Draft

by Indra Permana

Submission date: 19-Apr-2022 03:36PM (UTC+0700)
Submission ID: 1814353343
File name: ft_Jurnal_Floor_Support_Structure_Analysis_-_Jurnal_Angkasa.docx (334.78K)
Word count: 3243
Character count: 16905

Angkasa Jurnal Ilmiah Bidang Teknologi
ISSN : 2085-9503 (Print)
ISSN : 2581-1355 (On Line)
Accredited Fourth Grade by Kemenristekdikti, Decree No: 21/E/KPT/2018
DOI : 10.28989/Angkasa.vxxx.xxx

Preliminary Stress Analysis of Aircraft Floor Support Structure Subjected to Emergency Landing Loads using Finite Element Method

Indra Permana[1], Muhammad Ikhsan[2]
[1,2]Faculty of Engineering, Universitas Tunas Pembangunan Surakarta, Indonesia
Email: indrapermana@lecture.utp.ac.id

Received: April, 2022; Accepted : - ; Published : -

ABSTRACT (10 PT)

Floor support structure is a structural member of aircraft fuselage that have a function to support passenger and other payload loads inside the cabin. One type of load cases that should able to be carried on by floor support structures without permanent deformation is the emergency landing loads. This study aims to evaluate the strength of floor support structure of regional transport aircraft that consist of floor beams, seat tracks, and stanchions due to emergency landing loads. The study is carried on using Finite Element Method in MSC Patran/Nastran software. The structures are modeled using 1D element and the load is modeled as inertial loads given to the lumped mass of passenger and seat. The margin of safety is then calculated to evaluate the yielding of the structure material. The results show that in all cases the margin of safety is larger than zero which means that the structure is sill in elastic zone and there is no permanent deformation due to emergency landing load cases.

Keywords: floor support structure, Emergency landing loads, Finite element method

1. INTRODUCTION

The modern transport aircraft commonly use semi-monocoque construction to carry all possible loads during its operation. Semi-monocoque construction use skin reinforced by longitudinal members called stringer or longeron to withstand the fuselage loads [1] To maintain the fuselage into desired cross-sectional shape, the skin-stringer is supported by frame assemblies and bulkheads. Transport aircraft needs to have floor to accommodate passenger and seat inside the cabin. The floor panel should have structural supports to withstand the passenger and seat loads during the flight. Together with skin, stringer and frame, floor support structure establishes the structural integrity for the fuselage.

Commonly, floor support structure consists of lateral floor beams located in each fuselage frame supported by vertical struts to reduce the bending moments in the floor beams [2]. The passenger seats are connected to seat track beams that attached longitudinally to the floor beam from forward to rear cabin. The seat tracks also act as stiffening members and provide 9.0g forward crash load restraint for passenger seats [2].

Floor support structure should be designed to carry floor and seat loads. One of the critical loads that should be supported by this structure is the loads in emergency landing conditions. Based on CASR part 25.561, seat and its supporting structure must not deform in emergency landing conditions, in such a way that the deformation would interfere the rapid evacuation process [3]. It has been stated also in this section that all inertia forces due to emergency landing should be supported by the structure. Evaluation should be carried on by the aircraft manufacturer to prove that the floor support structure able to withstand this type of loads.

One method that can be used to showing compliance with the applicable requirements is by analysis. The analysis of aircraft structures commonly performed using finite element method. This method is widely used by researchers as well as aircraft manufacturers and become one of the robust methods to ensure that the aircraft structure is strong enough to withstand all possible loads and comply with the safety requirements. This is proven by many studies regarding aircraft structural analysis, including static, dynamic, fatigue, and other structural analysis aspect that can be conducted using finite element method. Some of the related studies are Hartini [4] that use finite element method to evaluate the strength of fuselage stringer after getting repaired using angle and Hadi, et al [5] which conduct the study of flutter speed characteristic of high aspect ratio composite wing. The finite element method has been already well-known and reliable method to study the strength of the aircraft structures.

