

Finite Element Method Analysis on VTP to Determine Initial Crack Location at Nxxx Aircraft using MPC to Distribute Loads

¹Shofwan Bahar, ^{2*}Eflita Yohana, ³Elroy Efodiarlo Umbu Lolo, ⁴Susilo Adi Widyanto

^{1,2,3,4}Department of Mechanical Engineering, Diponegoro University, Semarang, Indonesia

*Corresponding Author's E-mail: efnan2003@gmail.com

Abstract - In the field of aviation, every aircraft component plays a crucial role in ensuring safety, stability, and optimal performance during flight. One of these components, which plays a critical role, is the Vertical Tail Plane (VTP) or vertical stabilizer. The VTP not only provides vertical stability but also functions as a vital control tool for pilots to maintain the flight path and respond to external changes. Inspection and thickness improvements, along with the application of Multi-Point Constraint (MPC) to the VTP structure, are essential steps in the manufacturing process to prevent production errors that could cause damage to the aircraft. This study aims to determine the location of the initial crack in the VTP structure, which will then be used to calculate the crack propagation rate. If the crack propagation rate is fast, the geometry of the structure needs to be modified. The method used in this analysis is the Finite Element Method (FEM), implemented using MSC PATRAN software. By using FEM, thickness improvements and MPC application can be performed quickly and efficiently, ensuring that the analysis produces a safe and high-quality VTP component.

Keywords: Efficient, Initial crack, Manufacturing, Finite element method, Multi-point constraint, Vertical tail plane.

I. INTRODUCTION

In the field of aviation, every aircraft component plays a crucial role in ensuring safety, stability, and optimal performance during flight. One of the components that holds a critical role is the Vertical Tail Plane (VTP) or vertical stabilizer. The VTP not only provides vertical stability but also functions as a vital control tool for pilots to maintain the flight path and respond to external changes [1].

The Finite Element Method (FEM), commonly referred to as the finite element method, is a method proven to be reliable in analyzing stress occurring in structures. The basic concept of this method involves dividing objects into smaller shapes, with each shape retaining the properties of the original material. This method is widely used to solve technical and

structural problems (buildings, aircraft, bridges, ships, cars, etc.). The technical problems that can be addressed using the finite element method are classified into two categories: structural analysis problems and non-structural issues [2].

This method is able to solve complex mechanical solid-body structural problems to produce solutions such as stress, strain, deflection, and fatigue life. The advantage of the FEM is its ability to minimize time and cost. Moreover, this method can also be used to evaluate a structure's performance before manufacturing an actual prototype by using MSC PATRAN and NASTRAN software [3].

Developing a VTP requires an in-depth understanding of its structural and aerodynamic aspects. In this regard, the FEM approach plays a significant role. The commonly used software for FEM applications is MSC PATRAN and NASTRAN, with the former focused on structural editing and the latter specialized for structural analysis. NASTRAN, derived from "NASA Structural Analysis," is software that has been widely used in the aviation industry to analyze the structural performance of various aircraft components, including the VTP. By dividing the structure into smaller finite elements and applying finite element analysis, NASTRAN provides insights into how the VTP responds to loads during flight, such as aerodynamic loads and inertia forces. The results of NASTRAN analysis help identify critical points that require special attention so that these points can provide information on where an initial crack might occur in the VTP.

For the finite element modeling of the VTP, MSC PATRAN serves as an auxiliary tool. PATRAN is analysis software that allows engineers and designers to model and/or edit the finite element structure of the VTP based on the results of NASTRAN analysis. This software facilitates the implementation of structural changes while considering aesthetic and functional aspects [4].

This study aims to explain the process and steps for structural editing using PATRAN and structural analysis using

NASTRAN to determine the initial crack and locate stress concentration points in the VTP structure due to operational loads.

II. MATERIAL AND METHOD

2.1 Aircraft NXXX

The NXXX aircraft is a turboprop aircraft with a passenger capacity of approximately 19, as shown in Figure 1. This aircraft is designed to serve short to medium-haul flights and can be used for various purposes, including passenger transportation, troop transportation, cargo or logistics, medical evacuation, surveillance and patrol, search and rescue, among others. The aircraft has the following general specifications (Table 1).

