Journal of Metallurgical Engineering and Processing Technology, Vol. 5, No. 1
August, 2024
P-ISSN: 2723-6854, E-ISSN: 2798-1037, page 24-41
DOI: https://doi.org/10.31315/jmept

Behavior of welded joints on the roof truss of KOJK Office using LISA V.8 FEA

Aco Wahyudi Efendi[1]
[1]Universitas Sebelas Maret, Surakarta, Indonesia
*Corresponding author : aw.efendi2018@gmail.com

Received 14/03/2024; Revised 25/07/2024; Published 05/08/2024

Abstract

Welded joints play an important role in the integrity of steel structures in buildings, especially in supporting heavy loads and resisting various forces such as wind, seismic and gravity loads. Several welds were identified that did not meet the standards, so it was necessary to analyze several welding elements in the connection of the steel roof truss structure. This research was conducted by identifying the stresses that occur in the welded joints of the roof truss of the KOJK Central Java building, with an initial analysis of rigid and rigid connection conditions, and a second condition with conditions that occur according to findings in the field. The second condition is the condition that occurs according to the findings in the field, where the connection is not standardized and there are several holes due to imperfect welding. The modeling analysis uses Finite Element Analisys software, namely LISA V.8 FEA (License), to obtain the stress behavior that occurs in the KOJK Central Java roof truss elements. From the results of analysis and modeling by making two conditions, namely the condition of the welding joint according to the standard and the welding joint of the field conditions. The stress ratio at the weld joint is 1.351 and the stress ratio at the truss pull bar is 1.054.

Keywords: FEA; LISA; Roof; Stress; Welded

Introduction

Welded joints play a crucial role in the structural integrity of buildings, particularly in supporting heavy loads and resisting various forces such as wind, seismic, and gravitational loads. The Otoritas Jasa Keuangan (OJK) Office building, standing tall and prominent in the urban landscape, represents a significant architectural and engineering marvel. However, ensuring the stability and safety of such structures requires meticulous analysis and understanding of the behavior of critical components like welded joints in the roof truss system.

Despite advancements in structural engineering and computational analysis tools, there exists a research gap concerning the behavior of welded joints specifically in the roof truss system of high-rise buildings like The Otoritas Jasa Keuangan (OJK) Office building. Previous studies have primarily focused on overall structural analysis or specific components without delving deep into the intricacies of welded joint behavior under varying loads and environmental conditions. Additionally, while there have been studies utilizing finite element analysis (FEA) software, such as LISA V.8 FEA (License), their application to welded joints in roof trusses of real-world structures like The Otoritas Jasa Keuangan (OJK) Office building remains limited.

This research was carried out by identifying the stresses that occur in the welded joints of the truss trusses of the roof of the KOJK Central Java building, with initial analysis conditions of rigid and stiff connections, and second conditions with conditions that occur according to field findings, where the joints are non-standard and there are several holes due to welding. not perfect. Modeling analysis uses Finite Element Analisys software, namely LISA V.8 FEA (License), to obtain the stress behavior that occurs in the roof truss elements of KOJK Central Java.

Research Methods

The Otoritas Jasa Keuangan (OJK) Office building's structural specifications, including material qualities, geometric dimensions, and weld details, will be gathered for the study. creating a finite element model with precise weld joint representations for the roof truss system using LISA V.8 FEA (License) software. application of different loading conditions in accordance with applicable design codes and standards, such as dead loads, live loads, wind loads, and seismic loads. Examination of the failure modes, deformation patterns, and stress distribution inside the welded joints under various loading conditions.analysis of the findings and creation of suggestions for improving the construction and design methods for welded joints.

Collection of Structural Data

Detailed structural data and specifications of The Otoritas Jasa Keuangan (OJK) Office building will be collected, including material properties, geometric dimensions, and weld details. This information will serve as the basis for developing an accurate finite element model of the roof truss system.

Finite Element Model Development

A finite element model of the roof truss system will be developed using LISA V.8 FEA (License) software. This model will accurately represent the geometry of the truss members, as well as the welded joints connecting them. The weld details obtained from the structural data will be incorporated into the model to ensure realistic simulation of joint behavior(Ertas and Akbulut 2021).

Material Properties Assignment

Material properties, including the modulus of elasticity, yield strength, and ultimate tensile strength, will be assigned to the structural elements of the roof truss. These properties will be based on the material specifications provided for the construction of The Otoritas Jasa Keuangan (OJK) Office building.

Loading Conditions Application

Various loading conditions, including dead loads, live loads, wind loads, and seismic loads, will be applied to the finite element model. These loads will be in accordance with relevant design codes and standards applicable to The Otoritas Jasa Keuangan (OJK) Office building's location and structural requirements(Bryson and Giraldo 2020; Efendi 2023a; Fu et al. 2022; Jideani and Grabe 2019).

Behavior of Welded Joints in Structural Steel Systems.

Welded joints play a critical role in transmitting loads and ensuring the structural integrity of steel systems. Research studies have investigated the behavior of welded joints in structural steel frames under various loading conditions. These studies have highlighted the importance of understanding weld performance in enhancing the overall stability and safety of steel structures(Li, L. et al. 2019).

Finite Element Analysis of Welded Joints

Finite element analysis (FEA) is a powerful computational tool for simulating the behavior of welded joints in structural systems. Research conducted by Wang et al. (2020) utilized FEA to study the stress distribution and fatigue behavior of welded joints in steel bridges. Such studies provide valuable insights into the factors influencing the performance of welded connections and their implications for structural design and maintenance(A. W. Efendi 2022a; Efendi 2023d; Elsanadedy et al. 2021; Li, L. et al. 2019, 202AD, 202AD, 202AD).

Optimization of Welded Joint Design

Optimization techniques have been employed to enhance the design of welded joints in structural systems. Research by Zhang et al. (2018) focused on the optimization of weld geometry and detailing to improve the fatigue performance of welded connections in steel bridges. These optimization strategies can be adapted to address the specific challenges associated with welded joints in the roof truss of the KOJK building(Zhang, Z, Zhang, Y., and Liu, H. 2018).

Analysis of Welded Joint Behavior

The finite element model will be analyzed to study the behavior of welded joints under different loading scenarios. This analysis will involve examining stress distribution, deformation patterns, and potential failure modes within the welded joints(Efendi 2023).