There are some researchers using finite element method to evaluate the strength of the aircraft floor structures. Yadav [6], [7] perform study about static structural analysis, modal analysis, and life estimation of aircraft floor beam made from carbon fiber reinforced plastic (CFRP) material. His study shows that the floor beam made from CFRP material is estimated have longer life cycle than the existing floor beam made from aluminum material. Kotresh B, et al. [8] in 2016 was published an article about design, analysis, and optimization of fuselage floor beam made from CFRP material. Xianfei, et al [9] performed nonlinear finite element analysis to evaluate crashworthiness of aircraft fuselage that have under-floor cargo compartment. They evaluate the effect of luggage to the crashworthiness characteristic of the fuselage. The crash simulation is performed on fuselage section finite element model with aluminum material that is subjected to impact velocity of 9.14 m/s.

In this study, preliminary analysis of the floor support structure of regional transport aircraft due to emergency landing loads is presented. The study is carried on by performing linear stress analysis using finite element method in MSC Patran/Nastran software. The objective of the analysis is to obtain the critical loads and evaluate the strength of floor support structures subjected to inertial loads due to emergency landing conditions. The load cases are constructed from all possible inertial loads mentioned in CASR 25.561(b)(3) [3] combined with two seat arrangement positions that are considered give the worst effect to the structures. The structural strength is evaluated by finding the margin of safety which calculate the ratio between the maximum or minimum stress results and the yield strength of the material.

2. RESEARCH METHODOLOGY

The preliminary structural analysis of floor support structure is conducted using Finite Element Method in MSC Patran/Nastran software. The type of analysis is linear static stress structural analysis. The analysis flowchart is presented in the Figure 1.

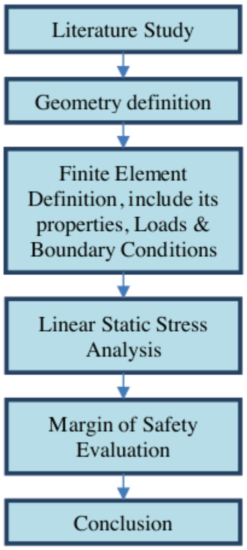


Figure 1. Analysis Flowchart

The pre-processing step is to define the geometry of each structural member, define the element type, material properties, load and boundary conditions. The solution process is the step performed by the computer, in which it computes the element stiffness matrix and find the solution for given loads and boundary conditions. The post-processing shows the stress distribution and deformation as a result of linear static stress analysis. The stress result is then compared with the yield strength of the material to find the Margin of Safety. The maximum tensile stress is compared with the tensile yield strength, while minimum compression stress is compared with the compressive yield strength. The evaluation of the margin of safety is then made to see if the floor support structure is safe under given loads and to determine the critical load cases in each structural component.

One element commonly used in finite element model is the 1D beam element. The beam is a structural element which is relatively long compared to a characteristic cross-section [10]. Floor support structures have geometry that match with this definition, so that in this analysis it is modeled using 1D element to see the global characteristic of the structures. In MSC Patran/Nastran software, the floor beams and seat tracks are modeled using CBEAM element, while the stanchions are modeled using CBAR element. One of the main differences between these two elements is that the CBEAM element commonly used when the beam cross section is unsymmetric, in which the neutral axis and shear center may not coincide. This feature is not present in the CBAR element. The 1D element used in this analysis that performed in MSC Patran/Nastran software is derived from classical beam theory, which valid when the plane cross sections remain plane during deformation [11].

2.1. Floor Support Structure Configuration

Floor support structure for regional transport aircraft being studied in this article consists of floor beams, stanchions, and seat tracks. Floor beams are connected to the fuselage frame supported by two vertical stanchions. Four seat tracks are used as longitudinal supporting structure to attach the seats and transfer it loads to the floor beams. The arrangement of the floor support structure is shown by centerline diagram in Figure 2.

