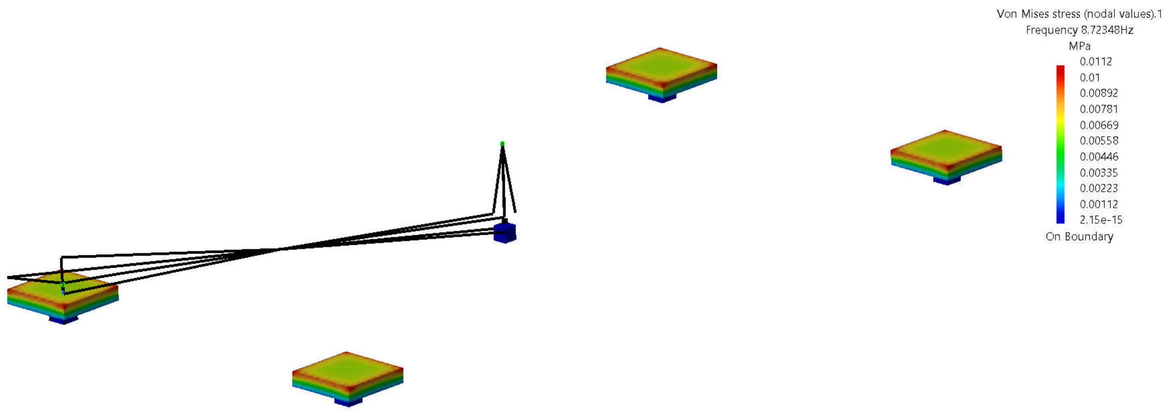

A Chalmers University of Technology Master’s Degree project report

Design Optimization and Dynamic Analysis of engine mounts and flanges of a marine drivetrain

Master’s thesis in Applied Mechanics

Brijesh Niranjan

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES

CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026
www.chalmers.se

Master's Thesis in Applied Mechanics

Design Optimization and Dynamic Analysis of engine mounts and flanges of a marine drivetrain

A Master of Science Thesis in Applied Mechanics (MPAME)

Brijesh Niranjan

Department of Mechanics and Maritime Sciences
Division of Dynamics
Chalmers University of Technology
Gothenburg, Sweden 2026

Design Optimization and Dynamic Analysis of engine mounts and flanges of a marine drivetrain
Master of Science Thesis in Applied Mechanics (MPAME)
Brijesh Niranjan

Supervisor: Badr Jalabi, Scania CV, Research and Development.
Examiner: Petri Piiroinen, Mechanics and Maritime Sciences.

Master's Thesis 2026
Department of Mechanics and Maritime Sciences
Chalmers University of Technology
SE-412 96 Gothenburg
Sweden
Telephone +46 31 772 1000

Cover: Visualization of base model of a marine drivetrain system showing its behaviour in resonance

Typeset in LaTeX
Gothenburg, Sweden 2026

Design Optimization and Dynamic Analysis of engine mounts and flanges of a marine drivetrain
Master of Science Thesis in Applied Mechanics (MPAME)

Brijesh Niranjan
Department of Mechanics and Maritime Sciences
Division of Dynamics
Chalmers University of Technology

Abstract

In reality, marine vessels undergo various dynamic loads while breezing through the waters and crashes into the waves. This leads to g-forces experienced by the marine vessel. A marine vessel consists of the main engine, electric machine, and the gearbox coupled by flanges and their respective brackets mounted to the vessel by rubberized mounts. Due to varying loads, the brackets and the mounts experience forces on them and, in return, produce reaction forces. In addition to this, the weights of each component contribute to the moment of the system at the flanges. An analytical method was developed in [1] to estimate these forces on the mounts and the moments on the flanges. This thesis focuses on optimizing the design to minimize these loads experienced and to conduct a dynamic analysis of the whole system.

Keywords: Marine vessels, G-forces, Flanges, Mounts, Dynamic Analysis, Optimizing

Preface

This report presents the outcome of our master's thesis project carried out at the Scania CV Power Solutions during the spring of 2026.

Acknowledgements

Firstly, I would like to thank my parents Niranjan Reddy and Sujatha.G.O, my grandparents Anji Reddy and Seshamma and my brother Nutan for their constant support throughout my MSc Programme. I would not have completed my thesis without their constant support.

Next, I would like to thank my manager Guirguis Diana for giving me the opportunity to showcase my knowledge, Badr Jalabi (Supervisor) who has been more like a friend to me in this project has guided me with his knowledge and helped me in completing my thesis and Erik Andersson for his tips and expertise on this project.

Lastly, I would like to thank my friends who has always been very supportive academically and emotionally motivating me to always do better than yesterday. You all played a big part over the 2 years and I am grateful to all of them.

Brijesh Niranjan, Gothenburg, August 2026

Declaration

The work is originally conducted by the author who is fully responsible for the results obtained. This is author's own work. There has been the use of AI for enhancing the language, grammar, and phrasing of the sentences. However AI tools were not used to analyse the results or any data analysis. The author is responsible for the accuracy and authenticity of this thesis work.

