Stress Behavior of Rampdoor Jetty Bumi Harapan Port Sepaku-IKN Using LISA FEA V.8.

Aco Wahyudi Efendi[1]

1) Jurusan Teknik Sipil, Universitas Sebelas Maret, Indonesia
Email : aw.efendi2018@gmail.com

ABSTRAK

Alat berat yang digunakan dalam pekerjaan ini adalah 1 buah crane barge di area laut, dan 2 buah excavator breaker di area bongkar muat material di dermaga Pelabuhan PT MLM yang akan dibongkar. Excavator yang digunakan dalam pekerjaan pembongkaran ini dimobilisasi melalui dermaga terdekat. Sebagai aspek keamanan dan keselamatan agar tidak terjadi apa-apa, maka kondisi rampdoor harus diperhitungkan beban yang akan diderita saat excavator melintasi rampdoor tersebut, apakah kuat atau tidak.

Pada penelitian ini, metode pemodelan rampdoor dilakukan dengan menggunakan software finite element analysis dengan LISA FEA V.8 (lisensi) untuk mengetahui kapasitas tegangan dan kemampuan layan dari rampdoor tersebut.

Dari hasil pemodelan dan analisis numerik menggunakan analisis elemen hingga LISA FEA V.8 (license) didapatkan beberapa perilaku tegangan dan defleksi yang terjadi akibat beban yang diberikan dengan dua kondisi pemodelan beban dimana kondisi tersebut adalah saat mesin mulai melintasi rampdoor dan saat berada ditengah bentang rampdoor, dapat dilihat bahwa rasio defleksi yang terjadi dari kedua kondisi tersebut adalah sebesar 1.754, dimana defleksi awal saat memasuki rampdoor terjadi defleksi sebesar 0.1231 m pada tengah bentang dan saat mesin berada ditengah bentang menjadi 0.216 m, sedangkan momen yang terjadi berbeda dengan rasio 1.576 dengan 2 kondisi yang sama pada permodelan beban dan rasio tegangan yang terjadi sebesar 0.844.

Kata Kunci : Defleksi, FEA, LISA, Rampdoor, Tegangan

Abstract

The heavy equipment used in this work is a crane barge in the sea area, and 2 excavator breakers in the loading and unloading area of the material at the jetty of PT MLM Port to be unloaded. The excavators used in this demolition work are mobilized through the nearest jetty. As a security and safety aspect so that nothing happens, the condition of the rampdoor must be taken into account the load that will be suffered when the excavator crosses the rampdoor, whether it is strong or not.

In this study, the rampdoor modeling method was carried out using finite element analysis software with LISA FEA V.8 (license) to determine the stress capacity and serviceability of the rampdoor.

From the results of modeling and numerical analysis using finite element analysis LISA FEA V.8 (license) obtained some stress and deflection behavior that occurs due to the load given with two load modeling conditions where the condition is when the machine starts to cross the rampdoor and when it is in the middle of the rampdoor span, it can be seen that the deflection ratio that occurs from these two conditions is 1.754, where the initial deflection when entering the rampdoor occurs a deflection of 0.1231 m in the middle of the span and when the machine is in the middle of the span to 0.216 m. while the moment that occurs is different from the ratio of 1.576 with the same 2 conditions in the load modeling. and the stress ratio that occurs is 0.844.

Keywords: Deflection, FEA, LISA, Rampdoor, Stress

INTRODUCTION

In connection with the Bumi Harapan port development work in order to provide support for development in the Nusantara National Capital Region in Sepaku, where in the port activities there will be material loading and unloading activities at the PT. MLM Port jetty, which involves various types of heavy equipment that will be used for the work. The work of loading and unloading materials at the jetty of PT MLM Port has two areas used, namely the land area and the sea area. With these two areas, the planning between heavy equipment and the work field must be calculated as much as possible. The calculation aims to produce a feasibility value so that all work goes as originally planned [1].