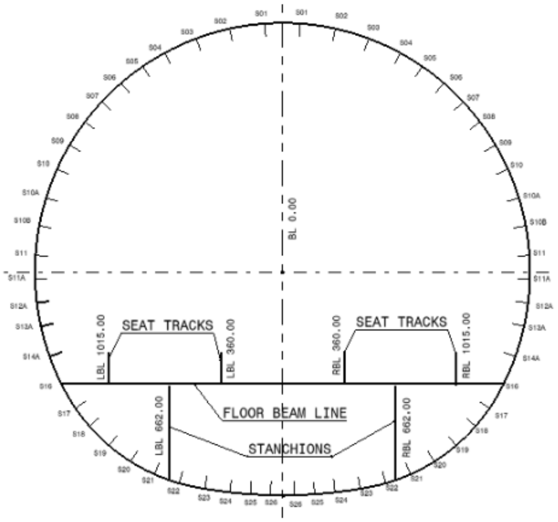


Figure 2. Floor Support Structure Centerline Diagram in Cabin Cross Section View

2.2. Floor Support Structure Finite Element Model

In this study, only floor support structure in two frames are modeled and evaluated to represent the whole typical floor support structure in the entire fuselage. Floor beam, stanchions, and seat tracks are modeled using 1D element. The cross section of each part is shown in Figure 3. Floor beam and seat track have 2 mm thickness, while floor beam have 1.8 mm thickness.

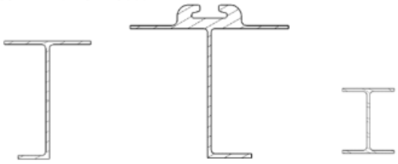


Figure 3. Cross-section of Floor Beam, Seat Track, and Stanchion, respectively (not in scale)

The finite element model including the boundary conditons of the floor support structure is shown in Figure 4. The floor beam and seat track have "t" section which is not symmetry, while stanchion has "I" section which is symmetry. The consequence is the floor beam and seat track have neutral axis that does not coincide with the shear center. In MSC Patran/Nastran software, floor beam and seat track are modeled using CBEAM element and stanchion use CBAR element. The following boundary conditions are used in the model:

1. Floor beam fix boundary condition, represents the connection between floor beam to fuselage frame.

2. Stanchion fix boundary condition: represents the connection between stanchions to fuselage frame.
3. Seat track boundary condition: restraints translation displacement in X direction and rotation in Y and Z direction.

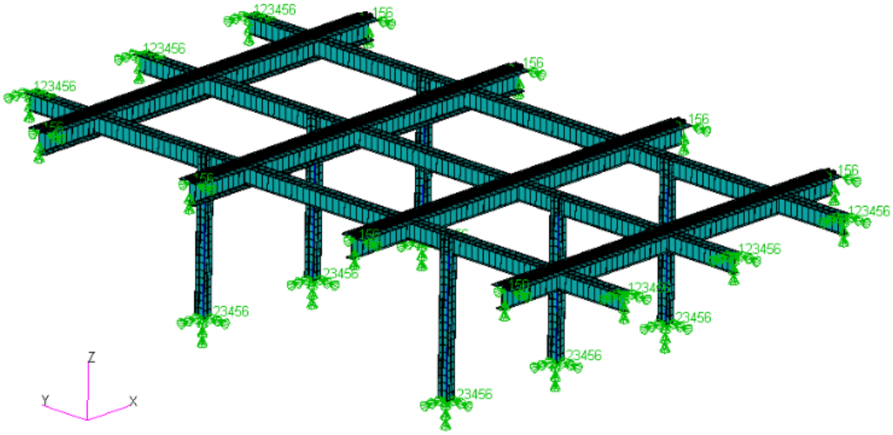


Figure 4. Finite Element Model of Aircraft Floor Support Structure

2.3. Material Properties

Floor support structure use Aluminum alloy 7075-T76511 extrusion. The important material properties of Al 7075-T76511 used in this study are summarized in Table 1, specified for extrusion thickness between 0.062-0.249 inches, A-basis and longitudinal grain boundaries.