Table 1: General specifications of NXXX

Length	16.49 m
Wingspan	19.50 m
Takeoff distance	435 m
Landing distance	509 m
Payload capacity	2,313 kg
Maximum takeoff weight	7,030 kg
Engine	2 Pratt & Whitney PT6A-42 engines

Below is an image of the VTP (Vertical Tail Plane) of the NXXX aircraft along with the components that form the VTP structure. The VTP structure is shown in Figure 2.

The VTP consists of several components that support the VTP structure to withstand the loads experienced by the aircraft during operation, as shown in Figure 3. Some of the components present in the VTP are as follows: (a) Rib (Functions to maintain the shape of the VTP so it does not deform when subjected to operational loads. There are 12 ribs in the VTP of the NXXX aircraft), (b) Stringer (Functions to enhance rigidity and resist buckling loads on the skin), (c) Spar (Functions to withstand bending loads on the VTP. The spar consists of two parts: the spar web and the spar cap), (d) Fitting (Functions to connect the horizontal tail plane (HTP) to the VTP and to connect the VTP to the fuselage of the NXXX aircraft), (e) Skin (Functions to form the physical shape of the VTP and resist tension and compression loads experienced by the VTP). In the simulation, the first part to experience loads during aircraft operation is assigned an MPC, which functions to distribute forces/loads.