Structural Analysis Using LISA V.8 FEA (License)

LISA V.8 FEA (License) is a widely used software tool for conducting structural analysis and design. Studies such as that by Kaveh et al. (2017) have demonstrated the effectiveness of LISA V.8 FEA (License) in simulating the behavior of complex structural systems, including bridges and high-rise buildings. By utilizing LISA V.8 FEA (License), this research aims to accurately model and analyze the behavior of welded joints in the roof truss of the KOJK building, providing valuable insights for structural optimization(Kaveh, A. and Talatahari, S. 2017).

LISA is a very popular elemental analysis program used to estimate temperature rise for three different heat exchanger models. Line, shell, and solid element models are the three simplest and easiest types of models to build. For line element models, LISA provides a menu of frequently used element shapes and structural geometries; the user only needs to add the element dimensions in one dialog box and the thermal conductivity in another dialog box. Figure 4 shows the model lines in the LISA pre-processing window after meshing. Due to the limited number of nodes provided, the line modeling is simple and straightforward, which makes the analysis processing easier(Efendi 2020; A. W. Efendi 2022b; I. A. W. Efendi 2022; Efendi 2023d, 2023b, 2023a, 2023c; Efendi and Safitri n.d.; Efendi and Weijia 2023).

Consideration of Building Codes and Standards

Structural analysis and design must adhere to relevant building codes and standards to ensure the safety and integrity of buildings. Research by ASTM has provided guidelines for the design and evaluation of welded connections in steel structures, emphasizing the importance of compliance with established standards. The application of these codes and standards will inform the analysis and design recommendations for welded joints in the KOJK building's roof truss(ASTM International 2016).

Results of Research and Discussion

This study was conducted to assess the feasibility of the results of welds performed on steel truss elements in the framework of roof retrofitting at the Central Java Otoritas Jasa Keuangan (OJK) Office Building located in Semarang. The research location is at jalan Kyai Saleh No.12 - 14, Mugassari, Kec. Semarang Sel., Kota Semarang, Jawa Tengah 50243.

Figure 1. Research location.
Source: AW Efendi, 2024

The analysis is based on asbuilt drawing data and the results of visual and detailed inspection at the research site. The areas that were strengthened and tested are shown in Figure 2.

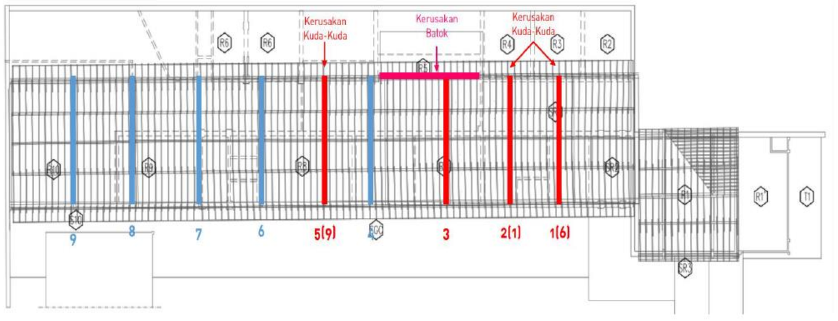


Figure 2. Layout Structural damage treatment of building C.
Source: KOJK document data, 2023

The material and size of the steel frame structure of the building truss used in the field are the parameters in inputting data in the modeling. as in table 1.

Table 1 Material data used

No		**Modulus Elastisitas** (N/mm^2)	**Poison Rasio**	**Specific Gravity** (N/mm^2)
1	Steel	210,000	0.3	0.000078500
2	Concrete fc' 20 MPa	21,019	0.2	0.000024000
3	Red Brick	1,633.5	0.15	0.000020000

The material parameters in table one are input into LISA V.8 FEA (License) as shown in Figure 3.

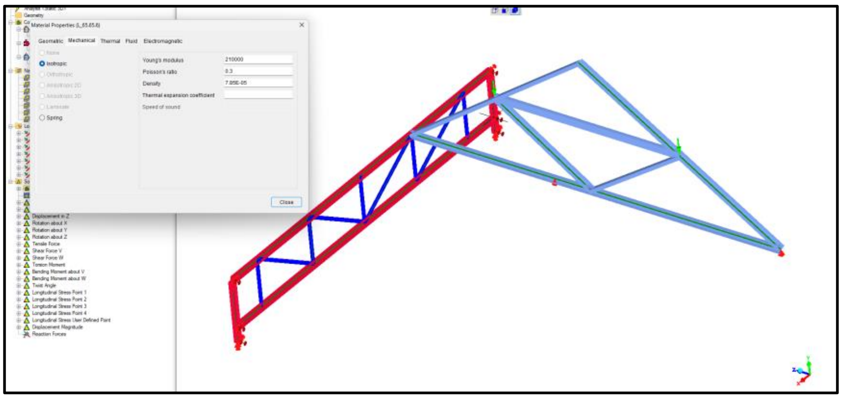

Material Properties (L_65.65.6)
Isotropic
Young's modulus: 210000
Poisson's ratio: 0.3
Density: 7.85E-05
Thermal expansion coefficient:

Figure 3. material properties
Source: LISA V.8 FEA (License), 2024

The size and dimensions of the steel truss frame adjust the existing conditions based on field measurements and asbuilt drawing data as shown in Figure 4 and the geometric model of the truss is also reviewed based on both data, shown in Figure 5.

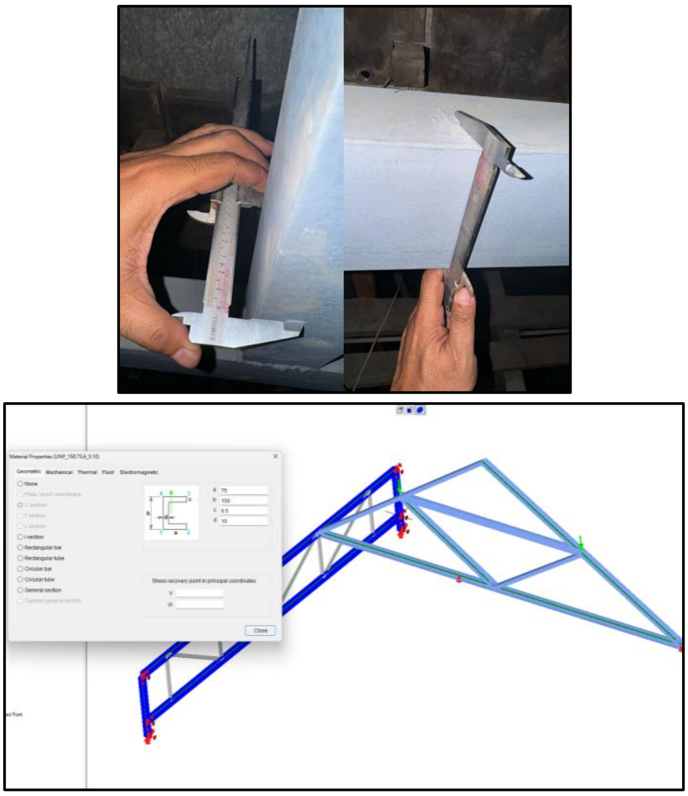

Material Properties (UNP_150.75.6_5.10)
C section
a: 75
b: 150
c: 6.5
d: 10

Figure 4. The size and dimensions of the steel truss frame adjust the existing conditions
Source: AW Efendi, 2024