Table 1. Al 7075-T76511 Material Properties [12]

Tensile Ultimate Strength, ksi (MPa)	71 (489.53)
Tensile Yield Strength, ksi (MPa)	61 (420.58)
Compression Yield Strength, ksi (MPa)	61 (420.58)
Tensile Modulus of Elasticity, 10^3 ksi (GPa)	10.4 (71.705)
Compression Modulus of Elasticity, 10^3 ksi (GPa)	10.7 (73.774)
Poisson's Ratio	0.33
Density, lb/in^3 (kg/m^3)	0.101 (2795.7)

2.4. Loads Model

In this study, floor support structure is loaded with emergency landing loads specified in CASR 25.561(b)(3) [3]. The seat and passenger mass need to be modeled in the finite element model to represent these loads. In this study, seat is assumed to have four legs which connected to the seat track. The passenger mass, refer to CASR 25.785(f) [3], is 77 kg.

Seat and passenger mass are modeled using one mass element located at its center of gravity with the total mass of each mass element is 174 kg. There are some possible locations (in aircraft longitudinal direction) of the mass element relative to the fuselage frame: it can be exactly at the middle of two fuselage frames, same locations with frame, or somewhere between two frames. From that many possibilities, only two locations are evaluated: first if the mass CG location is exactly at the middle of two frames (shown in Figure 5) and second if the mass CG location is exactly in the same position with frame (shown in Figure 6). The nodal mass of seat and passenger is connected to seat track element node using RBE2.

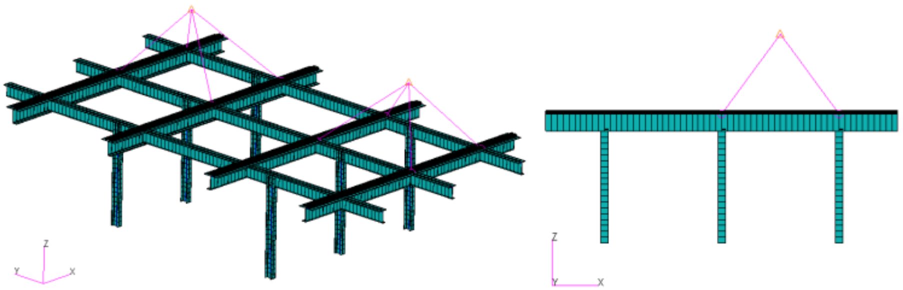


Figure 5. Seat and Passenger Mass Model Location 1

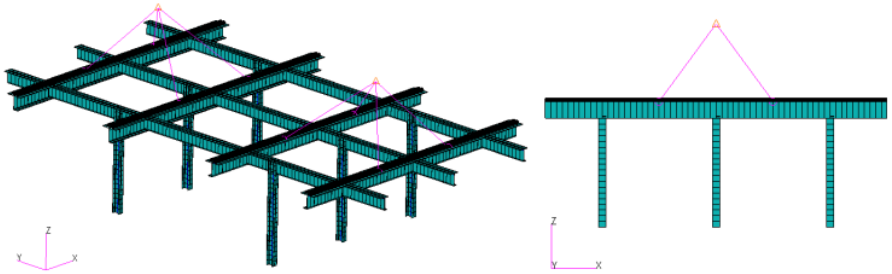


Figure 6. Seat and Passenger Mass Model Location 2

3. RESULTS

The results of stress analysis are divided into two parts. First part shows the result of stress analysis in Floor beam and Seat track as both components are modeled using CBEAM element. The second part shows the result of stress analysis in Stanchion as it is modeled using CBAR element.

