard protocol for tensile testing of metallic materials. Subsize plate specimens were employed in this study. The tensile test results are summarized in Table 4, with the corresponding stress–strain curve shown in Figure 5. Images of the tested specimens are presented in Figure 6.



Data Result

Peak Load(Fm)	31350
T'S (Rm)	870.83
Elongation(A)	16
Yield Strength	701.67
0.2% Y'S(Rp)	701.67
0.5% Y'S(Rt)	48.750
Young's Modulus(E)	2859.72
Area(So)	36.000
Upper Y.(Reh)	700
Lower Y.(Rel)	700
Reduction of area	21.50
Load at 3mm	30150.00

Figure 5. Stress–Strain curve of knuckle material specimens.

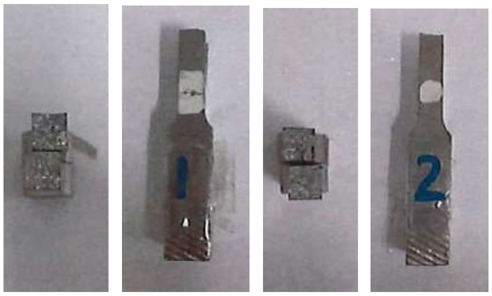


Figure 6. Tensile test specimens of knuckle material.

Table 4. Tensile Test Results of Knuckle Material in accordance with ASTM E8/E8M.

Item	Gauge Length	Tensile Strength (MPa)	Elongation	Tensile Strength (MPa)	% Reduction in Area

doi:10.20944/preprints202603.1138.v1

	L_0	L_1	Target	Actual	Target	Actual	Target	Actual	Target	Actual
1	25	27.37	≥ 827	834.17	≥ 14	9.48	≥ 690	719.44	≥ 30	9.30
2	25	28	≥ 827	904.17	≥ 14	12	≥ 690	777.78	≥ 30	7.52
Average	25	27.68	≥ 827	869.17	≥ 14	10.74	≥ 690	748.61	≥ 30	8.41

Table 4 presents the measured tensile properties of the specimens. The results indicate that the actual tensile strength exceeded the AAR-specified minimum of 827 MPa, with an average value of 869.17 MPa. The average yield strength of 748.61 MPa also surpassed the required minimum of 690 MPa. However, the elongation and reduction in area values were substantially lower than the AAR-specified minimums, indicating limited ductility.

The material used in the stress analysis of the knuckle and coupler was selected in accordance with AAR M 201–05 Grade E standards [26]. The mechanical properties of the materials considered in this study are summarized in Table 5.

Table 5. Mechanical Properties of AAR M-201Grade E Material and a manufacturer's improved version.

Property	Unit	AAR M-201-05 Grade E	Improved Material
Tensile Strength	MPa	827	869.17
Yield Strength	MPa	690	748.61
Elongation	%	14	—
Reduction of Area	%	30	38
Hardness (HB10/3000)	—	241–311	258
Charpy Impact Energy	J	27	38.3

6. Finite Element Analysis

In evaluating structural strength using finite element analysis (FEA), Adams and Askenazi [22] classified numerical investigations into two primary approaches: **component-level stress analysis** and **assembly-level stress analysis**, each serving distinct analytical objectives.

Component-level stress analysis examines an individual component in isolation, neglecting interactions with adjacent parts. This approach is well suited for identifying localized stress concentrations in geometrically complex regions. For knuckle couplers, critical locations typically include the pivot pin hole and pulling lug, where high stresses and crack initiation are frequently reported. Component-level analysis enables accurate identification of the magnitude and location of these stress concentrations.

Assembly-level stress analysis, in contrast, evaluates the structural behavior of interconnected components. Modeling the knuckle together with the coupler body allows realistic assessment of load transfer through contact interfaces, which strongly influences stress distribution and deformation. Adams and Askenazi [22] emphasized that accurate representation of contact conditions, frictional behavior, and boundary constraints is essential for reproducing actual service conditions.

Consistent with this framework, the present study employs both component-level and assembly-level analyses to evaluate the structural durability of the Type E knuckle coupler, which is subjected to severe tensile and compressive loads during freight train operations.

doi:10.20944/preprints202603.1138.v1

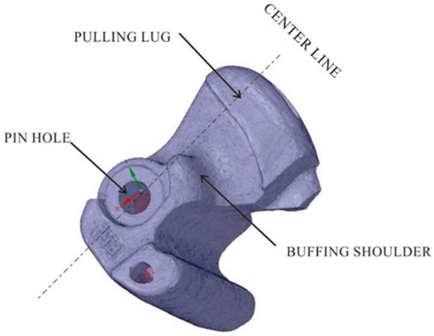


Figure 7. illustrates the three-dimensional CAD model of the Type E knuckle coupler.