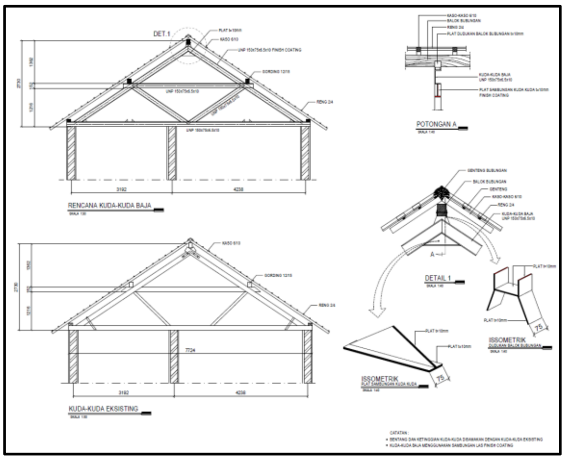


Figure 5. Asbuilt drawing.
Source: KOJK document data, 2023

3D (Existing) Modeling of R5 Beams and Strengthening Truss, structural modeling based on the results of visual inspection and asbuiltdrawing document information by adjusting the size based on document information and field implementation results is shown in Figure 6.

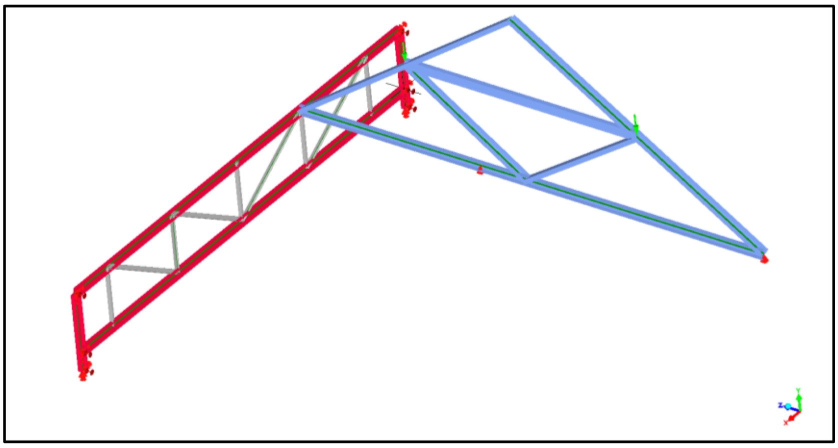


Figure 6. 3D modeling of the R5 strengthening frame and the additional truss
Source: LISA V.8 FEA (License), 2024

The material load is already inputted in the LISA V.8 FEA (License) software, but additional loads are required such as Super Dead Roof Load, adjusting to the conditions that have been installed at the research site.

The wind load of this plan uses the SKBI-1.3.5.3-1987 guidelines where the minimum blowing pressure is 25kg/m^2. In planning the house uses a triangular roof with a slope angle < 65^o, so the wind coefficient used is - 0.4.

- Then 0.4 x 25 x 6 x 4.5
  = 270 kg
  = 2700 N
- Ceiling and hanging load

= 0.2 kN/m^2 x 6 m x 4.5 m
= 5.4 kN
= 5400 N

- ME installation weight
  = 0.25 kN/m^2 x 6 m x 4.5 m
  = 6 kN/m
  = 6000 N

Roof Cover Load (Roof Tile)

- Distance between gording x weight of roof tile
  = 1.305 m x 50 kg/m^2
  = 65.25 kg/m
- Self Weight of Roofing + 10% weight of Roofing (Qx)
  = 8.32 kg/m + (10% x 8.32 kg/m)
  = 9.152 kg/m
- Total Dead Load (Qt) = Roof covering load (Roof Tile) + (Own weight of gording + 10% weight of gording)
  = 65.25 kg/m + 9.152 kg/m
  = 74.402 Kg/m x 4.5 m
  = 334.81 kg = 3348.1 N
- The total dead load is
  = 5400 N + 6000 N
  = 11,400 N + 2700 N + 3348.1 N
  = 17,448.1 N.

The total of the Work load is entered in the Work load parameters of the LISA V.8 FEA (License), shown in figure 7.

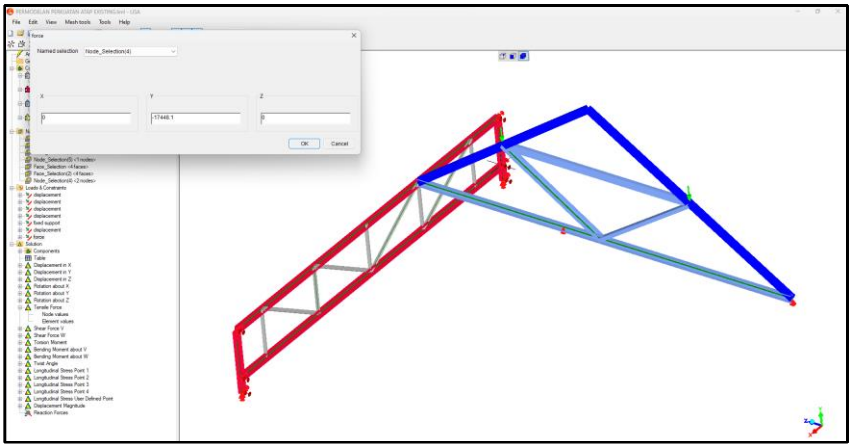

force
Named selection: Node_Selection(4)
X: 0
Y: -17448.1
Z: 0

Figure 7. Work load parameters
Source: LISA V.8 FEA (License), 2024

From the results of the load acting under service conditions, a deflection of 0.57 mm occurred in the truss press bar area, shown in Figure 8.

