

ANALISIS RESPON STATIS DAN DINAMIS RAMP DOOR DENGAN MATERIAL CORE *SANDWICH PLATE SYSTEM* MENGUNAKAN *METODE FINITE ELEMENT*

Revanza Naufal Rianto

ABSTRAK

Salah satu komponen penting pada kapal ro-ro yaitu pintu ramp door, yang berfungsi sebagai jalur keluar masuknya kendaraan selama proses bongkar muat. Struktur ramp door dapat menahan beban secara aman sesuai dengan persyaratan klasifikasi. Metode penelitian yang digunakan pada penelitian ini berbasis Finite Element Method (FEM) dengan menggunakan software ansys mechanical. Model ramp door pada penelitian ini dilakukan dengan variasi pada ketebalan material core yang ada pada sandwich panel sebesar 20 mm (Model A), 30 mm (Model B), dan 40 mm (Model C). Pembebanan yang digunakan berupa kendaraan sebesar 26 ton, kemudian ditingkatkan sebesar 5% hingga 25% dari beban awal. Analisis yang diamati mencakup tegangan dan deformasi maksimum, natural frequency, dan safety factor yang kemudian akan dievaluasi berdasarkan rules Biro Klasifikasi Indonesia. Hasil penelitian pada analisis statis, model aktual mendapatkan nilai tegangan maksimum dibawah nilai tegangan izin. Lalu pada ketiga model variasi A, B, dan C mendapatkan tegangan maksimum dalam batas aman dan menghasilkan deformasi yang lebih kecil dibandingkan dengan model aktual, serta safety factor berkisar 2,59 hingga 3,11. Hasil analisis dinamis menunjukkan peningkatan nilai natural frequency dengan baik. Berdasarkan semua hasil yang didapat membuktikan bahwa penerapan sandwich panel dapat meningkatkan kekuatan, kekakuan dan karakteristik dinamis struktur ramp door.

Kata Kunci : Ramp Door, Finite Element Analysis, Sandwich Panel, Ansys Mechanical, Analisis Statis, Analisis Dinamis, Rules BKI.

ANALYSIS OF THE STATIC AND DYNAMIC RESPONSES OF A RAMP DOOR WITH SANDWICH PLATE CORE SYSTEM USING THE FINITE ELEMENT METHOD

Revanza Naufal Rianto

ABSTRACT

One of the key components of a ro – ro ship is the ramp door, which serves as the entry and exit point for vehicles during loading and unloading. The ramp door structure is designed to safely withstand loads in accordance with classification requirements. The research method used in this study is based on the Finite Element Method (FEM) using ansys mechanical software. The ramp door models in this study were created with variations in the thickness of the core material in the sandwich panel 20 mm (model A), 30 mm (model B), and 40 mm (model C). The loading test involved a 26 metric ton vehicle , with the load subsequently increased by 5% to 25% of the initial load. The analysis examined maximum stress and deformation, natural frequency, and safety factor, which were the evaluated based on the rules of rhe Indonesian Classification Bureau. The results of the static analysis showed that the actual model had a maximum stress value below the allowable stress limit. Furthermodre, the three variant models A, B, C all exhibited maximum stress within safe limits and produced smaller deformations compared to the actual model, with safety factors ranging from 2,59 to 3,11. The results of the dynamic analysis demonstrated a significant increase in natural frequency. Based on all the results obtained, it is evident that the application of sandwich panel can enhance the strength, stiffness, and dynamic characteristics of the ramp door structure.

Keywords : Ramp door, Finite Element Method (FEM), Sandwich Panel, Ansys Mechanical, Static Analysis, Dynamic Analysis, BKI Rules.