The heavy equipment used in this work is a crane barge in the sea area, and 2 excavator breakers in the loading and unloading area of the material at the jetty of PT MLM Port to be unloaded. The excavators used in this demolition work are mobilized through the nearest jetty. As a security and safety aspect so that nothing happens, the condition of the rampdoor must be taken into account the load that will be suffered when the excavator crosses the rampdoor, whether it is strong or not.

In this study, the rampdoor modeling method was carried out using finite element analysis software with LISA FEA V.8 (license) to determine the stress capacity and serviceability of the rampdoor.

The ramp door structure is planned with the following general provisions:

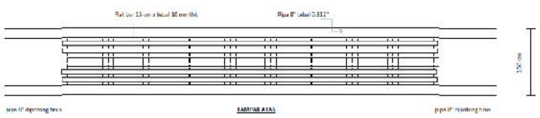


Figure 1. Top view of Ramp Door structure

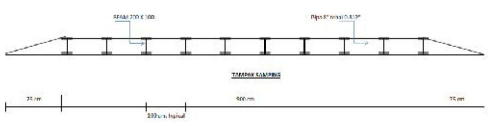


Figure 2. Side view of Ramp Door structure

The work location is in Bumi Harapan Village in the IKN Area Development plan area, East Kalimantan. As shown in the picture below:

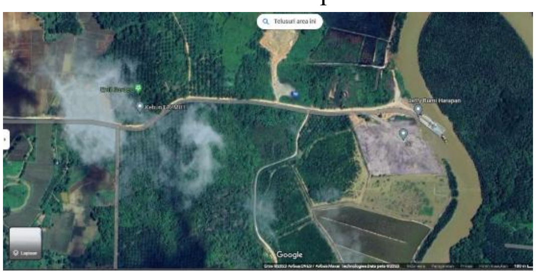


Figure 3. location of the research review

RESEARCH METHODS

This research focuses on the results of the rampdoor design that will be crossed by heavy equipment in the form of an excavator using LISA FEA V.8 (license) by simulating static loading still in safe boundary conditions in terms of finite element analysis.

The finite element method (FEM) is a mathematical technique for dealing with specialized examination issues. The restricted component strategy consolidates a few numerical ideas to create straight or nonlinear framework conditions. The number of conditions created is usually very large, reaching more than 20,000 conditions. Therefore, this strategy is very low in value unless a reasonable PC is utilized.

The finite element method uses component discretization to solve the problems of tracking node/association/grid relocation and primary strength. The discrete component condition is linked to the lattice technique for primary examination and the results obtained are indistinguishable from traditional investigations for structures. Discretization should be possible with one-layer (line component), two-layer (plane component) or three-layer (volume/continuum component) components. This approach uses continuum components to determine answers that are close to reality[2], [3], [4], [5], [6], [7], [8], [9], [10], [11].

LISA FEA V.8. LISA, a numerical analysis testing software, was used to measure the temperature rise for three different intensity exchanger models. The three types of models are, organized by simple modeling, line component model, shell model, and robust model.

The finite element model is a mesh of elements. Each element has nodes that only point to the element. Elements can only be connected to other elements node-to-node. Edge-to-element-node elements have no connection at all. The elements themselves have very simple shapes such as lines, triangles, squares, cubes, and pyramids, seen as figure 2.

Each element is formulated to obey certain laws of science. For example, in static analysis, elements are formulated to relate displacement and stress according to the theory of mechanics of materials. In the case of modal vibration, elements are formulated to follow the shape of deflection and frequency according to the theory of structural dynamics. Similarly, in thermal analysis, elements relate temperature and heat according to heat transfer theory [2], [8], [12], [13], [14], [15], [16], [17], [18], [19], [20], [21].