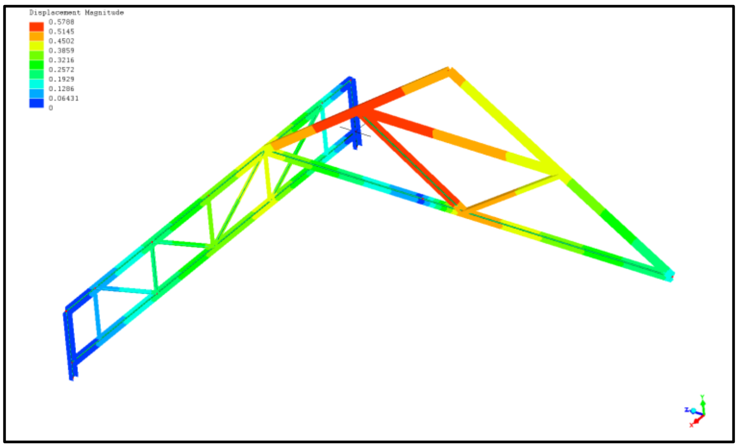


Figure 7. Deflection
Source: LISA V.8 FEA (License), 2024

In this modeling, there is a tensile force of 9886 N on the truss pull rod with a red area, shown in Figure 8, while the shear force that occurs is 1212 N on the truss pull rod, shown in Figure 9.

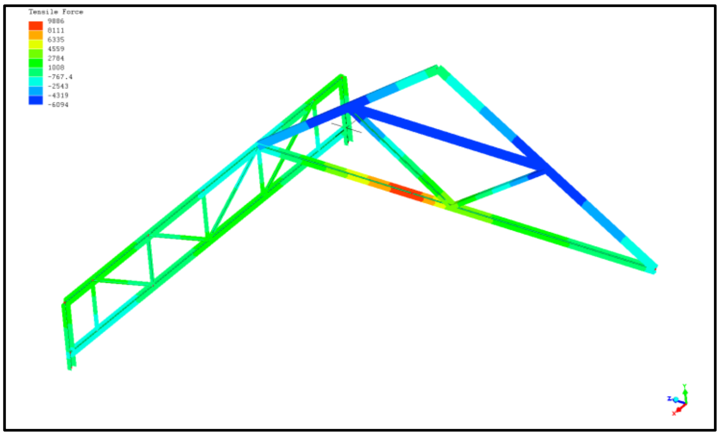


Figure 8. Tensile force
Source: LISA V.8 FEA (License), 2024

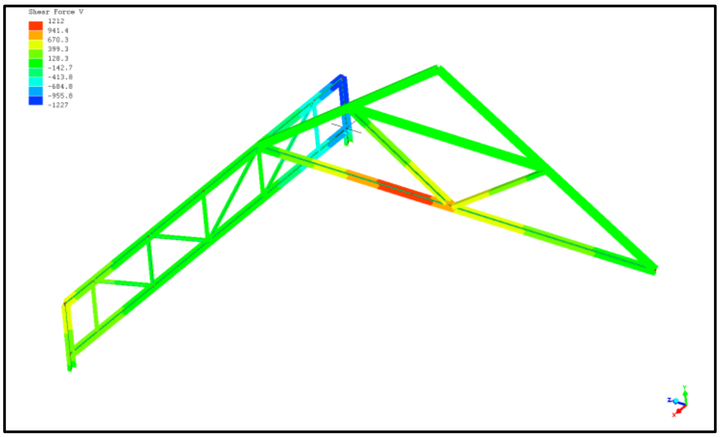


Figure 9. Shear force
Source: LISA V.8 FEA (license), 2024

From the results of the modeling analysis above, the maximum stress that occurs is 16.97 N/mm^2 on the R5 frame and truss, shown in Figure 10. With BJ 37 steel quality, the minimum yield stress is 240 N/mm^2, when referring to Allowable Stress Design (ASD) if FS 1.69 is taken for tensile bars or beams, the allowable stress is 142.011 N/mm^2, with the stress that occurs in the maximum roof truss is 16.97 N/mm^2 < 142.011 N/mm^2, the planning conditions are still within the permissible / safe limits.