List of Acronyms

Below is the list of acronyms that have been used throughout this thesis listed in alphabetical order.

3D	Three-dimensional
CATIA	Computer Aided Three-dimensional Interactive Application
COG	Centre of Gravity
EOM	Equation of Motion
FEM	Finite Element Method
LH	Left-hand
MAPE	Mean Absolute Percentage Error
MSc	Master of Science
RH	Right-hand
SQP	Sequential Quadratic Programming

Nomenclature

Below is the nomenclature of indices, parameters, and variables that have been used throughout this thesis.

Indices

i	Index for the support location
j	Index for the drivetrain component or flange

Parameters

L_1	Length of the first block
L_2	Length of the second block
L_3	Length of the third block
m	Total mass of the drivetrain system
k_b	Bracket stiffness
k_m	Mount stiffness
k_s	Equivalent stiffness of the support
$k_{LH,i}$	Effective stiffness of the left-hand support at location i
$k_{RH,i}$	Effective stiffness of the right-hand support at location i
n	Number of mounts
g	Gravitational acceleration

Variables

x_i	Longitudinal position of mount i
$\mathbf{x}$	Design-variable vector
x_0	Initial guess of the design-variable vector

x_{opt}	Optimized mount positions
F	Reaction force
F_i	Reaction force at mount i
δ	Displacement of the mount
M	Bending moment
ω_n	Natural circular frequency
f_n	Natural frequency
ω	Angular frequency
$\mathbf{M}$	Mass matrix
$\mathbf{K}$	Stiffness matrix
ϕ	Mode shape
$f(\mathbf{x})$	Objective function
$g(\mathbf{x})$	Inequality constraint function
$h(\mathbf{x})$	Equality constraint function

Contents

List of Figures

List of Tables

1 Introduction

1.1 Background

Marine drivetrain consists of three components such as the main engine, electric machine and a gearbox. The components are coupled to each other by "flanges" that are bolted and placed on the vessel floor by supports. The supports consists of a "bracket" and a "mount". The bracket is an extension of the component itself made of the same material, such as steel and are placed on top of a vibrational isolators such as rubberized mounts which helps to nullify any vibrations caused during operation and to allow the engine to perform smoothly. The marine mounts have a compression limit (this is referred to as "bottoming-out" limit) and loading beyond this limit will lead to failure of the structural integrity. Marine vessels are known to undergo various types of loading when traversing through water, one of which are the g-forces caused by crashing into waves. These g-forces can cause the rubberized mounts that support the drivetrain to "bottom out" and can also impact the bending moment experienced by the flanges at the junctions between components.

Consider application engineer from Scania on field with a client who needs to check and validate the design and ensure the industrial safety standards are matched and all the protocols are met. To overcome this, an analytical method was introduced in [1] to determine the reaction force on each mounts and to evaluate the moments on the flanges. These calculations are parametrized and defined in the coding software called GNU Octave. This script allows one to input the values such as mount position, weight of the components, centre of gravity and all other necessary physical parameters. The output of this script is used to determine if there are any mounts that bottoms out and to locate the position of the failed mount. In addition, the script obtains the reaction forces and the moments at the mounts and flanges, respectively.

1.2 Societal, Ethical and Ecological impact

Scania is known for its sustainable solutions towards transport problems. This research helps in getting a relation between power consumption and the safety of the mounting system. The system's ability to handle dynamic loads directly impacts the safety of the passengers, crew members, and technicians. A failure in such a system leads to hazardous impacts such as mechanical damage and an unsafe working environment. The mounting systems show economic and environmental impacts.

The longer the system sustains, the less the replacement of the component, leading to saving of the material and the cost. The vibration of the system directly leads to power loss. The decrease in fuel efficiency impacts the environment as there is an increase in pollutants released. The present work does not include a complete environmental lifecycle assessment, but the ecological implications will guide result interpretation and establish boundaries for the current modelling method.

1.3 Purpose

Scania Power Solutions guides their customers to consult Trelleborg Anti-vibration Solutions for mounting solutions for industrial and marine drivetrain. Over the years Scania Power solutions have very good data of how the drivetrain system behaves under static loading conditions, which are backed by enormous amount of experimental data, on-field experience, and analytical methods developed in Scania Research and Development.

After a lot of interdepartmental collaboration, it has come to an understanding that there is lack of research on how the drivetrain system behaves under dynamic loading condition. This is crucial for the application engineer at Power solutions to be able to suggest the customer on the type of mounts to be used to ensure safety, durability and compliance with the industrial standard. Without a reliable dynamic prediction system, engineers need to rely on assumptions which leads to over-designing (leading to unnecessary cost and stiffness) or under designing (leading to safety concerns, mount bottoming out and high stresses in flanges in turn reducing the system lifespan).

The purpose of the project is twofold. First, the project aims to develop a method to reduce the reaction forces experienced by the mounts and the moments at the flanges. Reducing these loads and moments improves the drivetrain safety and durability. Second, the project intends to develop a simulation method for evaluating the dynamic behaviour taking into consideration of the whole system consisting of the engine, e-machine, and the gearbox.