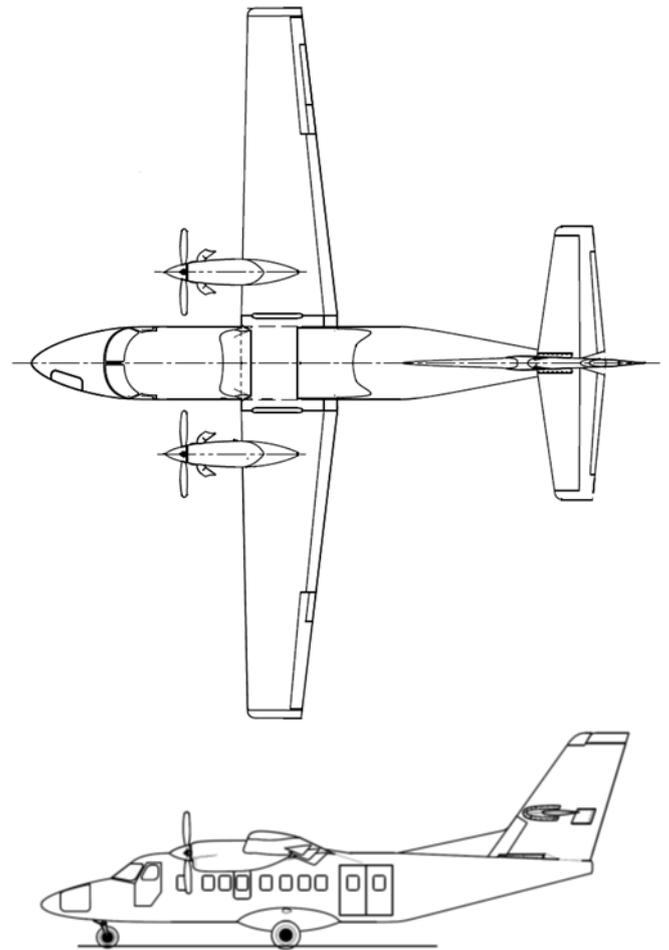


Figure 1: Top and side view of the NXXX aircraft

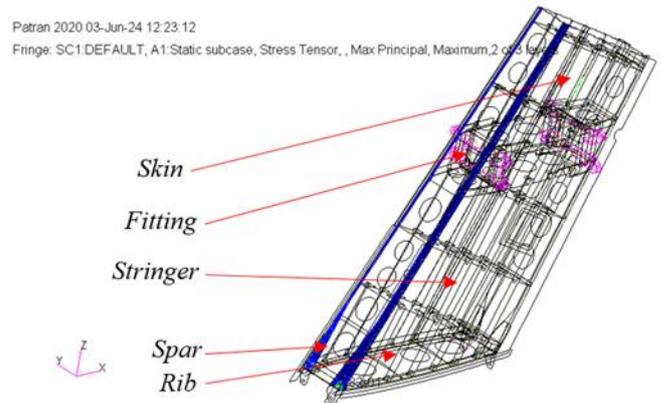
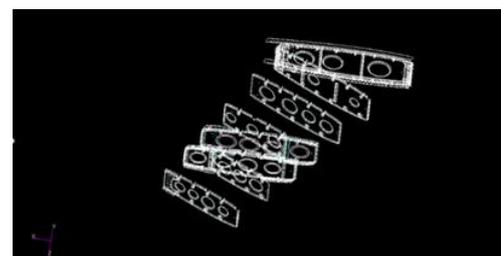


Figure 2: Structure of VTP



(a)

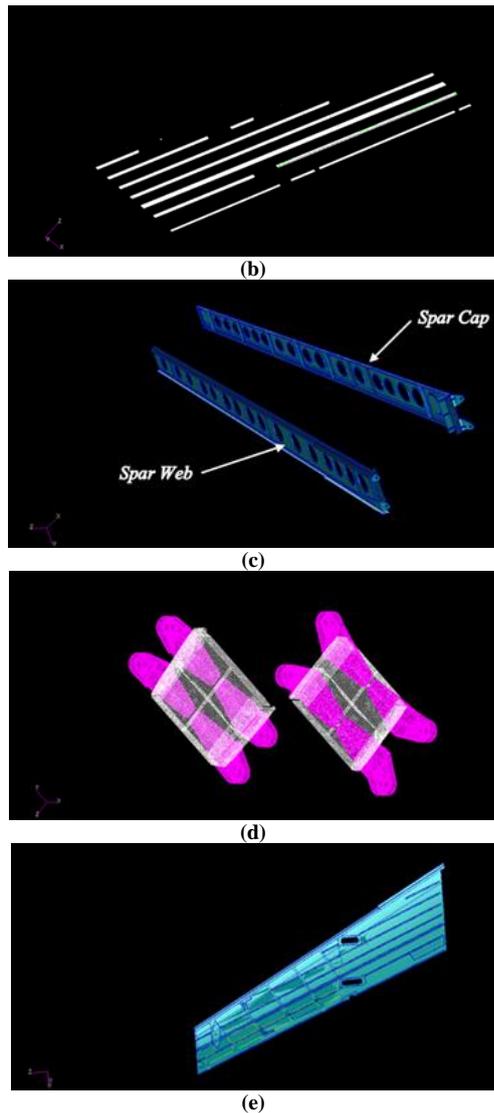


Figure 3: VTP Components: (a) rib, (b) stringer, (c) spar, (d) fitting, (e) skin

The loads experienced by the VTP during operation are divided into two categories. The first is aerodynamic loads, which result from air pressure distribution on the VTP. The second is inertial loads caused by the inertia of the VTP. Inertial loads arise due to the mass of the VTP and the varying acceleration of the aircraft during flight [5].

Multi Point Constraint or MPC functions to distribute loads so that concentrated forces are not produced on the VTP structure. The type of MPC used in this analysis is MPC RBE 3. The reason for using MPC RBE 3 is to distribute loads proportionally, thus avoiding the creation of concentrated forces [6].

2.2 A1 2024-T3

Aluminum is a chemical element of Group IIIA found in the periodic table, with atomic number 13 and atomic weight

of 26.98 grams per mole. This material has a low specific gravity and excellent corrosion resistance. A1 2024-T3 is a type of aluminum alloy widely used in the aviation industry due to its durability, lightweight, and suitability for VTP structures [7].

2.3 MSC PATRAN

MSC PATRAN is software used for modeling, simulation, and finite element analysis, particularly in engineering and design fields. PATRAN is software developed by Siemens Digital Industries Software, formerly known as MSC Software. PATRAN is often used as software for finite element analysis. The primary function of PATRAN is to facilitate structure modeling, load analysis, model analysis, and various other aspects of technical simulations and analyses. Within MSC PATRAN, users can create small elements in structures, assign properties to materials and structures (such as thickness, inertia, surface area, etc.), and apply loads and boundary conditions. The boundary conditions in structures may include fixed supports, hinges, rolling conditions, etc. [8].