6.1. *Finite Element Modeling of the Knuckle and Coupler*

6.1.1. Finite Element Modelling and Meshing of the Knuckle and Coupler

Prototype three-dimensional CAD models were developed based on a knuckle used in Thailand. Geometry acquisition was performed using a combination of physical measurements and 3D laser scanning (GOM Compact Scan 2M, GOM Metrology, Germany). The scanned surface data were imported into SolidWorks and converted into a complete solid model incorporating both external and internal features, as shown in Figure 8. The finalized geometry was exported to ANSYS Workbench in IGES format.

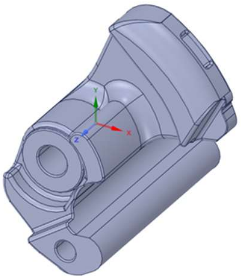


Figure 8. 3D model of the Type E knuckle (SolidWorks software).

The solid model was discretized using 10-node quadratic tetrahedral elements (SOLID187), which are well suited for complex and irregular geometries [23]. Linear elastic material behavior was assumed throughout the analysis. Meshing was performed using proximity- and curvature-based sizing with medium relevance and smoothing. The final mesh consisted of 399,450 elements and 629,633 nodes (Figure 9). A mesh convergence study confirmed that an element size of 0.004 m provided sufficient accuracy with reasonable computational cost.

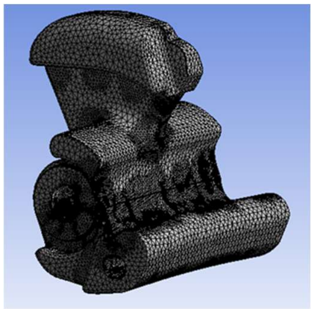
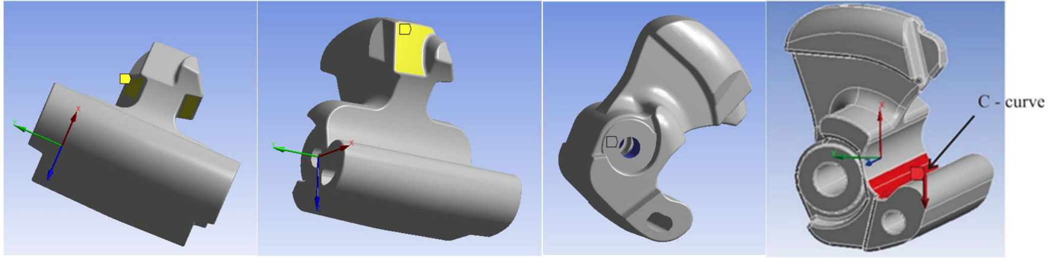


Figure 9. Meshing of the Type-E knuckle 3D model.

6.1.2. Boundary Conditions

- **Knuckle Component**

Boundary conditions for the knuckle component were defined to replicate its mechanical interaction with the coupler body under tensile loading. A tensile force was applied normal to the pulling face and directed toward the knuckle base, representing operational draft loading, as illustrated in Figure 10(d) [1].



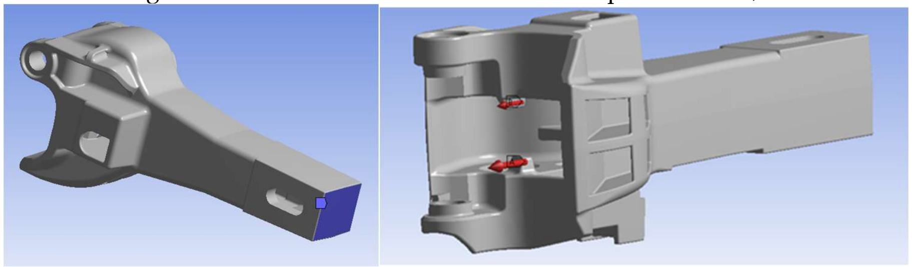
(**a**) (**b**) (**c**) (**d**)

Figure 10. Boundary conditions applied to the knuckle component: (a) overview of the boundary conditions, (b) fixed support at the tail backstop region, (c) cylindrical joint at the pivot pin hole allowing rotation about the pin axis, and (d) tensile load applied normal to the pulling face.