1.4 Goals

The objective of this project is to develop a method based on optimization, which would allow optimizing the load distribution in a marine drivetrain mounting system by changing the mount positions. With the change in mount distances, such that the loads are evenly distributed among the mounts, minimizing their overloading and increasing their durability.

Another objective of this project is the development of a methodology for simulation, which will allow determining the dynamic properties of the drivetrain system. This will include carrying out modal analysis. Moreover, the simulation approach will allow obtaining the highest values of the von Mises stresses over the entire range of operating frequencies.

1.5 Limitations

The constraints to be considered during the investigation are as follows:

- **Study on optimizers:** This work does not evaluate or rank different optimizers, nor analyse their relative performance. Only the selected optimizer is used within the methodology, and no claims are made about its superiority over other optimization techniques.
- **Material Selection:** Only commonly available marine-grade materials will be considered, such as rubbers, elastomers, and structural metals.
- **Design Parameters:** Mount stiffness, damping characteristics, flange size, and constraints will be limited to the range applicable to small to medium marine engine sizes.
- **Dimensional Constraints:** The design will be constrained by the available engine dimensions and the space available on the vessel.
- **Simplifications:** The engine will be assumed to behave as a rigid body. Mounts will be assumed to be linear elastic supports, although non-linear behaviour may be considered during simulations.

2 Theory

The purpose of this chapter is to establish the theoretical basis for the methods used in this thesis. The chapter first describes the stiffness characteristics of the drivetrain mounting system and the approach used to estimate its natural frequency. The theoretical formulation is then extended to the optimization problem, where the mount positions are treated as design variables and the reaction forces and flange moments are used to define the optimization objectives. The constraints governing the feasible mount positions are also introduced. Finally, the fundamental principles of dynamic analysis are presented to provide the basis for the CATIA V5 finite element simulations and subsequent evaluation of the drivetrain response.

These theoretical concepts form the basis for the analytical calculations, MATLAB optimization tool, and CATIA V5 simulation methodology developed and evaluated in the subsequent chapters.

2.1 Marine-drivetrain Support Systems in Marine Applications

A marine drivetrain consists of several heavy components such as the main engine, electric-machine and the gearbox. These components are bolted together, respectively, with their flanges and the respective component brackets are placed on a foundational support called as mounts. The combination of the bracket and mount is referred as support system. The mounts are used to absorb some of the vibrations produced during the operation and also allow some of the relative movements between the drivetrain and vessel structure.

Marine operations do not only deal with static loads caused by the weight of the components but also the dynamic loads from the vessel when traversing through water. These dynamic loads can be transmitted directly onto the mounts and the brackets, and can generate huge reaction forces and moments in the flanges. Excessive force can lead the mount to bottom out resulting in structural damage and misalignment. Therefore it is important to have a good understanding of the load distribution and to control it in the marine drivetrain system.

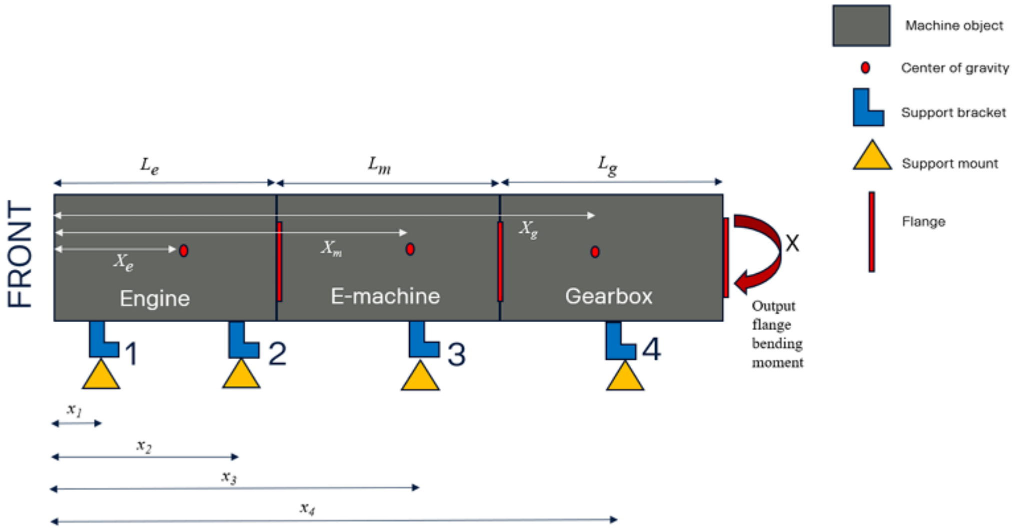

Figure 2.1: Schematic representation of the marine drivetrain setup. Adapted from [1].

The engine and electric machine shown in the schematic representation in Fig 2.1 correspond to the Scania DW-6 marine engine in Fig 2.2 and the Scania electric machine in Fig 2.3, respectively. The geometrical details of the Scania DW-6 marine engine used in the analysis are presented in Table 2.1.

Figure 2.2: Scania DW-6 Marine Engine.