2.4 MSC NASTRAN

MSC NASTRAN stands for "NASA Structural Analysis". This is structural analysis software originally developed by NASA in the 1960s. NASTRAN is one of the most well-known finite element analysis software and is widely used across various industries, especially in engineering and manufacturing fields. NASTRAN is utilized for conducting extensive structural analyses, including stress, deformation, vibration, dynamics, stability, and many other aspects of structural performance. NASTRAN operates by dividing structures into smaller elements (finite elements) and performing numerical analyses. In summary, NASTRAN is recognized as highly robust and influential structural analysis software, often employed to predict and respond to real-world structural performance issues [9].

III. RESULTS AND DISCUSSIONS

The simulation results using MSC NASTRAN software produced several images representing displacement and stress on parts of the VTP structure of the NXXX aircraft.

3.1 Displacement

Displacement refers to the positional shift occurring in the VTP when subjected to operational loads. The displacement simulation results are shown in Figure 4.

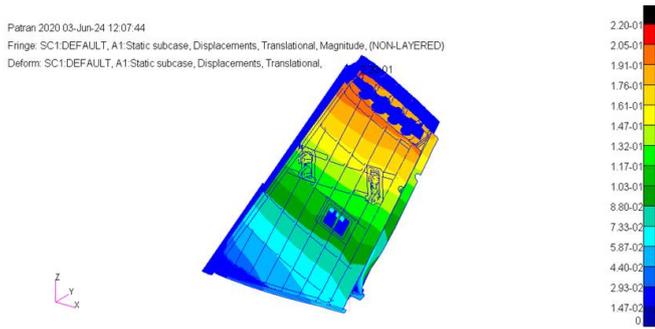


Figure 4: Displacement Simulation Results

From the simulation results, the largest deformation caused by operational loads occurs at the upper tip of the VTP. This is because when the aircraft moves through the air, wind forces acting on the VTP surface generate aerodynamic pressure. This pressure tends to be higher on the upper part of the VTP because, the farther from the base (where the VTP connects to the aircraft body), the greater the moment forces generated by the pressure. As a result, the upper tip of the VTP exhibits greater deflection or displacement compared to the lower parts closer to the aircraft body [10].

3.2 Stress on Rear Spar Cap

The stress occurring on the rear spar cap is depicted in Figure 5, and the location of the rear spar cap in the VTP structure is shown in Figure 6.

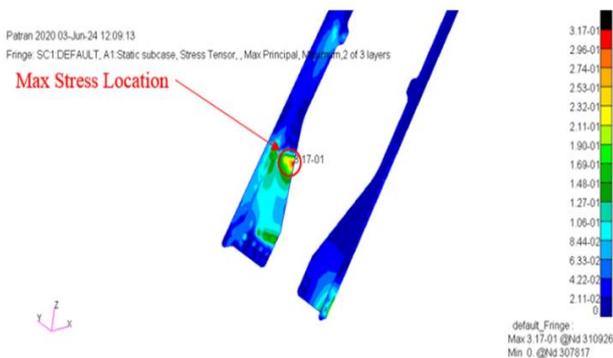


Figure 5: Stress Simulation Results on Rear Spar Cap

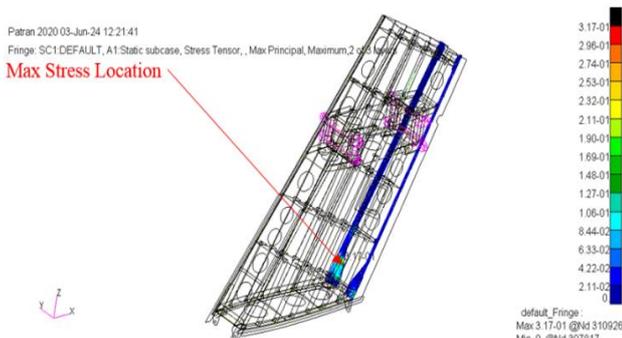


Figure 6: Rear Spar Cap Location on VTP Structure

From the simulation results, the highest stress is indicated in the red-colored area, while other regions show distributed stress effectively along the rear spar cap. The stress concentration in the red-colored area is caused by the fastener in the fitting structure, which connects the VTP to the aircraft body.