3.1. Stress Analysis Results of Floor Beams and Seat Tracks

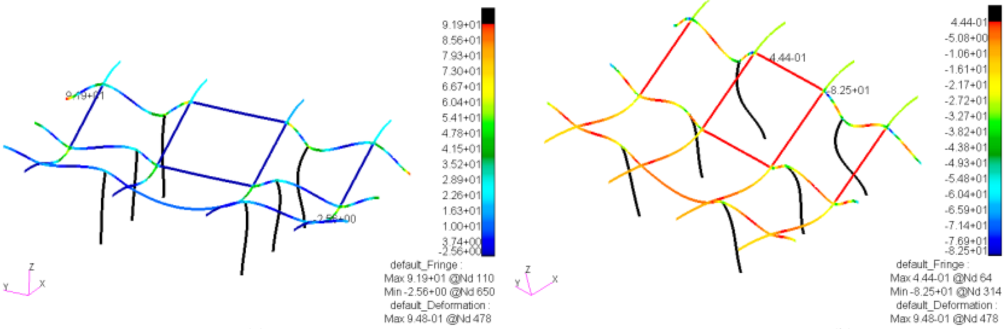
The results of stress analysis of floor beams and seat tracks are showed in Table 2 and Table 3, in which it presented in maximum and minimum combined stresses for each load case. The stress results comparison for two configurations of seat arrangement are also presented. The margin of safety values, which describe the ratio between material yield strength and the stress, is used here to evaluate the yielding of the material. The structure should have MS>0 in each case to show that the material deformation is still elastic.

Table 2. Maximum Stress Results of Floor Beams and Seat Tracks

CASE	Seat Attachment Location 1		Seat Attachment Location 2	
	Stress (MPa)	MS	Stress (MPa)	MS
1	33.8	11.44	33.6	11.52
2	91.9	3.58	77.1	4.45
3a	54.6	6.70	64.5	5.52
3b	54.3	6.75	63.6	5.61
4	61	5.89	46.8	7.99
5	13.7	29.70	12.7	32.12

Table 3 Minimum Stress Results of Floor Beams and Seat Tracks

CASE	Seat Attachment Location 1		Seat Attachment Location 2	
	Stress (MPa)	MS	Stress (MPa)	MS
1	-30.5	12.79	-23.4	16.97
2	-82.5	4.10	-76.1	4.53
3a	-54.3	6.75	-63.6	5.61
3b	-54.6	6.70	-64.5	5.52
4	-67.6	5.22	-67.2	5.26
5	-15.3	26.49	-12.8	31.86



(a) (b)

Figure 7 Floor Beam and Seat Track Stress Distribution Results (deformation not in true scale) (a) Maximum Combined Stress, Load Case 2, Seat Attachment Location 1; (b) Minimum Combined Stress Load Case 2, Seat Attachment Location 1;

The maximum stress result is 91.9 MPa that occurs when the structure is applied with load case 2, 9g forward inertial load, with the seat attached at location 1. The tensile margin of safety is 3.58, which shows that the material is not yield yet. Figure 7(a) shows the stress distribution result of Case 2 for maximum combined stress. The maximum stress occurs at the floor beam joint with frame. Note that the deformation of the structure in figure is not in true scale. The maximum structure deformation in load case 2 actually only 0.948 mm.

The minimum stress result is -82.5 MPa that occurs at load case 2 with the seat attached at location 1. Negative sign of the stress means that the structure experience compression load, thus it should be compared with the compressive yield strength to obtain the margin of safety. The result of margin of safety is 4.10, which means that the material is not yield yet. Figure 7(b) shows the stress distribution result of Case 2 for minimum combined stress. The critical stress occurs at the floor beam joint with seat track.

From the stress results, it can be seen that seat location 1 gives more severe stress to the floor beam and seat track structure than seat location 2 for symmetrical load case 1, 2, 4, and 5. But for non-symmetrical load case, which is load case 3a and 3b, seat location 2 gives more critical stresses to the structure than seat location 1.