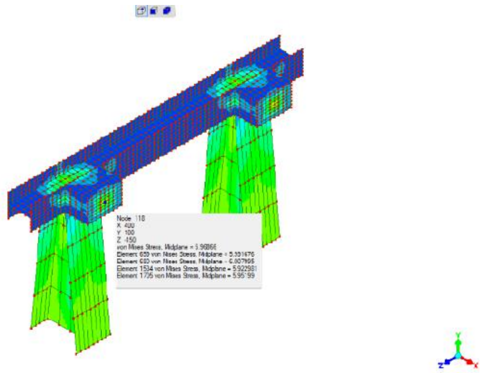


Figure 4. Element modeling in LISA program

Dead load

Ultimate load factor: KMS = 1.30. Self-weight is the weight of materials and parts of the bridge that are structural elements, plus the non-structural elements they carry and are fixed. The self weight of the upper structure is calculated as follows:

- Reviewed rampdoor width,
  b = 1.00 m
- Rampdoor length
  L = 1.50 m
- Thickness of ramdoor floor,
  h = ts = 0.01 m
- Weight of steel
  W_c = 77.00 kN/m^3
- Weight of WF
  profile 200x100x8x5.5
  W_1 = 21.30 kg/m = 0.213 kN/m^3
- Weight of steel pipe dia 8"
  W_2 = 42.10 kg/m = 0.421 kN/m^3
- Weight of WF
  profile 250x125x9x6
  W_1 = 29.60 kg/m = 0.296 kN/m^3
- Self weight,
  QMS = b * h * Wc = 2.843 kN/m
- Moment and shear force due to self weight,
  MMs = 1/12*QMS*L2
  = 0.533063 kNm
  VMS = 1/12 * QMS * L2
  = 0.400 kN.

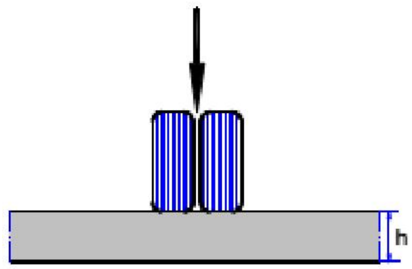
Additional Dead Load (MA)

Ultimate load factor: KMA = 2.00, Additional dead load (superimposed dead load), is the weight of all materials that cause a load on the bridge which is a non-structural element, and may change in magnitude during the life of the bridge. the bridge analyzed must be able to carry additional loads such as:

1) The addition of an asphalt layer (overlay) at a later date
2) Puddles of rainwater if the drainage system is not working properly.

Truck Load "T" (TT)

Ultimate load factor: KTT = 1.80, live load on the bridge deck in the form of double wheel load by Excavator PC 200 (T load) which amount, T = 110.00 kN. Dynamic load factor for truck loading at 0.2
Truck load "T" PTT = (1+DLA)* T = 132,000 kN.



Slab span length, L = 1.50 m

The load is made to be equally distributed with the case of moving vehicle movement assuming fixed wheel friction of 100%. Spread area

- L = 1.50 m
- P = 3.00 m
- Area = 4.50 m^2
- P per rc 132.00 kN
- 29.33 kN/m^2

RESULTS AND DISCUSSION

Analysis and modeling of the structure was carried out using LISA FEA V.8 (license). The plot of structural modeling and loading on the structure is shown in the figure below.