3.3 Stress on the Front Spar Cap

The stress that occurs on the front spar cap is shown in Figure 7, and the location of the front spar cap on the VTP structure is shown in Figure 8.

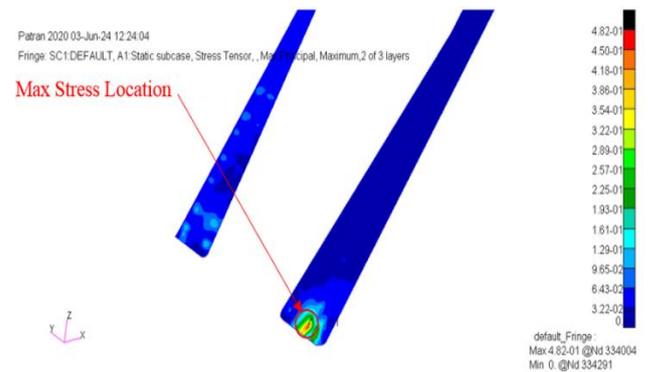


Figure 7: Simulation Result of Stress on the Front Spar Cap

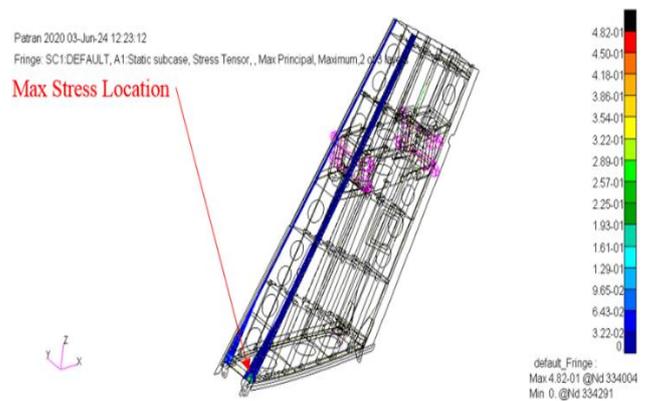


Figure 8: Location of the Front Spar Cap on the VTP Structure

Based on the simulation results, the highest stress is located in the fitting section that connects the VTP to the aircraft fuselage. In the fitting structure, there are fastener components that function as connectors in the aircraft structure. However, the fasteners also contribute to the occurrence of stress concentration in the VTP structure.

3.4 Stress on the Rear Spar Web

The stress that occurs on the rear spar web is shown in Figure 9, and the location of the rear spar web on the VTP structure is shown in Figure 10.

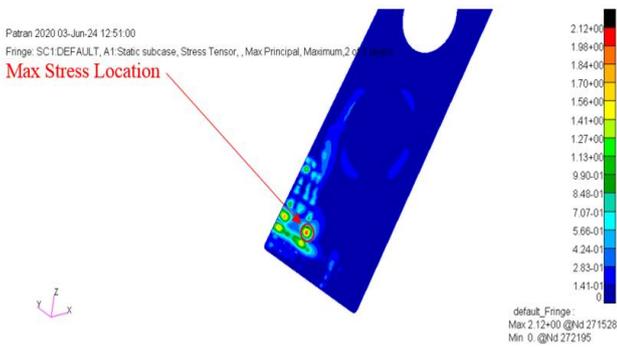


Figure 9: Simulation Result of Stress on the Rear Spar Web

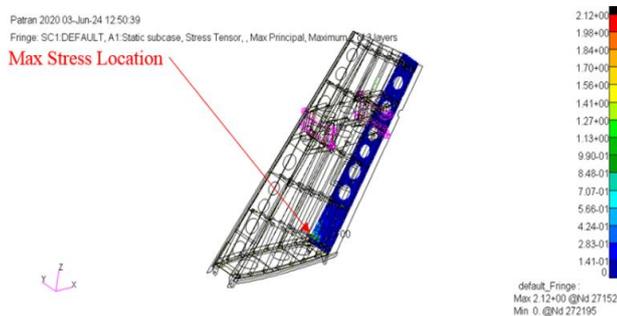


Figure 10: Location of the Rear Spar Web on the VTP Structure

Based on the simulation, the highest stress point is located at the fastener found on one of the fittings in the rear spar web structure. One of the causes of stress concentration in the rear spar web structure is the geometry and loading conditions experienced by that part of the structure.