Table 2.1: Physical Parameters of Scania DW-6 Engine.

Engine Parameter	Value
Mass(m_e)	1260 kg
Length (L_e)	1321.6 mm
Center of gravity from front of the engine(X_e)	818 mm

The geometrical details of Scania E-Machine are given below in Table 2.2

Figure 2.3: Scania E-Machine.

Table 2.2: Physical Parameters of Scania E-machine.

Engine Parameter	Value
Mass(m_m)	242 kg
Length (L_m)	495 mm
Center of gravity from front of the engine(X_e)	1613 mm

2.2 Rigid Body Representation of the Drivetrain

CATIA V5 is a computer-aided design and engineering software developed by Dassault Systems and is used in this work as the simulation platform. The software enables the three-dimensional modelling of the drivetrain assembly and its subsequent finite element analysis. In the present study, CATIA V5 is used to investigate

the dynamic behaviour of the drivetrain mounting system, with particular focus on the natural frequency, deformation, and stress response of the structure.
The CATIA model is divided into two configurations: a base model and a detailed three-dimensional (3D) model used for realistic visualization.The components in the drivetrain system are very stiff compared to the support system. For simplification, the drivetrain system can be assumed to be a rigid body supported by multiple elastic elements.

Under this assumption, the dynamic behaviour of the drivetrain is simplified to a single degree of freedom, representing the vertical motion of the drivetrain system.

2.3 Elastic Behaviour of Engine Mounts in Base Model

Component-Mount-Bracket systems are commonly modelled as mass-spring-spring systems where the two springs are in series in a 2D system. This setup is well backed by the research done in [1]. The relation between the applied load and the displacement can be described by the Hooke's Law

$$F = k_{eq}\delta, \tag{2.1}$$

where F is the reaction force, δ is the displacement of the mount, and k_{eq} is the equivalent stiffness of the mount and is given by

$$k_{eq} = \frac{n}{2} * \frac{k_{br}k_m}{k_{br} + k_m}, \tag{2.2}$$

where k_{br} and k_m represent the stiffness of the bracket and the mount, respectively, and n is the number of mounts in a 2D system. This equivalent stiffness determines how the load is distributed between the drivetrain and the supporting structure.

A design engineer must always keep the concept of natural frequency in mind. The natural frequency correspond to the vibration mode when no external force are applied. Natural frequency can be obtained by solving an eigenvalue problem. If external excitation occur at frequencies close to a natural frequency of the system we says the system is in resonance, leading to large amplitude vibrations. Evaluating the natural frequency is essential to analyse the dynamic performance of the system. Neglecting damping, the equation of motion can be expressed as

$$m\ddot{x} + k_{eq}x = 0, \tag{2.3}$$

where x is the displacement of the system and $\ddot{x}$ is the acceleration of the system. The corresponding natural frequency is

$$\omega_n = \sqrt{\frac{k_{eq}}{m}}, \tag{2.4}$$

and the natural frequency in Hertz is obtained from

$$f_n = \frac{\omega_n}{2\pi}, \tag{2.5}$$

The natural frequency provides an estimate of the vibration characteristics of the mounting system and serves as a useful reference before performing detailed finite element analyses.

2.4 Equivalent Stiffness System and Natural Frequency Estimation

The drivetrain assembly is supported by a number of elastic mounting points distributed on both the left-hand (LH) and right-hand (RH) sides of the structure. Each support location consists of a structural bracket connected to an elastic mount. Since both components deform under loading, the stiffness of an individual support is governed by the combined behaviour of the bracket and the mount.

2.4.1 Support Stiffness Formulation

At each support location, the bracket and mount are connected in series. For springs connected in series, the equivalent stiffness is given by

$$k_s = \frac{k_b k_m}{k_b + k_m} \tag{2.6}$$

where k_b is the bracket stiffness, k_m is the mount stiffness, and k_s is the equivalent stiffness of the support.
Since the drivetrain is supported by mounts on both the left-hand and right-hand sides, the stiffness of each support pair is evaluated separately as

$$k_{LH,i} = \frac{k_{b,LH,i} k_{m,i}}{k_{b,LH,i} + k_{m,i}}, \tag{2.7}$$

and

$$k_{RH,i} = \frac{k_{b,RH,i} k_{m,i}}{k_{b,RH,i} + k_{m,i}}, \tag{2.8}$$

where i denotes the support location along the marine drivetrain.
The terms $k_{LH,i}$ and $k_{RH,i}$ represent the effective stiffness of the left-hand and right-hand supports, respectively. This formulation accounts for the flexibility of both the mounting brackets and the rubber mounts.

2.4.2 Equivalent System Stiffness

The drivetrain is assumed to behave as a rigid body supported by multiple elastic supports. Since all supports act in parallel in the vertical direction, the total vertical stiffness of the mounting system is obtained by adding the stiffness contributions from all support locations, so that

$$k_{eq,system} = \sum_{i=1}^{n} k_{LH,i} + \sum_{i=1}^{n} k_{RH,i}, \tag{2.9}$$

where n is the number of mount locations along the drivetrain.
The equivalent stiffness represents the overall resistance of the mounting system to vertical deformation and forms the basis for estimating the static and dynamic characteristics of the drivetrain.