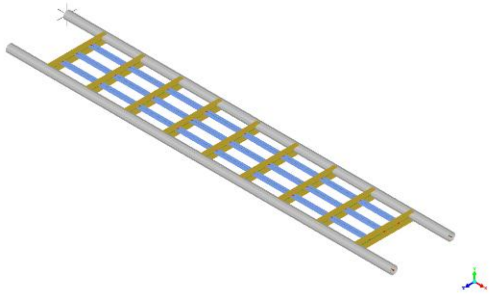


Figure 5. 3D modeling of the rampdoor

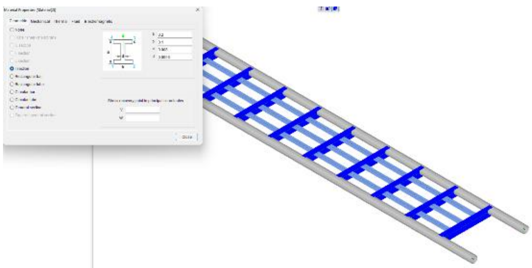


Figure 6. Section Properties WF 200x100x8x6,6

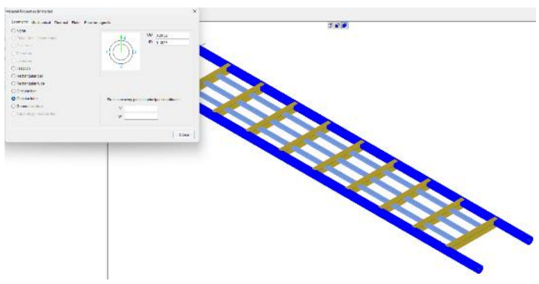


Figure 7. Pipe Section Properties

Structure Analysis Results

The first loading case assumes the machine is on the initial pedestal before crossing.

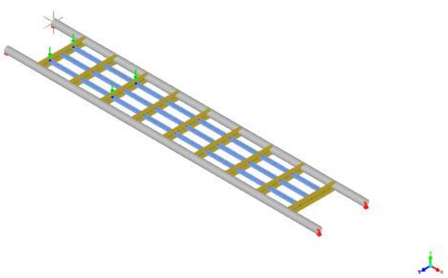


Figure 8. Loading Before Heavy Equipment Crosses

When the machine passes on the initial side of the support, a deflection of 0.084 m occurs on the side of the support where the machine stands according to Figure 10 and a deflection of 0.1231 m occurs at the center of the span according to Figure 10.

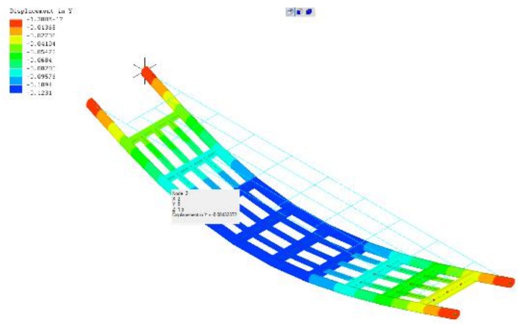


Figure 8. Deflection in the support area where the machine stands

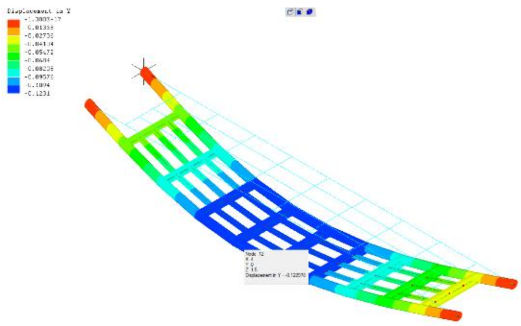


Figure 10. Deflection that occurs in the middle of the span

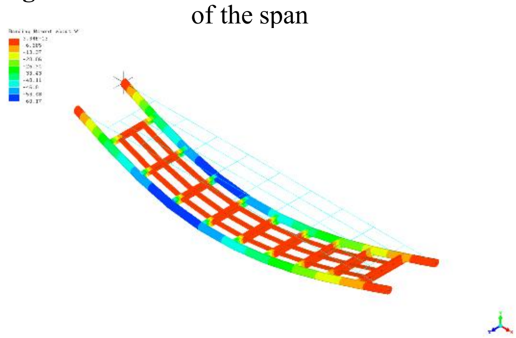


Figure 11. Moment that occurs

The moment that occurs when the machine passes in the support area is 60.17 kNm as shown in Figure 11 and the stress that occurs when the machine passes in the support is 37385.875 kN/m^2 as shown in Figure 12 and the allowable stress of steel is 1,600,000 kN/m^2 so the stress that occurs is still below the allowable stress and the element is still in a safe condition.