3.5 Stress on the Rib

The VTP structure contains 12 ribs that function to maintain the shape of the VTP so that it does not deform under operational loads, as shown in Figure 11.

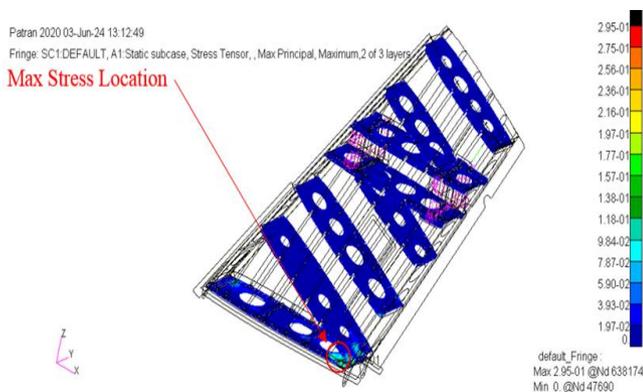


Figure 11: Simulation Result of Stress on the Rib

Based on the simulation, there is a stress concentration observed in rib 1, where the highest stress point is located on the fitting in rib 1. This rib functions to connect the VTP to the aircraft fuselage, as shown in Figure 12.

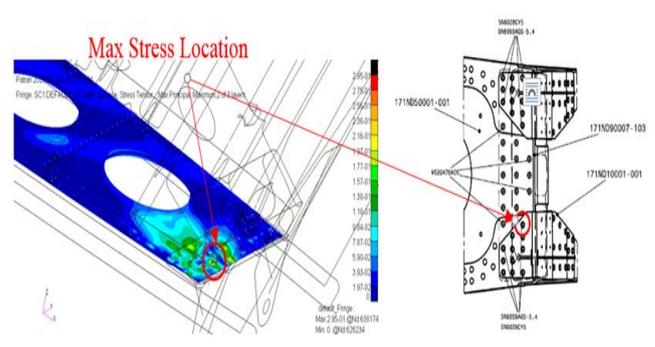


Figure 12: Simulation Result of Stress on Rib 1 and Fastener on Rib 1

3.5 Stress on the HTP Fitting Support

The stress on the HTP Fitting Support is shown in Figure 13, and the location of the HTP fitting support on the VTP structure is shown in Figure 14.

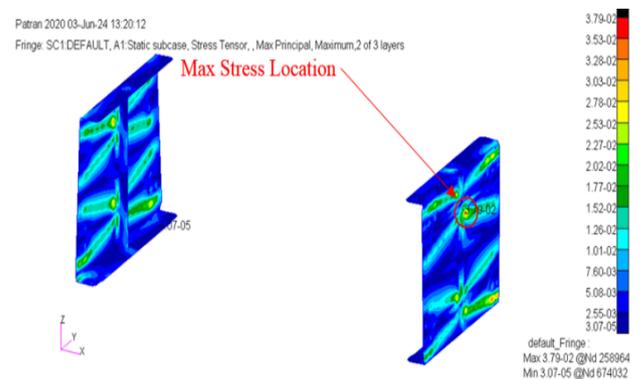


Figure 13: Simulation Result of Stress on the HTP Fitting Support

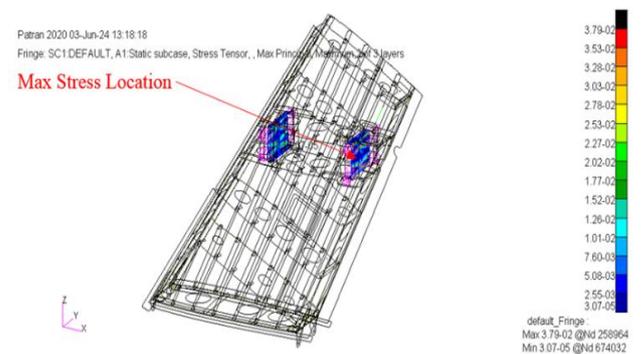


Figure 14: Location of the HTP Fitting Support on the VTP Structure

Based on the simulation results, the point with the highest stress is located at the fastener situated on the HTP fitting support structure. This structure functions to maintain the shape of the fitting to prevent deformation at the joint that connects the VTP and the HTP.