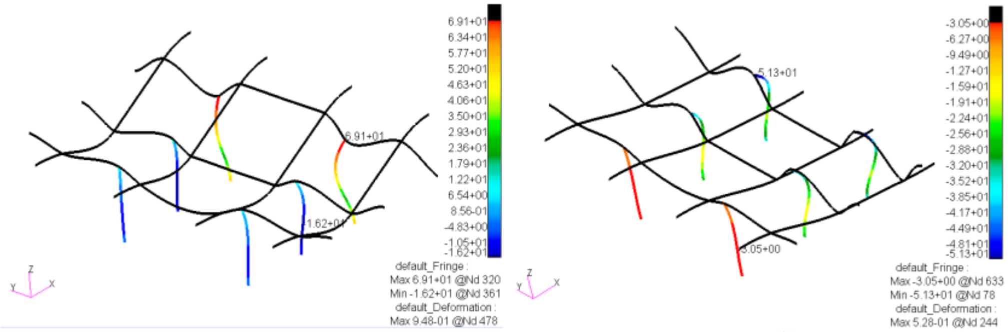
3.2. Stress Analysis Results of Stanchions

Table 4. Maximum Stress Results of Stanchions

CASE	Seat Attachment Location 1		Seat Attachment Location 2	
	Stress (MPa)	MS	Stress (MPa)	MS
1	25.7	15.36	19.8	20.24
2	69.1	5.09	53.8	6.82
3a	20.5	19.52	20.7	19.32
3b	20.6	19.42	21.6	18.47
4	12.6	32.38	10.9	37.59
5	7.23	57.17	8.67	47.51

Table 5. Minimum Stress Result of Stanchions

CASE	Seat Attachment Location 1		Seat Attachment Location 2	
	Stress (MPa)	MS	Stress (MPa)	MS
1	-6.28	65.97	-5.47	75.89
2	-43.4	8.69	-52	7.09
3a	-20.6	19.42	-21.6	18.47
3b	-20.5	19.52	-20.7	19.32
4	-51.3	7.20	-39.5	9.65
5	-11.5	35.57	-8.97	45.89



(a) (b)

Figure 8 Stanchions Stress Distribution Results (deformation not in true scale) (a) Maximum Combined Stress, Load Case 2, Seat Attachment Location 1; (b) Minimum Combined Stress Load Case 4, Seat Attachment Location 1;

The results of stress analysis of Stanchions are showed in Table 4 and Table 5 which presented in maximum and minimum combined stresses for each load case. The stanchions maximum stress is 69.1 MPa, which occurs under load case 2, seat attached at position 1. The margin of safety is 5.09. The stress distribution in stanchions for this case is presented in Figure 8(a).

The stanchions minimum stress is -52 MPa, which occurs under load case 2, seat attached at position 2. The margin of safety is 7.09. The result shows that load case 2, 9g forward loads, is the most critical load cases for the floor support structure. There is other load case that still gives lower stress result in stanchions than load case 2 but it is close enough, which is load case 4, 6g downward inertial load, with seat attached at location 1. This load case should also be considered to be critical load case because the stress value is so close and maybe it also critical in buckling.

4. CONCLUSION

The preliminary study of floor support structure of regional transport aircraft is performed to see the global stress distribution in each structural component. The linear static stress analysis is carried on using finite element method in MSC Patran/Nastran software. The result shows that the Case 2 (9g forward inertial load) combine with the seat attached at location 1 is the most critical load for floor beams and seat tracks, which lead to the lowest structural margin of safety of 3.58. For stanchions, the critical load in tension is also the load case 2 with seat located at position 1 resulting the structural margin of safety of 5.09. There are 2 load cases that considered critical for stanchions in compression, which are the load case 2 (9g forward inertial load) and load case 4 (6g downward inertial load), with compression margin of safety of 7.09 and 7.20 respectively. Based on this study, all margin of safety values is higher than 0, which means that the structure is still elastic and no yielding. The future study is to evaluate the buckling strength of the floor support structures due to emergency landing load cases and its strength for other load cases.