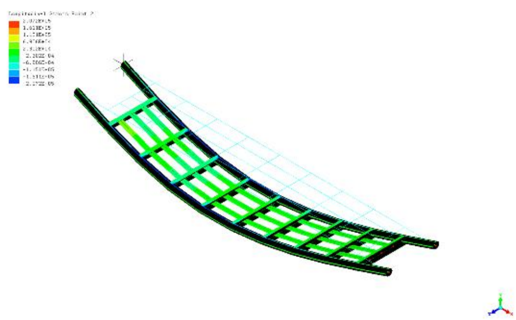


Figure 12. Stress that occurs

The second condition is loading in the middle of the span with the assumption that the excavator is in the middle of the rampdoor span with the same static load as the first condition. The load modeling is shown in Figure 13.

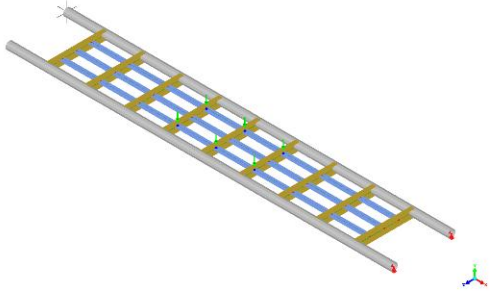


Figure 13. Loading when the machine is in the middle of the span.

When the machine passes through the center of the rampdoor, a deflection of 0.216 m occurs on the side of the support where the machine stands according to Figure 14.

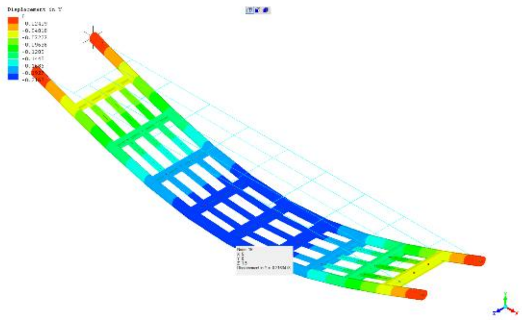


Figure 14. Deflection that occurs at the rampdoor when the machine is in the middle of the span

The moment that occurs when the machine passes in the middle of the rampdoor span is 94.86 kNm as shown in Figure 15 and the stress that occurs when the machine passes in the middle of the rampdoor span is 31564.413 kN/m^2 as shown in Figure 16 and the allowable stress of steel is 1,600,000 kN/m^2, the stress that occurs is still below the allowable stress and the element is still safe.

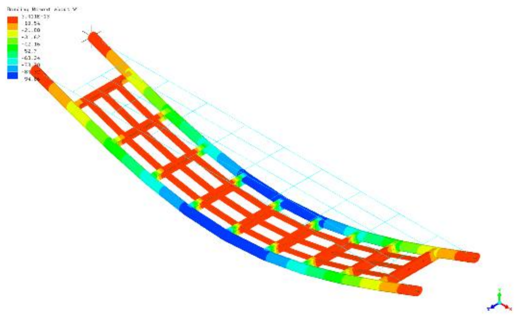


Figure 15. Moment that occurs

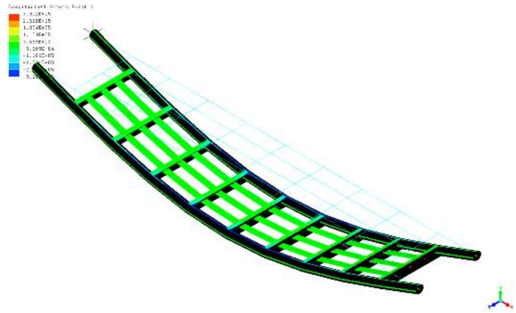


Figure 16. Stress that occurs

CONCLUSIONS

From the results of modeling and numerical analysis using finite element analysis LISA FEA V.8 (license) obtained some stress and deflection behavior that occurs due to the load given with two load modeling conditions where the condition is when the machine starts to cross the rampdoor and when it is in the middle of the rampdoor span, it can be seen that the deflection ratio that occurs from these two conditions is 1.754, where the initial deflection when entering the rampdoor occurs a deflection of 0.1231 m in the middle of the span and when the machine is in the middle of the span to 0.216 m. while the moment that occurs is different from the ratio of 1.576 with the same 2 conditions in the load modeling. and the stress ratio that occurs is 0.844.