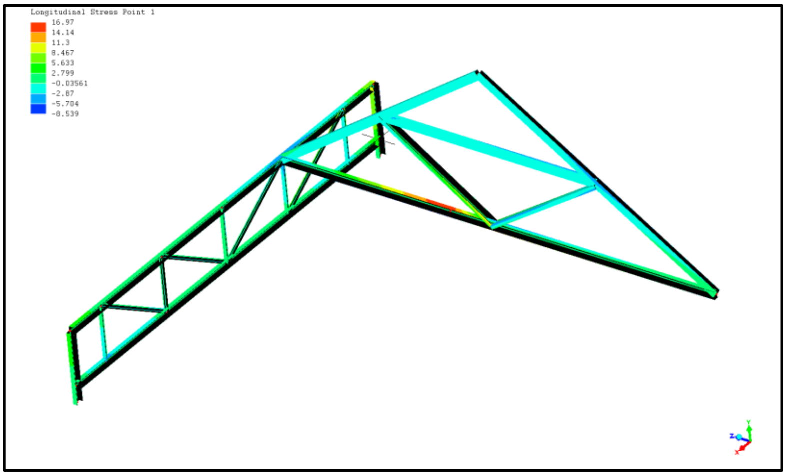


Figure 10. Stress that occurs
Source: LISA V.8 FEA (License), 2024

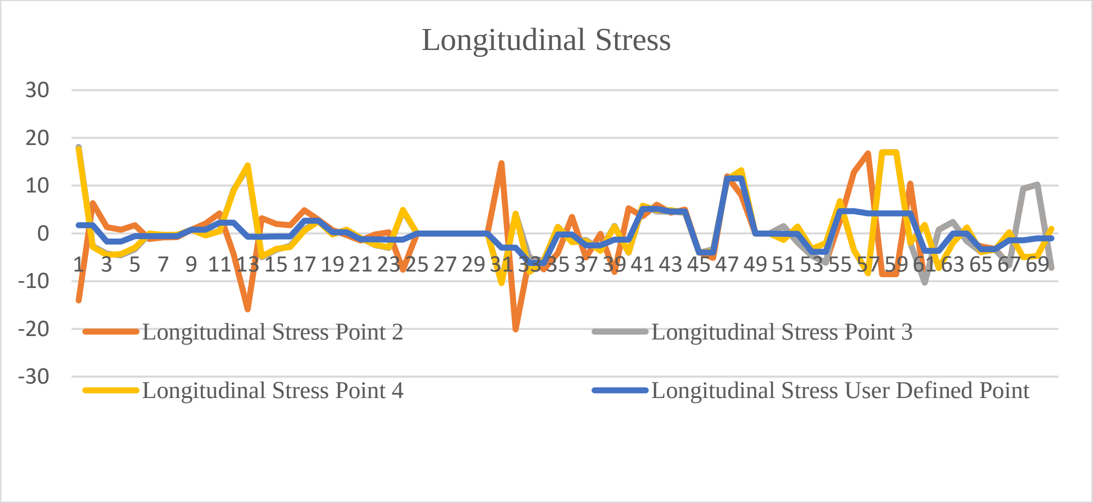


Figure 11. Graphs of the various stress conditions that occur

Nominal stresses that occur in several stress conditions according to the graph above, Longitudinal Stress Point 2 conditions with stress values of 0.262 N/mm^2 to 16.769 N/mm^2, at Longitudinal Stress Point 3 conditions 0.520 N/mm^2 to 18.118 N/mm^2, at Longitudinal Stress Point 4 conditions 0.237 N/mm^2 to 17.650 N/mm^2, and at Longitudinal Stress User Defined Point conditions 0.210 N/mm^2 to 11.532 N/mm^2.

According to the results of the visual inspection, it was found that some welding operations on the easel were identified as not in accordance with the requirements shown in Figure 12. where the standard conditions are in accordance with Figure 13. from this condition, the stress analysis that occurs when the conditions according to Figure 12 are analyzed using microanalysis LISA V.8 FEA (License).

Figure 12. One type of welding found in the field
Source: KOJK field findings, 2024

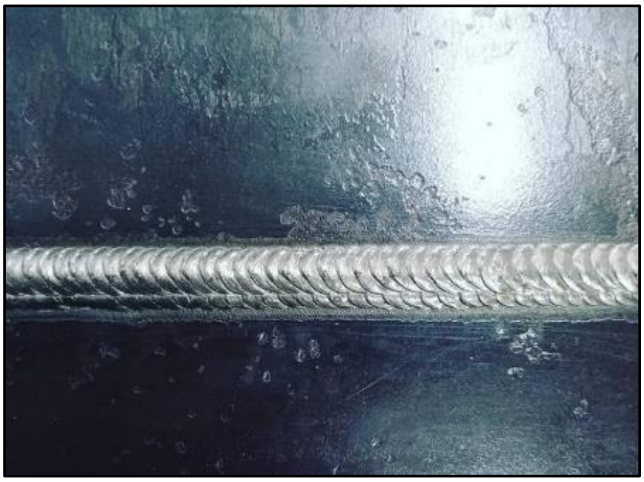


Figure 13. Welding according to standards
Source: Google, 2024

From the above two conditions, an analysis simulation of the two conditions is carried out, the first by analyzing the conditions according to standard welding where the connection between compressive and tensile rods is a perfect rigid condition, in accordance with Figure 13.

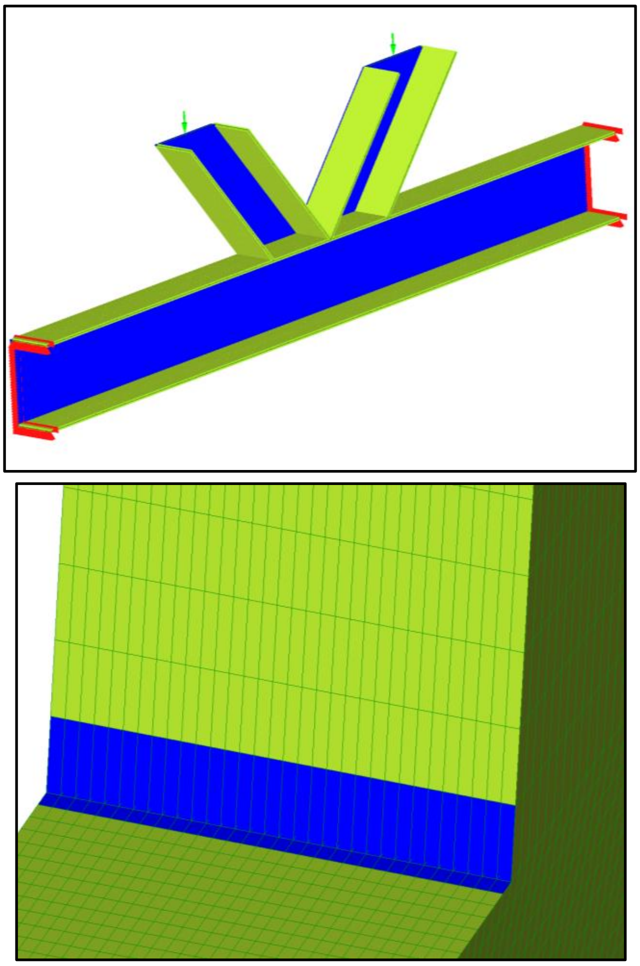
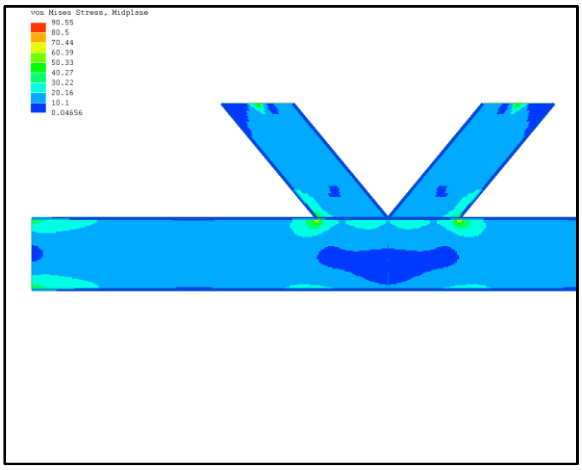


