This study investigates the structural behavior of the forecastle deck in the crane installation area of a Multi-Cat vessel by means of finite element analysis in accordance with ABS requirements. The analysis focuses on the stress distribution and structural response of the deck and its supporting members under crane loading, with particular attention to the influence of crane slewing position on the critical stress state. The results are intended to support structural verification of the crane foundation region and to provide a practical reference for deck reinforcement design in similar working vessels.

The contribution of this work can be summarized as follows. First, a three-dimensional finite element model is established for the forecastle deck together with its surrounding supporting structure, so that the local load transfer mechanism in the crane foundation region can be evaluated in detail. Second, the crane loading is examined under a series of slewing angles, which makes it possible to identify the most unfavorable operating position rather than relying on a single loading direction. Third, the calculated stresses are assessed against an ABS-based allowable stress criterion, allowing the numerical results to be directly linked to practical structural acceptance in ship design.

2 Research Methods

2.1 Vessel particulars

The vessel analyzed in this study is a Multi-Cat boat classified by ACS and operating under the Iranian flag, with an overall length of 19 m, a moulded breadth of 7.20 m, a moulded depth of 2.20 m, a displacement of 160 tonnes, and a midship draft of 1.70 m (Table 1).

Table 1 Principal particulars of the multi-cat vessel

Ships Name	MULTI CAT BOAT
Classification	ACS
Flag	Iran
GROSS/NET Tonnage	---
Length overall	19 m
Length Between Perpendiculars	18 m
Breadth (moulded)	7.20 m
Depth @ MID (moulded)	2.20 m
DISPLACEMENT	160 tonnes
Draft (mid)	1.70 m
Class Notation	SPECIAL SERVICE, FISH FARM SUPPORT CRAFT
Navigation area	INTERNAL & TERRITORIAL WATERS

2.2 Abaqus software

The software used in this study is SIMULIA Abaqus FEA, which provides comprehensive capabilities for modeling, analysis, and simulation based on the Finite Element Method (FEM).

2.3 System of units

A consistent unit system was adopted throughout the numerical analysis, including millimeters for length, tons for mass, ton/mm³ for density, newtons for force, and MPa (N/mm²) for stress and pressure (Table 2).

Table 2 System of units used in the analysis

Unit	Parameter
mm	Length
Ton	Mass
ton/mm3	Density
N	Force
MPa (N/mm2)	Tension (Pressure)