2.4.3 Natural Frequency and Static Deflection of the Baseline Model

The baseline model assumes that the drivetrain is supported by four identical mounts acting in parallel. The equivalent stiffness of the support system is therefore given by

$$k_{eq,system} = n * k \tag{2.10}$$

where k is the stiffness of a single mount and n is the number of mounts.
The natural circular frequency of the system is calculated as

$$\omega_n = \sqrt{\frac{k_{eq,system}}{m}}, \tag{2.11}$$

where m is the total supported mass.
The corresponding natural frequency in Hertz is obtained from

$$f_n = \frac{\omega_n}{2\pi} \tag{2.12}$$

2.5 Optimization of Mount Locations

Due to the different masses of the components in the system the reaction forces at mounts vary with the positions and stiffness of each mounts. By adjusting the position of the mounts, one can alter the load path through the system and by doing so reduces the reactions forces and bending moments at the critical locations such as mounts and flanges.

Optimization methods provide a systematic approach for identifying the best configuration of mount positions. In general an optimization problem can be expressed as

$$\begin{aligned} \min \quad & f(x) \\ \text{subject to} \quad & g(x) \leq 0 \\ & h(x) = 0 \end{aligned} \tag{2.13}$$

where x represents the design variables, $f(x)$ is the objective function, and $g(x)$ and $h(x)$ represent inequality and equality constraints. In the context of this project, the design variable (x) here is the location of the mounts along the drivetrain with reference from the front end of the engine. The objective function can be defined as a measure of reaction force or the bending moments experienced by the flanges.

The optimization process evaluates the different mount configurations and identifies the best configuration to minimize the objective function, which in our case is the reaction force or the bending moment at the flanges keeping in mind of all the geometrical constraints.Constraints are introduced to ensure that the solutions obtained from the optimization are physically feasible and practically applicable. These constraints may include geometrical limitations, minimum spacing requirements between the mounts, and other structural restrictions imposed by the drivetrain design.

2.6 Modal Analysis of Mechanical Systems

The common way to determine the dynamic behaviour of a system is by doing Modal analysis of the system. The modal analysis helps in determining the natural frequency of the system and corresponding mode shapes for the structure which helps in making vital design changes. In general, the governing equation of motion (EOM) can be expressed in as in (2.14). For determining the natural frequency, the damping and external forces are neglected, resulting in a simplified equation of motion is given by

$$m\ddot{x} + kx = 0, \tag{2.14}$$

Assuming harmonic motion, the displacement can be expressed as

$$x = \phi e^{i\omega t}, \tag{2.15}$$

Substituting this expression into the equation of motion leads to the eigenvalue problem

$$(k - \omega^2 m)\phi = 0, \tag{2.16}$$

Solving this equation, the natural frequency of the system and corresponding mode shapes can be determined. Modal analysis is important in marine application where the system is subjected to wave impacts, engine excitation and operational vibrations. It is crucial to ensure the system's natural frequency never coincides with the excitation frequency and necessary design changes are suggested with such studies.

3

Methods

3.1 Mount Optimization

3.1.1 Overview of the Approach

The method adopted here is to optimize the mount position to improve load distribution in the marine drivetrain system. This approach combines an analytical method for load evaluation with a constrained non-linear optimization framework.

The analytical method developed in prior work is used as a computational tool to evaluate reaction forces and bending moments for a given mount configuration. To facilitate the interaction between the user and the computational model, a graphical user interface was developed to eliminate the need for direct modification of hard-coded input parameters.The interface was implemented as a web-based application using Google AI Studio to make a user friendly interface. This interface can be found in Appendix A.1. Now, the focus is placed on minimizing these reaction forces or minimizing the bending moment by varying the mount positions.

3.1.2 Optimization Problem Formulation

The optimization problem is defined to reduce extreme loading conditions on a certain mount or to reduce the moment at the flanges. The mount positions are considered to be the design variables, varying along the drivetrain and given by:

$$x = [x_1, x_2, x_3, x_4],$$

where two objective functions can be chosen depending on the user's needs:

- Minimize the maximum absolute reaction force on all mounts.
- Minimize the maximum absolute flange moment on all flanges.