Figure 14. Shear force
Source: LISA V.8 FEA (license), 2024



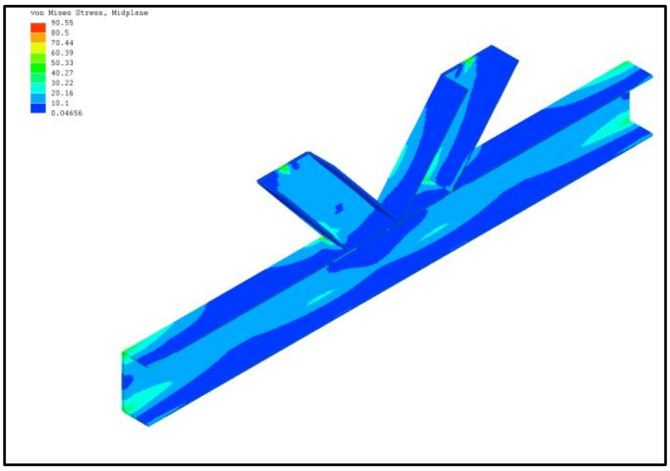
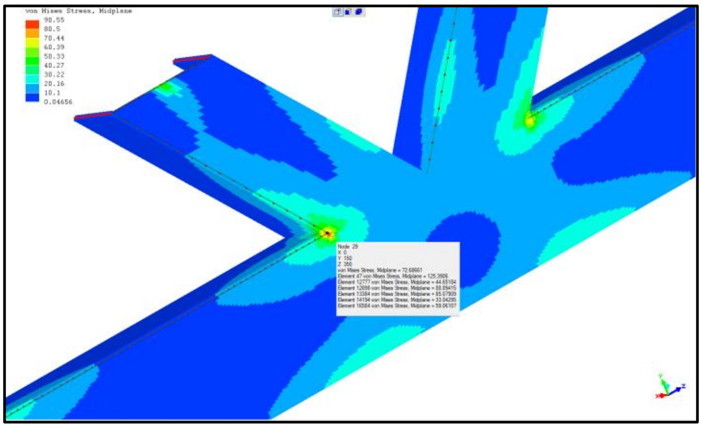
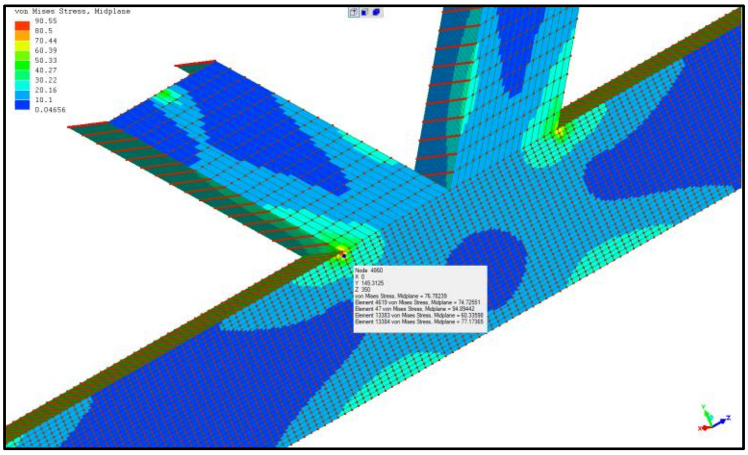


Figure 15. Stresses
Source: LISA V.8 FEA (license), 2024



```
Node 29
X 0
Y 150
Z 350
von Mises Stress, Midplane = 72.68961
Element 47 von Mises Stress, Midplane = 125.3906
Element 12777 von Mises Stress, Midplane = 44.65184
Element 12888 von Mises Stress, Midplane = 88.89415
Element 13384 von Mises Stress, Midplane = 85.07909
Element 14194 von Mises Stress, Midplane = 33.04295
Element 16584 von Mises Stress, Midplane = 59.06107
```

Figure 16. Stresses that occur at the intersection of the truss's compressive and tensile bars
Source: LISA V.8 FEA (license), 2024



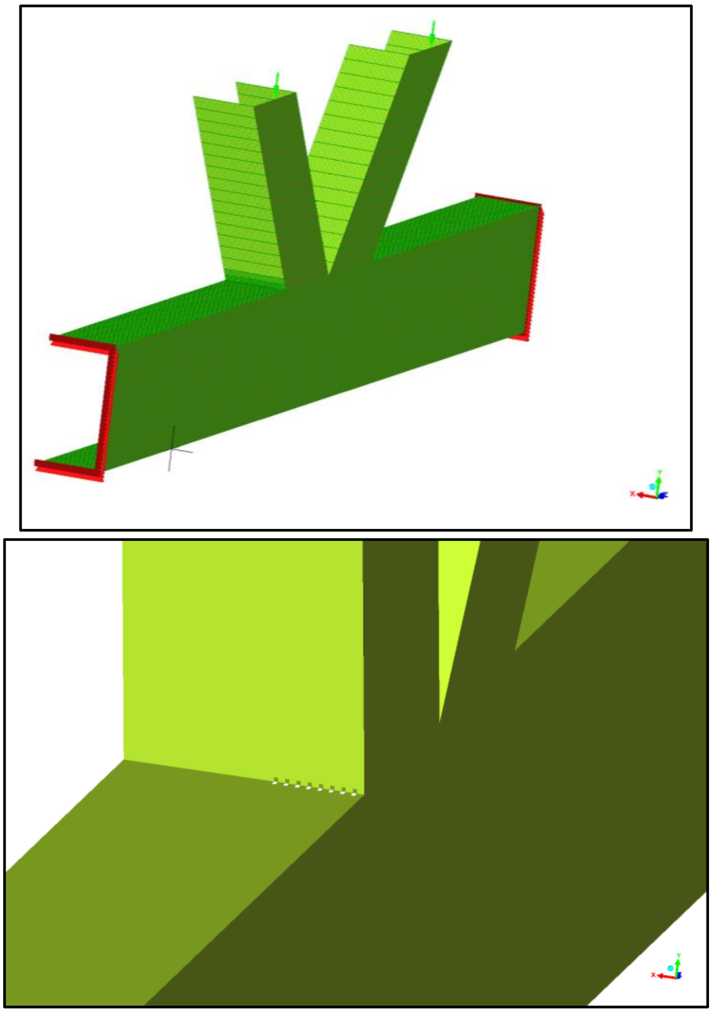
```
Node 4860
X 0
Y 145.3125
Z 350
von Mises Stress, Midplane = 76.78239
Element 4619 von Mises Stress, Midplane = 74.72551
Element 47 von Mises Stress, Midplane = 94.89442
Element 13383 von Mises Stress, Midplane = 60.33598
Element 13384 von Mises Stress, Midplane = 77.17365
```

Figure 17. Stresses that occur in the truss tensile bars
Source: LISA V.8 FEA (license), 2024

The stress that occurs at the intersection of compressive and tensile bars with welding conditions according to the clause and rigid conditions is 72.69 N/mm^2, shown in Figure 16. And the stress that occurs in the tensile bar is 74.72 /mm^2 shown in Figure 17. BJ 37 The minimum yield stress is 240 /mm^2 when referring to Allowable Stress Design (ASD) if FS 1.69 is taken for tensile bars or beams, the allowable stress is 142.011 N/mm^2, with the stress that occurs in the maximum roof truss is 72.69 N/mm^2 < 142.011 N/mm^2, the planning conditions are still within the allowable / safe limits.