A cylindrical joint constraint was applied at the pivot pin hole to model the pin connection between the knuckle and coupler body (Figure 10(c)). This constraint restricts translational motion while allowing free rotation about the pin axis. Additionally, a fixed support was applied at the tail backstop region to prevent rigid-body motion and represent structural support from the coupler body (Figure 10(b)).

- **Coupler Component**

For the coupler component, a fixed support was applied at the coupler tail, while tensile loading was transmitted through contact interactions at the knuckle–coupler interface, as shown in Figure 11.



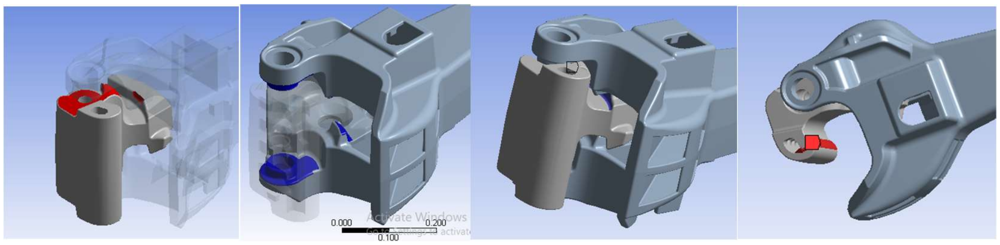
(**a**) (**b**)

Figure 11. (a) boundary on the coupler Component (b) Tensile load applied to transfer compressive force at the knuckle–coupler interface.

- **Knuckle–Coupler Assembly**

Assembly-level boundary conditions followed the methodology reported by Kevin and Kari [5]. A draft load of 650,000 lbs (2670 kN) was applied to the pulling face of the knuckle. Surface-to-surface contact interactions were defined at all critical interfaces, including the pivot pin hole, pulling lug, and knuckle–coupler contact surfaces.

Frictional contact was employed with hard contact behavior in the normal direction, allowing compressive force transfer while preventing penetration. In the tangential direction, a Coulomb friction model with a coefficient of friction $\mu = 0.3$ was adopted, consistent with dry steel-to-steel contact reported in previous studies. The contact formulation allowed separation under tensile loading, ensuring realistic load redistribution within the assembly (Figure 12).



(**a**) (**b**) (**c**) (**d**)

Figure 12. (a) overview of contact regions (b) frictional contact defined at the pivot pin and pulling lug interfaces (c) contact interaction enabling compressive force transfer while allowing separation under tensile loading, and (d) tensile draft load applied normal to the red curved surface.

6.2. Model Verification and Mesh Convergence

To ensure the reliability and numerical stability of the finite element results, a mesh convergence study was performed. Mesh convergence analysis is an essential step in finite element modeling because the predicted stress values may vary significantly depending on the discretization level of the model. Following commonly adopted procedures in structural simulations, adaptive mesh refinement was applied to regions exhibiting high stress gradients while maintaining a coarser mesh in regions with relatively uniform stress distribution [23,27]

The convergence criterion proposed by Bozkurt [16] was adopted in this study, which limits the allowable variation in maximum equivalent stress between successive mesh refinements to 2%. During the convergence process, the mesh density was gradually increased by refining the element size in critical regions such as the pivot pin hole, pulling lug, and curved pulling face of the knuckle.

Figure 13 presents the convergence history of the maximum stress obtained during successive mesh refinements, while the corresponding mesh parameters are summarized in Table 6. The results show that the variation in maximum stress between the two finest meshes was less than 2%**,** indicating that mesh independence had been achieved. The converged mesh contained 629,633 nodes and 399,450 elements, which provided an appropriate balance between numerical accuracy and computational efficiency.

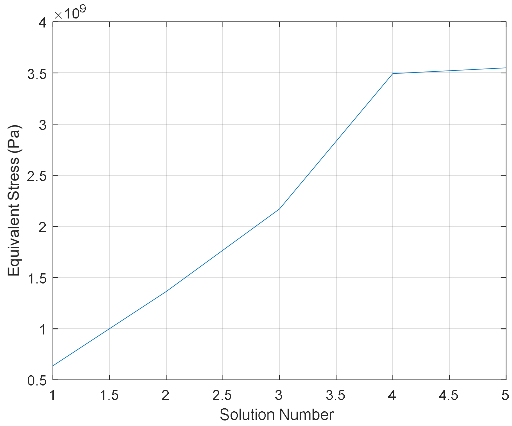


Figure 13. Convergence history of the knuckle obtained using adaptive mesh refinement.