REFERENCES

[1] FAA, *Aviation Maintenance Technician Handbook – Airframe, Volume 1*, vol. 1. Oklahoma: U.S. Department of Transportation, Federal Aviation Administration., 2018.

[2] M. C. Y. Niu, *Airframe Structural Design*. Hong Kong: Conmilit Press Ltd., 1988.

[3] “Civil Aviation Safety Regulations Part 25 Amdt 6 Airworthiness Standards: Transport Category Airplanes.” Republic of Indonesia Ministry of Transportations, pp. 1–419, 2014.

[4] D. Hartini, “Strength Analysis on Airplane Fuselage Structure Stringer,” *Angkasa J. Ilm. Bid. Teknol.*, vol. 12, no. 2, pp. 185–194, 2020, doi: 10.28989/angkasa.v12i2.545.

[5] B. K. Hadi and I. Permana, “The Design of a High Aspect Ratio HALE Aircraft Composite Wing. Part II: Buckling and Flutter Speed analysis,” *J. Mech. Eng.*, vol. 12, no. 2, pp. 13–25, 2015.

[6] S. Yadav, “Modal Analysis and Life Estimation of Floor Beam Aircraft Structure,” *Int. J. Sci. Res. Dev.*, vol. 7, no. 05, pp. 842–846, 2019, doi: 10.13140/RG.2.2.22287.12967.

[7] S. Yadav, “Design and Static Structural Analysis of Aircraft Floor Beam,” *GRD Journals- Glob. Res. Dev. J. Eng.*, vol. 04, no. 08, pp. 7–10, 2019, doi: 10.13140/RG.2.2.11381.93929.

[8] Kotresh, M. Haneef, and M. Imran, “Design , Analysis and Optimization of Fuselage Floor Beam,” *Int. J. Sci. Res. Dev.*, vol. 4, no. 06, pp. 931–935, 2016.

[9] Z. Xianfei, F. Yunwen, X. Xiaofeng, and Q. Qiang, “Evaluate the crashworthiness response of an aircraft fuselage section with luggage contained in the cargo hold,” *Int. J. Crashworthiness*, vol. 22, no. 4, pp. 347–364, 2017, doi: 10.1080/13588265.2016.1258957.

[10] S. Krenk and J. Hogsberg, *Statics and Mechanics of Structures*. Dordrecht: Springer, 2013.

[11] “MSC Nastran 2021.4 Linear Static Analysis User’s Guide.” Hexagon, 2021.

[12] *MIL-HDBK-5J Metallic Materials and Elements for Aerospace Vehicles Structures*. U.S. Department of Defense, 2003.

Draft

ORIGINALITY REPORT

4%	2%	1%	2%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

PRIMARY SOURCES

1	ejournals.itda.ac.id Internet Source	1%
2	Submitted to Coventry University Student Paper	<1%
3	www.ijsrd.com Internet Source	<1%
4	Submitted to Manchester Metropolitan University Student Paper	<1%
5	Submitted to University of Warwick Student Paper	<1%
6	Submitted to Glasgow Caledonian University Student Paper	<1%
7	Pfeiffer, H, M Patitsa, I Pitropakis, and M Wevers. "Structural Health Monitoring in an operational airliner", Emerging Technologies in Non-Destructive Testing V, 2012. Publication	<1%

8	Submitted to Universiti Pertahanan Nasional Malaysia Student Paper	<1 %
9	Zibo Jin, Daochun Li, Jinwu Xiang. "Crash simulation of fuselage section in the rebound process and the secondary-impact process", Aircraft Engineering and Aerospace Technology, 2020 Publication	<1 %
10	dspace.vutbr.cz Internet Source	<1 %

Exclude quotes	On	Exclude matches	Off
Exclude bibliography	On		