IV. CONCLUSION

From this study, the authors can draw several conclusions. In conducting the analysis using MSC PATRAN

software, high precision is required in performing thickness checks, thickness corrections, and the application of MPC (Multi Point Constraint) to avoid errors in the manufacturing process, particularly in the VTP (Vertical Tail Plane) section of the aircraft. From the results of the stress simulation, where the loads applied to the VTP structure of the aircraft are operational loads, several stress concentrations were identified in specific areas of the VTP structure. These stresses are caused by operational loads, and from these stresses, the crack propagation rate can be calculated. If the propagation rate is rapid, modifications to the geometry of the section should be made.

REFERENCES

- [1] Ciliberti, D., Vecchia, P. D., Nicolosi, F., & Marco, A. D. (2017). Aircraft directional stability and vertical tail design: A review of semi-empirical methods. *Progress in Aerospace Sciences*, 140-172.
- [2] Suvin, V., Ooi, E. T., Song, C., & Natarajan, S. (2024). Adaptive scaled boundary finite element method for hydrogen assisted cracking with phase field model. *Theoretical and Applied Fracture Mechanics*.
- [3] Su, P., Chen, J., Yang, R., & Xiang, J. (2024). A New Isogeometric Finite Element Method for Analyzing Structures. *CMES - Computer Modeling in Engineering and Sciences*, 1883-1905.
- [4] Rohini, D., AmarKarthik, A., Abinaya, R., Mathan, A., Midhun, S., & Dhushyanth, D. (2022). Buckling analysis of a commercial aircraft wing box and its structural components using Nastran patran. *Materials Today: Proceedings*, 895-901.
- [5] Sung, D., Hong, S., & Lee, J. (2023). Numerical analysis of the rail surface crack propagation under rail uplift force: A parametric study on initial crack geometry. *Engineering Failure Analysis*, 153.
- [6] Wu, H., Guo, Y., Xu, H., Dong, H., Li, Z., & Ke, Y. (2024). Investigation of angular deformation control for T-joint rib using multi-point constraint method. *Journal of Constructional Steel Research*, 213.
- [7] Daoud, M., Chatelain, J., & Bouzid, A. (2017). Effect of rake angle-based Johnson-Cook material constants on the prediction of residual stresses and temperatures induced in Al2024-T3 machining. *International Journal of Mechanical Sciences*, 392-404.
- [8] Zhang, S. R., Wang, L. J., Du, X. J., Jin, H. L., Zhang, Y. H., Sun, X. Y., . . . Zhang, K. (2021). Research of Fatigue Life Prediction for Optimization of the Gearbox Design Based on MSC.Patran. *Strength of Materials*, 636-645.
- [9] Kusni, M., Widiramdhani, M., & Hadi, B. K. (2023). Composite wing structure optimization with static and buckling constraints using MSC-Nastran. *In AIP Conference Proceedings*, Vol. 2680, No. 1.
- [10] Han, L., He, X., Ning, Y., Zhang, Y., & Zhou, Y. (2024). An aircraft structural risk assessment method considering fatigue crack propagation based on fatigue damage diagnosis and prognosis. *International Journal of Fatigue*.

Citation of this Article:

Shofwan Bahar, Eflita Yohana, Elroy Efodiarlo Umbu Lolo, & Susilo Adi Widyanto. (2025). Finite Element Method Analysis on VTP to Determine Initial Crack Location at Nxxx Aircraft using MPC to Distribute Loads. *International Research Journal of Innovations in Engineering and Technology - IRJIET*, 9(5), 417-422. Article DOI <https://doi.org/10.47001/IRJIET/2025.905046>