Table 6. Mesh convergence results showing maximum equivalent stress and mesh parameters.

Equivalent Stress (Pa)	**Change**	**Node**	**Element**
1.194 x 10E9		219,317	149,869

2.383 x 10E9	64.6	368,012	257,317
4.007 x 10E9	50.85	682,329	485,146
5.529 x 10E9	31.91	213,850	117,764
8.590 x 10E9	43.36	541,153	340,678
8.615 x 10E9	0.29	629,633	399,450

It should be noted that extremely high localized stress values appeared in certain mesh refinement stages, reaching values on the order of 10^9 Pa. These peaks are attributed to numerical stress singularities caused by sharp geometric features and idealized boundary conditions in the finite element model. Such singular stresses do not represent physically meaningful stress levels in the actual structure and were therefore excluded from engineering interpretation. Instead, representative stress values away from singular regions were used for evaluating the structural behavior of the coupler components.

Based on the convergence analysis, the mesh with an element size of 0.004 m was selected for all subsequent simulations, ensuring that the numerical results were independent of mesh density while maintaining reasonable computational cost.

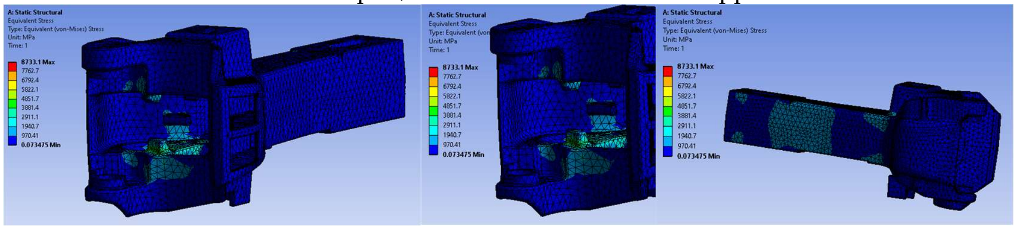
7. Finite Element Analysis Results and Discussion

This section presents the finite element analysis results obtained under a tensile load of 650,000 lbs (2670 kN). Both component-level and assembly-level simulations were performed to evaluate stress distribution in the modified Type E knuckle and coupler, with emphasis on identifying critical stress regions and load-transfer behavior.

7.1. Component-Level Stress Analysis

7.1.1. Coupler

The component-level analysis of the coupler revealed notable stress concentrations near the pulling face and internal contact regions, as shown in Figures 14(a)–14(c). These areas play a key role in load transmission to the knuckle and are therefore susceptible to fatigue damage. Stress levels decreased toward the rear of the coupler, where the influence of the applied tensile load diminished.



(a) (b) (c)

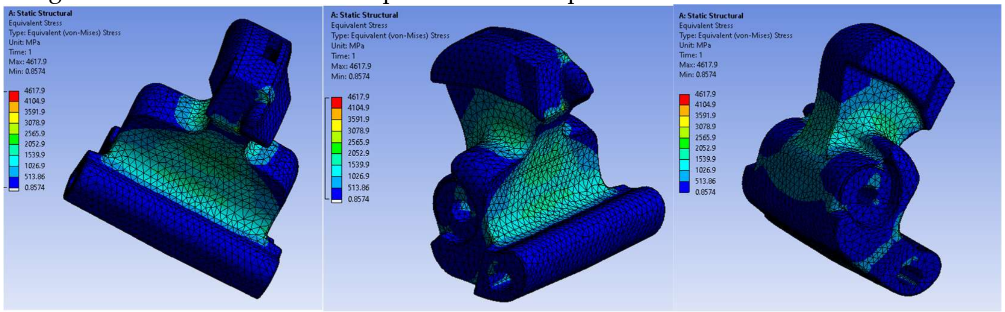
Figure 14. (a) Stress distribution on the coupler in the region of tensile load application (b) enlarged view of principal stress on coupler (c) principal stress on coupler in the back view.

doi:10.20944/preprints202603.1138.v1

The maximum stress in the coupler ranged between approximately 300 and 500 MPa, remaining below the ultimate tensile strength of Grade E cast steel. These findings are consistent with previous studies, which identified the pulling lug and pin-hole regions as primary stress-bearing zones.