The optimization problem is expressed as

$$\min f(x) = \max_{i=1,\dots,4} |F_i(x)| \qquad \text{or} \qquad \min f(x) = \max_{j=1,\dots,2} |M_j(x)|$$

where F_i denotes the reaction force at the i^{th} mount, with $i = 1, 2, 3, 4$, and M_j denotes the bending moment at the j^{th} flange, with $j = 1, 2$. This type of formulation ensures the most critical loading conditions is reduced rather than the average response, which is important for structural safety concerns

3.1.3 Constraint Definition

To be physically feasible and have a realistic mount configuration some constraints are imposed on the design variable. These constraints are implemented in MATLAB using a non-linear constraint function, where all inequality constraints $c(x)$ are defined such that $c(x) \leq 0$ The constraints applied in the optimization are summarized as follows:

- **Geometric bounds:** Each mount must lie within the physical limits of the drivetrain, which is divided into three segments of lengths L_e, L_m, and L_g. The allowable mount positions are therefore defined by

$$\begin{aligned} x_1 &\leq L_e, \qquad (3.1)\\ x_2 &\leq L_e,\\ L_e \leq\ &x_3 \leq L_e + L_m,\\ L_e + L_m \leq\ &x_4 \leq L_e + L_m + L_g. \qquad (3.2) \end{aligned}$$

  where L_e is the length of the engine, L_m is the length of the e-machine, and L_g is the length of the gearbox.
- **Ordering constraint:** The mounts must be in the correct order along the drivetrain such that

$$x_1 < x_2 < x_3 < x_4. \qquad (3.3)$$

- **Minimum spacing constraint:** A minimum gap variable is introduced to ensure that the mounts do not overlap with each other and maintain some distance with each other. This leads to

$$x_2 - x_1 \geq \text{min\_gap}, \qquad x_3 - x_2 \geq \text{min\_gap}, \qquad x_4 - x_3 \geq \text{min\_gap}. \qquad (3.4)$$

- **Inequality form used in the optimizer:** These minimum gap distance are introduced into inequality constraints and expressed as

$$\begin{aligned} (x_1 + \text{min\_gap}) - x_2 &\leq 0,\\ (x_2 + \text{min\_gap}) - x_3 &\leq 0,\\ (x_3 + \text{min\_gap}) - x_4 &\leq 0. \qquad (3.5) \end{aligned}$$

3.1.4 Optimization Algorithm

The optimization is performed in MATLAB's optimization solver, $fmincon$. This solver is used because the problem involves non-linear objective functions and constraints. The solver calculates the gradient of the objective function numerically and updates the design variables to reduce the objective function. The algorithm for the optimization process is given in Fig 3.1.

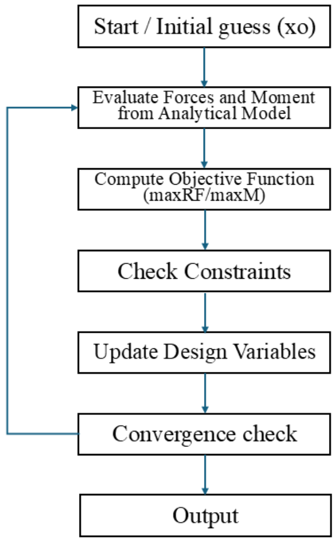

Figure 3.1: Schematic representation of Optimization Algorithm used in MATLAB *fmincon* solver.

3.1.5 Objective Function Switching Strategy

A switch case mechanism is implemented that allows the user to select the cases in the code depending on the failure of the structural integrity, where *maxRF* case can be used in case of a mount failure and *maxM* case can be used in case of a flange failure.
There is always a trade-off between the two cases to either to reduce the reaction force or the moments. Further investigation can be made to achieve best possible configuration for both cases all together, but it is beyond the scope of this project.

3.1.6 Role of analytical Model in Optimization

The analytical model is integrated into the optimization loop and this acts as function to evaluate the reaction force and moment for the particular mount positions. The function *run_mount_analysis* consists of the analytical method developed. The optimization algorithm depends on the output of this analytical method to evaluate the objective function. The analytical method is very efficient in calculating the outputs and hence the time for this is computation is reasonably quick.

3.2 Input Parameters and Optimization Setup

The optimization model is developed to determine the optimal mounting positions for a marine drivetrain system. This incorporates geometric constraints, mass distribution, and spacing requirements to ensure structural feasibility and balanced load distribution.

3.2.1 Objective Function

The objective function is defined to allow the user to select between the two switch cases referred in Sec 3.1.5. The objective functions is expressed as

$$\min f(x) = \max_{i=1,\dots,4} |F_i(x)| \qquad \text{or} \qquad \min f(x) = \max_{j=1,2} |M_j(x)|$$

This corresponds to minimizing the maximum reaction force $|F_i(x)|$ among all mounts and minimizing the maximum moments $|M_j(x)|$ at the flanges, thereby ensuring uniform load sharing and reducing high moments to avoid localized overstressing.

3.2.2 System Configuration

The drivetrain configuration shown in Fig 2.1 is used to define the input parameters for the optimization model. The system is represented by three main components, referred to as blocks, and is supported by four mounting locations. These parameters define the configuration of the marine drivetrain drivetrain

3.2.3 Block Properties

The physical properties of the drivetrain components are assigned to the optimization model based on the component data presented in Tab 2.1 and 2.2. The three drivetrain components are represented as blocks in the optimization model. For each block, the length, mass, and longitudinal position of the centre of gravity are defined as follows:

- **Block lengths (m):**

$$L = [1.3216,\ 0.495,\ 0.556]$$

- **Block masses (kg):**

$$m = [1260,\ 242,\ 350]$$

- **Block centres of gravity (m):**

$$x_{cg} = [0.818,\ 1.613,\ 1.967]$$

.