Modeling according to Field Conditions

The second modeling is in the welding conditions found in the field according to Figure 12. Where in the FEM modeling several hole points will be made in the connection as shown in Figure 18.



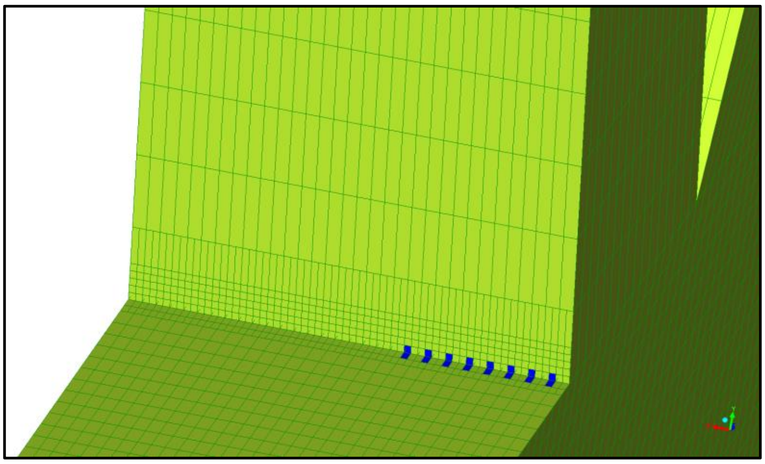


Figure 18. Modeling according to Field Conditions
Source: LISA V.8 FEA (license), 2024

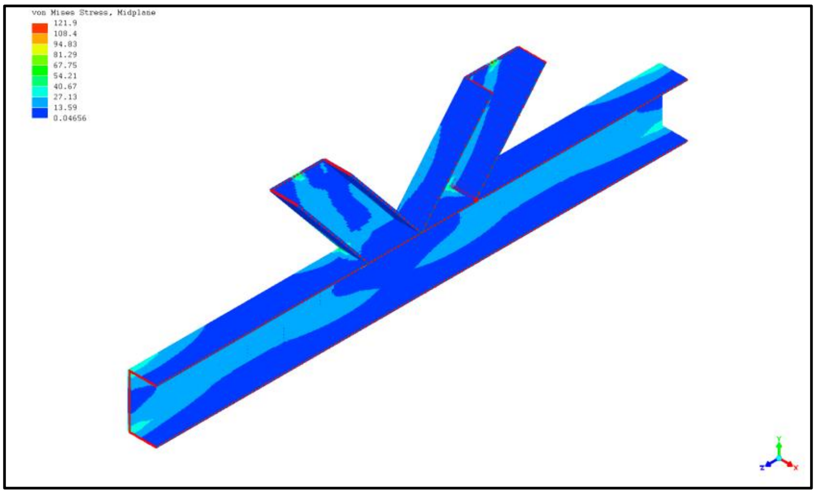
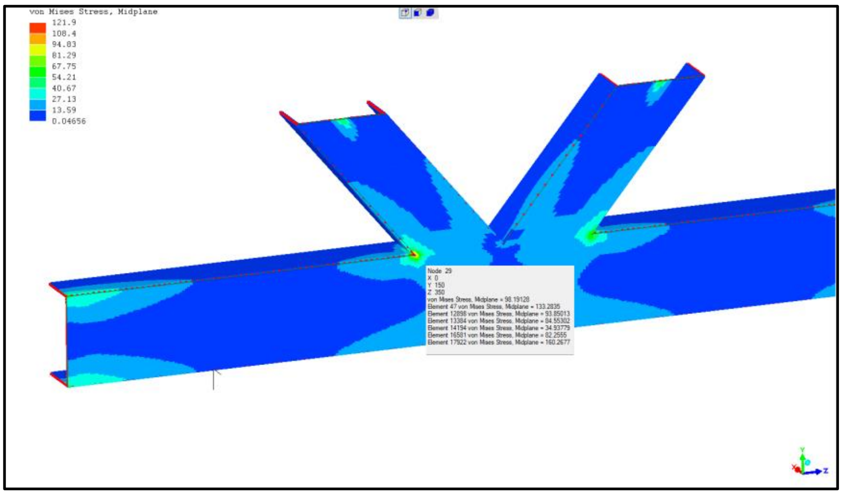


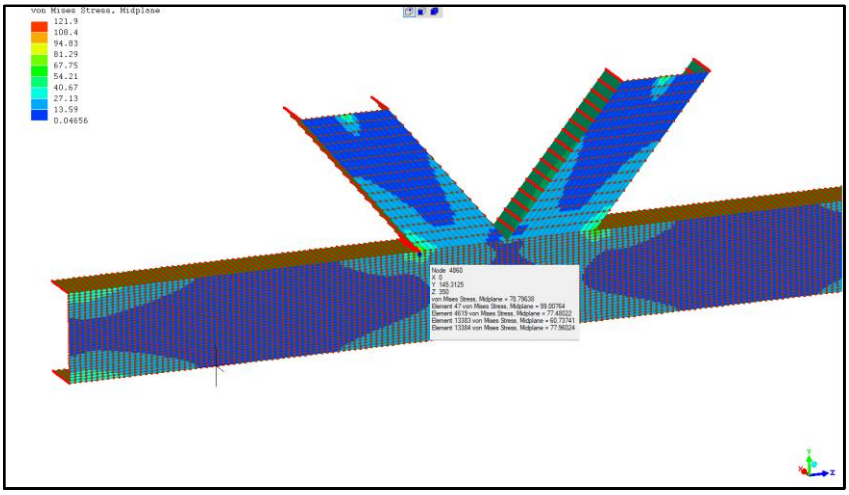
Figure 19. Stresses
Source: LISA V.8 FEA (license), 2024

While the stress that occurs after weakening some of the connection elements that are modeled, the nominal stress that occurs is 121.9 N/mm^2. BJ 37 The minimum yield stress is 240 N/mm^2 when referring to Allowable Stress Design (ASD) if FS 1.69 is taken for tensile bars or beams, the allowable stress is 142.011 N/mm^2, with the stress occurring in the maximum roof truss being 121.9 N/mm^2 < 142.011 N/mm^2, the planning condition is still within the allowable/safe limit.



```
Node 29
X 0
Y 150
Z 350
von Mises Stress, Midplane = 98.19128
Element 47 von Mises Stress, Midplane = 133.2835
Element 12898 von Mises Stress, Midplane = 93.85013
Element 13384 von Mises Stress, Midplane = 84.55302
Element 14194 von Mises Stress, Midplane = 34.93779
Element 16581 von Mises Stress, Midplane = 82.2555
Element 17922 von Mises Stress, Midplane = 160.2677
```