7.1.2. Knuckle

The knuckle exhibited a highly non-uniform stress distribution under tensile loading. As shown in Figure 15, pronounced stress concentrations occurred between the pivot pin hole and the curved pulling face due to geometric discontinuities and localized load transfer. Extremely high local stress peaks were observed near sharp geometric features; however, these were attributed to numerical stress singularities and excluded from quantitative interpretation.



(**a**) (**b**) (**c**)

Figure 15. (a) Stress distribution on the knuckle in the region of tensile load application (b) enlarged view of principal stress on knuckle (c) principal stress on knuckle in the back view.

Representative von Mises stress levels in the knuckle were found to be in the range of approximately 1000–1050 MPa. These values correspond to localized stresses predicted under linear elastic assumptions. In practical applications, plastic deformation would occur in these regions once the yield strength is exceeded, redistributing stresses and reducing peak stress levels. Therefore, the reported stresses should be interpreted as indicators of critical stress concentration zones rather than actual service stresses. Nevertheless, the stress levels approach the material's ultimate tensile strength of 869 MPa, indicating a potential risk of localized yielding and fatigue crack initiation in the pivot pin hole and pulling face regions under repeated service loading.

These results correspond to failure-prone locations identified in prior studies [1,6,20,26]. The pivot pin area was identified as the most structurally sensitive section due to geometric discontinuities and localized force concentration.

7.2. Assembly-Level Stress Analysis

The assembly-level analysis indicated a global average stress of approximately 207.9 MPa on the knuckle. Highly localized stress concentrations were observed at sharp geometric corners, as illustrated in Figure 16. These peaks are attributed to numerical stress singularities caused by sharp edges and contact discretization in the finite element model. Such singular stresses do not represent physically meaningful values in real structures and were therefore excluded from quantitative evaluation. Similar observations have been reported by Pugazhenthi et al. [28], where singular stresses were neglected in favor of representative stress values for engineering interpretation. Accordingly, subsequent discussions focus on representative stress levels obtained away from singular regions, which were approximately 950 MPa in the critical load-transfer regions of the knuckle.

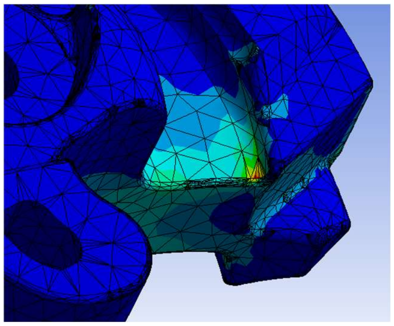


Figure 16. Numerical stress singularity observed in the assembly-level finite element model, illustrating localized stress concentration at sharp geometric edges.

To obtain a more realistic representation of structural behaviour, an assembly-level model was developed by integrating the knuckle with the coupler body and defining frictional contact interactions at critical interfaces. These interfaces include the pivot pin hole, pulling lug surfaces, and the contact region between the knuckle and the coupler body.

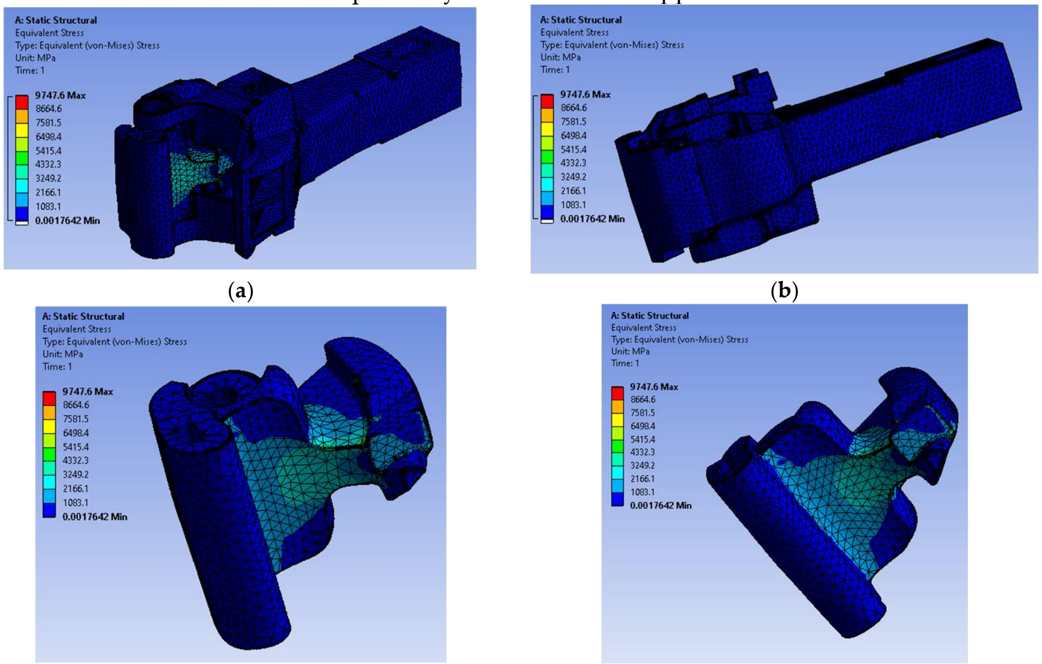
The simulation results demonstrate that the presence of contact interactions redistributes the applied load across the assembly. As a result, the peak von Mises stress in the knuckle decreases compared with the component-level model. The representative stress in the assembly-level model was approximately 950 MPa in the critical regions around the pivot pin hole and pulling face.