3.2.4 Design Variables

The design variable vector is defined as

$$x = [x_1,\ x_2,\ x_3,\ x_4]$$

where each x_i represents the longitudinal position of a mount along the structure. For the algorithm to start, the user needs to input an initial guess which allows the algorithm to run iterations. In this case, one can input the current mount configuration x_0 Scania is using providing a good starting point. All the dimension are referred to the Left-Hand side of the engine and are as follows

- **Initial guess (m):**

$$x_0 = [0.001040,\ 0.660800,\ 1.521664,\ 2.372600]$$

3.2.5 Geometric Constraints

The feasible mount positions are defined based on the geometry of the three drive-train blocks. The lower and upper bounds are assigned to each mount position to ensure that the mounts remain within their corresponding structural segments. The bounds used in the optimization model are defined as follows

- **Lower bounds (m):**

$$x_{\mathrm{lb}} = \left[0.001,\ \frac{L_1}{2},\ L_1,\ L_1 + L_2\right] = [0.001,\ 0.6608,\ 1.3216,\ 1.8166]$$

- **Upper bounds (m):**

$$x_{\mathrm{ub}} = \left[\frac{L_1}{2},\ L_1,\ L_1 + L_2,\ L_1 + L_2 + L_3\right] = [0.6608,\ 1.3216,\ 1.8166,\ 2.3726]$$

These bounds define the allowable search region for each mount position and ensure that each mount remains within its corresponding structural segment during the optimization process.

3.2.6 Minimum Spacing Constraint

A non-linear constraint is imposed to ensure adequate spacing between adjacent mounts. A minimum gap of 0.05 m is specified between consecutive mounts to prevent excessive clustering of the mounting points.

3.2.7 Optimization Solver Settings

The optimization is performed using MATLAB's `fmincon` solver with the following parameters

- Optimality tolerance: 1×10^{-8}
- Step tolerance: 1×10^{-8}
- Constraint tolerance: 1×10^{-8}

These settings ensure precise convergence of the optimization process.

3.2.8 Output Parameters

The optimization yields the following outputs:

- Optimized mount positions: x_{opt}
- Reaction forces at each mounts
- Bending moments at each flanges
- Displacement of mounts

3.3 3-D CATIA Model

The brackets and mounts are already designed by Scania providing a part model in CATIA. Theses part models are imported initially into the CATIA *Analysis and Simulation* workbench along with
EST - CATIA - ELFINI STRUCTURAL ANALYSIS 2 Product
and
GDY-CATIA-GENERATIVE DYNAMICS RESPONSE ANALYSIS 2 Product
shareable product CATIA licences . The part assigned to have 3D properties along with the pre-defined material in built in CATIA. Once the material is assigned, they are meshed to the default size. A convergence study is not conducted as this is not the scope of this project. Further investigation can be done as a part of future work to study the effects of mesh size and shape. In our case a parabolic mesh is considered.

In order to achieve reliable results and to ensure the simulation method is working as expected, a base model is constructed and checked with hand calculations. This comparison is discussed in the Section 4 of the report. This section focuses on the setup. Fig 3.2 shows the 3D setup consisting of 4 bracket and mount combination as a base model.

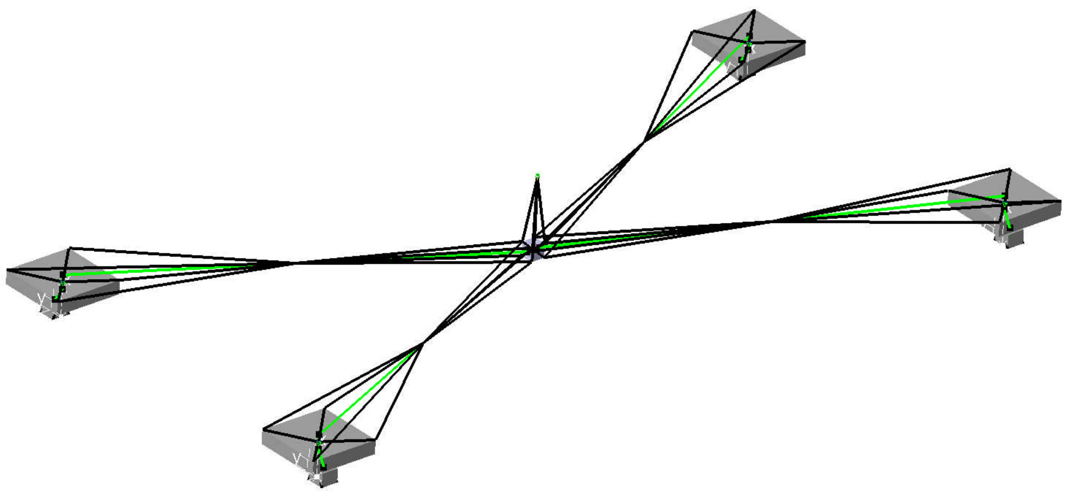

Figure 3.2: Base Model Setup of a 4-mount configuration.