This research needs to be upgraded to experimental research to get the actual behavior and become validity in the behavior that occurs.

REFERENCE

[1] A. W. Efendi, “Modeling of soil subsidence in IKN using numerical analysis of the finite element method LISA V. 8.” researchgate.net, 2020. [Online]. Available: https://www.researchgate.net/profile/Aco-Efendi/publication/367090313_Modeling_of_soil_subsidence_in_IKN_using_numerical_analysis_of_the_finite_element_method_LISA_V8/links/63c0f9964c7e7c4e5125262a/Modeling-of-soil-subsidence-in-IKN-using-numerical-analysis-of-the-finite-element-method-LISA-V8.pdf

[2] A. W. Efendi, “Behavior Analysis of Building Structures After a Fire with FEA LISA V. 8,” *Kurva J. Keilmuan Dan Apl. Tek.* ..., 2022, [Online]. Available: http://ejurnal.untag-smd.ac.id/index.php/TEKNIKD/article/view/6413

[3] J. M. Carbonell, L. Monforte, M. O. Ciantia, M. Arroyo, and ..., “Geotechnical particle finite element method for modeling of soil-structure interaction under large deformation conditions,” ... *and Geotechnical* .... Elsevier, 2022. [Online]. Available:

https://www.sciencedirect.com/science/article/pii/S1674775521001943

[4] M. T. Chen and B. Young, "Experimental and numerical investigation on cold-formed steel semi-oval hollow section compression members," *J. Constr. Steel Res.*, 2018, [Online]. Available: https://www.sciencedirect.com/science/article/pii/S0143974X18301937

[5] A. Recep and S. Sedat, "Investigation of the Consolidation Behavior of Soft Soil Improved with Vertical Drains by Finite Element Method," *Int. J. Eng. Appl. …*, 2021, [Online]. Available: https://dergipark.org.tr/en/pub/ijeas/issue/66141/1002115

[6] C. Tong, J. Wu, M. D. Zakari, and Y. Gao, "Hysteretic behavior of a new precast concrete interior beam-to-column joint with a controllable bending moment," *J. Build. Eng.*, 2022, [Online]. Available: https://www.sciencedirect.com/science/article/pii/S2352710221014157

[7] A. W. Efendi, "Computing steel frame analysis of behavior semi-rigid connection," *Computing*. researchgate.net, 2023. [Online]. Available: https://www.researchgate.net/profile/Aco-Efendi/publication/365344223_Computing_steel_frame_analysis_of_behavior_semi-rigid_connection/links/63708e4054eb5f547cca827d/Computing-steel-frame-analysis-of-behavior-semi-rigid-connection.pdf

[8] A. W. Efendi, "PEMODELAN PENURUNAN TANAH DI IBU KOTA NEGARA NUSANTARA MENGGUNAKAN ANALISIS NUMERIK METODE ELEMEN HINGGA LISA V. 8," *PADURAKSA J. Tek. Sipil …*, 2023, [Online]. Available: https://www.ejournal.warmadewa.ac.id/index.php/paduraksa/article/view/5643

[9] H. M. Elsanadedy, Y. A. Al-Salloum, M. A. Alrubaidi, and ..., "Finite element analysis for progressive collapse potential of precast concrete beam-to-column connections strengthened with steel plates," *J. Build. …*, 2021, [Online]. Available: https://www.sciencedirect.com/science/article/pii/S2352710220335087

[10] T. M. Gondhalekar and S. K. Panigrahi, "Transient Analysis of Railway Sleeper using Three-Dimensional Finite Element Method," *J. Inst. Eng. …*, 2021, doi: 10.1007/s40030-021-00582-5.