Although highly localized stress peaks appeared near sharp contact edges, these peaks are attributed to numerical stress singularities associated with idealized geometric features and contact discretization. Such singular stresses do not represent physically meaningful values in real structures and were therefore excluded from the engineering interpretation.

Overall, the assembly model demonstrates that the load is transferred through multiple contact interfaces within the coupler system, which reduces the concentration of stress within the knuckle component.

These representative stress levels are consistent with the peak stress range reported in the abstract and conclusions, confirming the internal consistency of the numerical results.

Figure 17 illustrates the stress distribution in the assembly-level model, where the interaction between the knuckle and the coupler body redistributes the applied load across the contact interfaces.



(**a**) (**b**)

(**c**) (**d**)

Figure 17. Assembly-level von Mises stress distribution of the knuckle–coupler system subjected to a tensile load of 650,000 lbs (2670 kN): (a) front view, (b) rear view of the assembly, (c) stress distribution on the knuckle component, and (d) representative stress levels around the S-surface and pulling face regions.

7.3. Comparative Assessment

Component-level analysis provides detailed insight into localized stress behavior and is well suited for evaluating geometric modifications. However, it neglects contact forces and system-level constraints. In contrast, assembly-level analysis captures load redistribution, frictional effects, and interaction between components, resulting in a more realistic stress field and slightly reduced peak stresses.

Table 7. Comparative Assessment.

Analysis Approach	Representative Stress (MPa)	Critical Region	Engineering Implication
Component-Level (Knuckle)	~ 1050	Pivot pin hole, pulling face	High localized stress; increased risk of localized yielding and fatigue initiation
Assembly-Level (Knuckle)	~950	the pulling lug interface, the S surface curve and curved pulling face regions	Stress redistribution due to contact interactions; more realistic load transmission
Stress Reduction	~10		Assembly interaction distributes load across the coupler system

These findings are consistent with the framework proposed by Adams and Askenazi [29] and the observations of Yadav and Vyas [11], who emphasized the importance of assembly-level analysis for discontinuous systems such as AAR couplers. The results also align with Kevin and Kari [5], who demonstrated that contact configuration strongly influences stress levels in the pivot pin and pulling lug regions.

Overall, the two modeling approaches are complementary: component-level analysis supports geometric optimization, while assembly-level analysis provides a system-level assessment of structural performance under operational conditions.

7.4. Comparative Discussion

A comparison between the two modeling approaches highlights the importance of considering assembly interactions in structural analysis. The component-level model predicted peak stresses of approximately 1050 MPa, while the assembly-level model reduced the peak stress to approximately 950 MPa. This represents a reduction of roughly 10% in peak stress due to load redistribution within the coupler–knuckle system.

The stress reduction occurs because the assembly model captures realistic mechanical interactions between the knuckle and the coupler body. Contact forces at the pivot pin, pulling lug, and backstop interfaces distribute the applied tensile load over a larger structural region. Consequently, the load is no longer concentrated solely within the knuckle geometry.

These findings demonstrate that assembly-level finite element analysis provides a more accurate representation of in-service structural behavior than isolated component analysis. While component-level analysis is valuable for identifying localized stress concentrations and guiding geometric optimization, assembly-level modeling is essential for evaluating overall system performance and load transfer mechanisms.