Figure 20. Stresses that occur at the intersection of the truss's compressive and tensile bars
Source: LISA V.8 FEA (license), 2024



```
Node 4860
X 0
Y 145.3125
Z 350
von Mises Stress, Midplane = 78.79638
Element 47 von Mises Stress, Midplane = 99.00764
Element 4619 von Mises Stress, Midplane = 77.48022
Element 13383 von Mises Stress, Midplane = 60.73741
Element 13384 von Mises Stress, Midplane = 77.96024
```

Figure 21. Stresses that occur in the truss tensile bars
Source: LISA V.8 FEA (license), 2024

The stress that occurs at the intersection of compressive and tensile bars with welding conditions according to the clause and rigid conditions is 98.19 N/mm^2, shown in Figure 20. And the stress that occurs in the tensile bar is 78.79 N/mm^2 shown in Figure 21. BJ 37 The minimum yield stress is 240 N/mm2 when referring to Allowable Stress Design (ASD) if FS 1.69 is taken for tensile bars or beams, the allowable stress is 142.011 N/mm^2, with the stress that occurs in the maximum roof truss is 98.19 N/mm^2 < 142.011 N/mm^2, the planning conditions are still within the allowable / safe limits.

Conclusion

From the results of analysis and modeling by making two conditions, namely the condition of welding joints according to standards and welding joints under field conditions. The stress ratio at the weld joint is 1.351 and the stress ratio at the truss pull rod is 1.054.

References

ASTM International. 2016. *ASTM AISC 360/AISC N690 – Specification for Structural Steel Buildings. Retrieved From*. ASTM International.

Bryson, L. S., and J. R. Giraldo. 2020. "Analysis of Case Study Presenting Ground Anchor Load-Transfer Response in Shale Stratum." *Canadian Geotechnical Journal*. doi: 10.1139/cgj-2018-0326.

Efendi, A. W. 2020. "Modeling of Soil Subsidence in IKN Using Numerical Analysis of the Finite Element Method LISA V. 8."

Efendi, A. W. 2022a. "Behavior Analysis of Building Structures After a Fire with FEA LISA V. 8." *Kurva S: Jurnal Keilmuan Dan Aplikasi Teknik ….*

Efendi, A. W. 2022b. "Structural Design Tuak River Pedestrian Suspension Bridge Anchor Block Type Rigid Symmetric with LISA." *Elektriese: Jurnal Sains Dan Teknologi Elektro*.

Efendi, A. W. 2023a. "Behavior of Railroad Bridge Girders Due to Brake Loads with LISA V. 8 FEA." *… Conference on Railway and Transportation (ICORT ….*

Efendi, A. W. 2023b. "Desain Kandang Sperma Ideal Untuk Sapi Menggunakan LISA V. 8 FEA." *Jurnal Sintesis: Penelitian Sains, Terapan Dan ….*

Efendi, A. W. 2023c. "PEMODELAN PENURUNAN TANAH DI IBU KOTA NEGARA NUSANTARA MENGGUNAKAN ANALISIS NUMERIK METODE ELEMEN HINGGA LISA V. 8." *PADURAKSA: Jurnal Teknik Sipil ….*

Efendi, A. W. 2023d. "Simulation of Fire Exposure Behavior to Building Structural Elements Using LISA FEA V. 8." *International Journal of Advanced Science and ….*

Efendi, A. W., and C. Weijia. 2023. "Buffer Stops Behavior Due to Rail Impact Loads with LISA FEA." *Journal of Railway Transportation and Technology*.

Efendi, Aco Wahyudi. 2023. "Stress Behavior of Rampdoor Jetty Bumi Harapan Port Sepaku-IKN Using LISA FEA V.8." *Jurnal Rekayasa Mesin Dan Inovasi Teknologi(Jurnal REMIT)* 4(2):274–79.

Efendi, Aco Wahyudi, and Novia Safitri. n.d. "Numerical Modeling the Effect of Extreme Temperatures on Rooms with Mortar Foam Walls Using LISA V.8 FEA." *CAFEO 41 Engineering Conference.* 41:28–36.

Efendi, I. A. W. 2022. "Repair Analysis of Pinang Bridge Oprit Subsidence with Mortar Form Using LISA FEA V. 8." *Journal of Research and Inovation in Civil Engineering ….*

Elsanadedy, H. M., Y. A. Al-Salloum, M. A. Alrubaidi, and ... 2021. "Finite Element Analysis for Progressive Collapse Potential of Precast Concrete Beam-to-Column Connections Strengthened with Steel Plates." *Journal of Building ….*

Ertas, A. H., and M. Akbulut. 2021. "Experimental Study on Fatigue Performance of Resistance Spot-Welded Sheet Metals." … *International Journal of Advanced Manufacturing ….* doi: 10.1007/s00170-021-06822-z.

Fu, L., Z. Tian, Y. Zheng, W. Ye, and S. Zhou. 2022. "The Variation of Load Diffusion and Ballast Bed Deformation with Loading Condition: An Experiment Study." *International Journal of Rail ….* doi: 10.1080/23248378.2021.1897958.

Jideani, T. C. U., and P. J. Grabe. 2019. "The Development of Suitable Cyclic Loading and Boundary Conditions for Ballast Box Tests." *Journal of the South African Institution of Civil ….*

Kaveh, A., and Talatahari, S. 2017. "Structural Analysis with the LISA System: Step by Step Approach." *CRC Press.*

Li, L., Zhang, Y., Liu, Y, and Wang, Y. 2019. "Finite Element Analysis of Welded Joint in Steel Structure." *Journal of Constructional Steel Research* (154):290-299.

Wang, Z, Wu, T, Wang, L., Zhao, X., and Xu, Y. 202AD. "Finite Element Analysis on Stress Distribution and Fatigue Life Prediction of High-Strength Steel Welded Joints." *Journal of Constructional Steel Research, 165, 105921.*

Zhang, Z, Zhang, Y., and Liu, H. 2018. "Optimization of Fatigue Performance for Welded Details of Steel Bridges." *Engineering Structures,* (156,):570-584.