[11] T. M. Lenkovskiy, V. V. Kulyk, Z. A. Duriagina, L. V. Dzyubyk, and ..., "Finite elements analysis of the side grooved I-beam specimen for mode II fatigue crack growth rates determination," *Journal of Achievements in …*. 2018.

[12] A. W. Efendi, "Simulation of fire exposure behavior to building structural elementsusing LISA FEA V.8," *Int. J. Adv. Sci. Comput. Appl.*, vol. 1, no. 1, pp. 41–50, Mar. 2022, doi: 10.47679/ijasca.v1i1.8.

[13] A. W. Efendi, "Behavior of Railroad Bridge Girders Due to Brake Loads with LISA V. 8 FEA," ... *Conf. Railw. Transp. ICORT …*, 2023, [Online]. Available: https://www.atlantis-press.com/proceedings/icort-22/125985594

[14] A. W. Efendi and C. Weijia, "Buffer stops behavior due to rail impact loads with LISA FEA," *J. Railw. Transp. Technol.*, 2023, [Online]. Available: https://jrtt.org/index.php/jrtt/article/view/20

[15] A. W. Efendi, "Modeling of soil subsidence in IKN using numerical analysis of the finite element method LISA V. 8." researchgate.net, 2020. [Online]. Available: https://www.researchgate.net/profile/Aco-Efendi/publication/367090313_Modeling_of_soil_subsidence_in_IKN_using_numerical_analysis_of_the_finite_element_method_LISA_V8/links/63c0f9964c7e7c4e5125262a/Modeling-of-soil-subsidence-in-IKN-using-numerical-analysis-of-the-finite-element-method-LISA-V8.pdf

[16] I. A. W. Efendi, "Repair analysis of Pinang Bridge oprit subsidence with mortar form using LISA FEA V. 8," *Journal of Research and Inovation in Civil …*. researchgate.net, 2022. [Online]. Available: https://www.researchgate.net/profile/Aco-Efendi/publication/364959931_Repair_analysis_of_Pinang_Bridge_oprit_subsidence_with_mortar_form_using_LISA_FEA/links/63612e80431b1f53005fc528/Repair-analysis-of-Pinang-Bridge-oprit-subsidence-with-mortar-form-using-LISA-FEA.pdf

[17] I. EFENDI, "Noise exposure impact zone hue modeling using LISA FEA V. 8," *Tek. J. Ilmu Tek. Dan …*, 2022, [Online]. Available: http://journal.stiestekom.ac.id/index.php/TEKNIK/article/view/153

[18] A. W. Efendi, "Desain Kandang Sperma Ideal untuk Sapi menggunakan LISA V. 8 FEA," *J. Sint. Penelit. Sains Terap. Dan …*, 2023, [Online]. Available: https://www.jurnal.iik.ac.id/index.php/jurnalsintesis/article/view/71

[19] A. W. Efendi, Y. Do, and N. F. Rachman, "Behavior of Rail Ballast Layer Using Mortar Foam with LISA-FEA," *J. Railw. Transp. …*, 2022, [Online]. Available: https://www.jrtt.org/index.php/jrtt/article/view/8

[20] A. W. Efendi, "Behavior of railroad bearing due to temperature and load using LISA FEA," *J. Railw. Transp. Technol.*, 2022, [Online]. Available: https://jrtt.org/index.php/jrtt/article/view/1

[21] A. W. Efendi and N. Safitri, "Numerical modeling the effect of extreme temperatures on rooms with mortar foam walls using LISA V.8 FEA," *CAFEO 41 Eng. Conf.*, vol. 41, pp. 28–36.