The results therefore confirm that incorporating realistic contact interactions between coupler components significantly improves the reliability of stress prediction and provides a more robust basis for design evaluation and structural optimization of railway coupler systems.

8. Conclusion

This study investigated the structural response of a modified AAR Type E knuckle coupler using component-level and assembly-level finite element analyses under a tensile load of 650,000 lbs (2670 kN). Both approaches consistently identified the pivot pin hole and pulling lug regions as critical stress locations, with representative stress levels ranging from 950 to 1050 MPa.

These stress levels approach the material's ultimate tensile strength, indicating a potential risk of localized yielding and fatigue damage under cyclic service loading. While geometric reinforcement improves stress distribution, it may not fully eliminate fatigue risk in critical regions.

Key implications for design include:

- Targeted geometric reinforcement near the pivot pin and fillet regions;
- Use of higher-toughness or fatigue-resistant materials;
- Experimental validation through fatigue testing under realistic loading conditions.

The combined use of component- and assembly-level modeling provides a robust framework for evaluating coupler performance and guiding future design improvements. Although the present study considered static loading only, the results highlight the importance of detailed assembly-level modeling for improving the safety, reliability, and durability of heavy-haul railway coupler systems.

9. Future Works

Future research will extend the present study to include transient and fatigue life analyses, as well as comprehensive experimental validation through full-scale testing. Although full-scale experiments are beyond the scope of the current work, the material properties employed in the numerical simulations were obtained from tensile testing conducted in accordance with ASTM E8/E8M. To further validate the numerical predictions, future studies will incorporate full-scale draft-load and fatigue tests under realistic service conditions. In addition, strain-gauge measurements on a scaled prototype will be performed to verify stress distributions and to corroborate the finite element results.

Subsequent investigations will also address nonlinear material behavior, including plastic deformation, fatigue damage accumulation, and fracture mechanics, in order to overcome the limitations associated with the linear elastic assumption adopted in the present analysis. Alternative geometric configurations and different material grades, including variations of Grade E steel, will be examined to evaluate their influence on structural performance and durability. Furthermore, practical assembly-level factors such as connection looseness, manufacturing tolerances, and wear effects will be considered to better replicate real-world operating conditions and to enhance the applicability and robustness of the proposed knuckle–coupler design framework.

Author Contributions: Conceptualization, methodology, Nutthapong Kunla and Anan Suebsomran; validation, formal analysis, Anan Suebsomran, investigation, Nutthapong Kunla and Anan Suebsomran; resource Nutthapong Kunla and Anan Suebsomran; data curation, Nutthapong Kunla; writing– original draft preparation, Nutthapong Kunla; writing– review and editing, Anan Suebsomran; visualization, Nutthapong

Kunla and Anan Suebsomran; supervision, Anan Suebsomran; project administration, Anan Suebsomran. All authors have read and agreed to the published version of the manuscript.

Funding: This work was funded by Thailand Science Research and Innovation (TSRI), Thailand, through the Program Management Unit for Competitiveness (PMU C), contract number C10F660055.

Institutional Review Board Statement: Not applicable.

Data Availability Statement: Data is provided within the manuscript or supplementary information files.

Acknowledgments: The authors would like to express their sincere gratitude to Italian–Thai Development Public Company Limited for kindly providing the material data on Carbon Steel Grade E, which was essential for the successful completion of this study.

Conflicts of Interest: The authors have no conflicts to disclose.:.

References

1. Chunduru, S.P., M.J. Kim, and C. Mirman, *Failure analysis of railroad couplers of AAR type E.* Engineering Failure Analysis, 2011. **18**(1): p. 374-385.
2. Song, Z., et al. *The Research on Coupler Knuckle of High Speed and Heavy Haul*. in *2009 International Conference on Measuring Technology and Mechatronics Automation*. 2009.
3. Qing, L., *Prediction for fatigue crack lifetime ofthe endflanged corner of coupler yoke.* Journal of the China Railway Society, 2002.
4. Steed, J. and R. Kimpton, *Improving the 10" Head Alliance Coupler and Knuckle.* no. June, 2015.
5. Gonzales, K., et al. *Effects of heavy axle loads on car coupling systems*. in *10th International Heavy Haul Association Conference (IHHA2010). February*. 2013.
6. Wang, G.X., et al., *The Finite Element Analysis of New Forged Coupler Knuckle.* Applied Mechanics and Materials, 2013. **367**: p. 122-125.
7. Li, C. and S. Sun. *Crack growth life estimation for No. 17 coupler knuckle in China Datong-Qinhuangdao railway freight trains*. in *IOP Conference Series: Materials Science and Engineering*. 2020. IOP Publishing.
8. Pipes, L.A., *Analysis of longitudinal motions of trains by the electrical analog.* Journal of Applied Physics, 1942. **13**(12): p. 780-786.
9. Davis, J.H. and B.M. Barry, *DISTRIBUTED MODEL FOR STRESS CONTROL IN MULTIPLE LOCOMOTIVE TRAINS.* Applied Mathematics and Optimization, 1977. **3**(2-3).
10. Wu, Q., M. Spiryagin, and C. Cole, *Longitudinal train dynamics: an overview.* Vehicle System Dynamics, 2016. **54**(12): p. 1688-1714.
11. Yadav, O.P. and N.S. Vyas, *The influence of AAR coupler features on estimation of in-train forces.* Railway Engineering Science, 2023. **31**(3): p. 233-251.
12. Yao, Y.P., et al., *The stability mechanism and its application to heavy-haul couplers with arc surface contact.* Vehicle System Dynamics, 2013. **51**: p. 1324 - 1341.
13. Zhang, H., et al., *Research on Simulation Calculation of the Safety of Tight-Lock Coupler Curve Coupling.* Symmetry, 2021. **13**(11): p. 1997.
14. Cascino, A., et al., *Development and Experimental Testing of a 3D Vision System for Railway Freight Wagon Monitoring.* Applied Sciences, 2025. **15**(21): p. 11547.
15. Kınacı, B.F., et al., *Investigation of fatigue life of draw hook equipment used in freight wagon: Miscellaneous result.* Engineering Failure Analysis, 2024. **155**: p. 107757.
16. Bozkurt, I. and Z. Pala, *Prediction of impact behavior of notched composite pipes repaired with different composite patches by machine learning algorithms.* Journal of Adhesion Science and Technology, 2025. **39**(10): p. 1521-1558.
17. Bozkurt, I., M.O. Kaman, and M. Albayrak, *Low-velocity impact behaviours of sandwiches manufactured from fully carbon fiber composite for different cell types and compression behaviours for different core types.* Materials Testing, 2023. **65**(9): p. 1349-1372.
18. Bozkurt, I., *Effect of geometric configurations and curvature angle of corrugated sandwich structures on impact behavior.* Polymer Composites, 2025. **46**(2): p. 1662-1685.

19. Bozkurt, I., *Effect of rubber interlayer and curvature angle on impact performance of honeycomb sandwich composite structures.* Mechanics of Advanced Materials and Structures: p. 1-26.
20. Bozkurt, I., *Effect of curvature angle and structural configurations on dynamic performance of honeycomb sandwich structures.* Mechanics of Advanced Materials and Structures: p. 1-22.
21. Lukoševičius, V., et al., *Damage Analysis of Rolling Stock Automatic Coupler under Cyclic Loads.* Applied Sciences, 2024. **14**(19): p. 9025.
22. Giannella, V., et al., *Numerical investigation on the fracture failure of a railway axle.* Engineering Failure Analysis, 2021. **129**: p. 105680.
23. Bremmer, F.H., Lou, Y., Pepin, K., Sahr, A.M., & Tumberg, J.D., *Rail Car Coupler*, in *National University Rail Center*. 2012, Michigan Technological University.
24. *AAR Manual of Standards and Recommended Practices*, in *Knuckles, Types E and F - Fatigue Test, Standard M 216*. 2009.
25. *AAR Manual of Standards and Recommended Practices*, in *Foundry and Product Approval Requirements for the Manufacture of Couplers, Coupler Yokes, Knuckles, Follower Blocks and Coupler Parts, Standard M-211*. 2010.
26. *AAR Manual of Standards and Recommended Practices* in *Castings steel,Standard M-201*. 2005.
27. Ghosh, R., et al., *Static analysis of multi-leaf spring using ANSYS workbench 16.0.* Int J Mech Eng Technol (IJMET), 2016. **7**(5): p. 241-249.
28. Pugazhenthi, R., et al., *Optimization of fatigue life and fractography analysis of knuckle joint.* Materials Today: Proceedings, 2021. **46**: p. 4344-4348.
29. Adams, V. and A. Askenazi, *Building better products with finite element analysis.* (No Title), 1999.