This connections is made for one single mount for explanation purpose. The method can be used for all the other mounts and brackets. Simple blocks are used to represent the bracket and the mount as shown in Fig 3.3 and the numberings in the Table 3.1 guides the user to make this simulation setup.

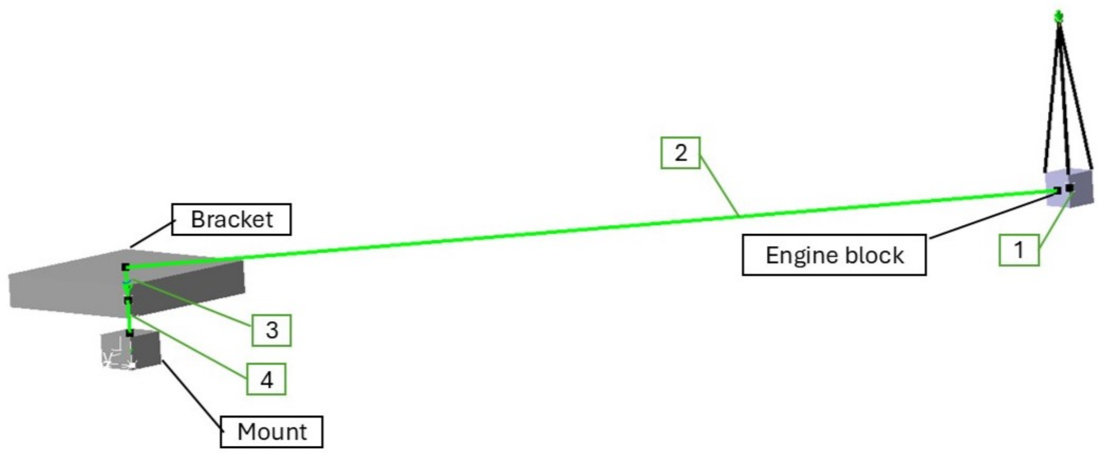

Figure 3.3: General Analysis Connection property assigned to the Baseline 4-mount configuration.

Table 3.1: General Analysis Connections used in the finite element model.

Connection	Interacting Bodies
1	Six faces of the engine block – Centre point of the engine block
2	Any face of the engine block – Upper face of the bracket
3	Upper face of the bracket – Point on the bottom face
4	Point on the bottom face – Upper face of the mount

The General Analysis Connection property establishes the relationship between two entities in the CATIA model. In the present setup, these connections are used to define the interaction between the engine block and the supporting system.

However, the type of connection assigned to each component plays a crucial role in determining the behaviour of the system. After several simulation trials, the connection types and corresponding properties presented in Table 3.2 were assigned to the General Analysis Connections. The resulting Rigid Connection Properties are illustrated in Fig 3.4.

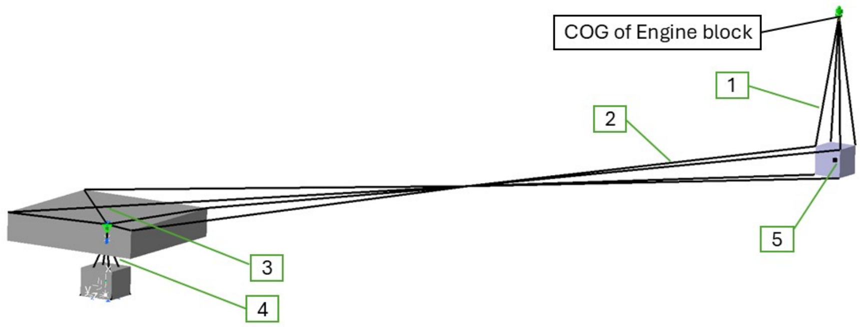

Figure 3.4: Rigid Connection Properties assigned to the General Analysis Connection for a Baseline 4-mount configuration.

Table 3.2: General Analysis Connections and Virtual Part Properties used in the finite element model.

Connection	Connection Type	Purpose
1	Rigid Virtual Part Property	Connects the centre of gravity (CoG) of the engine to one face of the engine block, allowing the engine mass to be represented as a concentrated mass while transferring inertial loads to the structure.
2	Rigid Connection	Establishes a rigid attachment between the engine block and the upper face of the bracket using the same attaching relation defined in the General Analysis Connection.
3	Rigid Connection	Establishes a rigid attachment between the bracket and the lower connection point using the same attaching relation defined in the General Analysis Connection.
4	Rigid Spring Virtual Part Property	Represents the elastic behaviour of the mount. The spring stiffness can be adjusted to simulate the equivalent stiffness of the bracket–mount assembly while preserving the load transfer path.
5	Rigid Connection	Establishes a rigid body behaviour of the engine block using the same attaching relation defined in the General Analysis Connection

The Translation Stiffness 1 referred to Fig 3.5 represents the stiffness of the mount in the vertical direction. Keeping in mind of the boundary condition discussed in 2.2 which is to eliminate any other axis movements, the translational stiffness values are assigned to be high in order of magnitude. This acts like a virtual boundary condition instead of fully restricting movement in other directions to simulate a real-life condition.

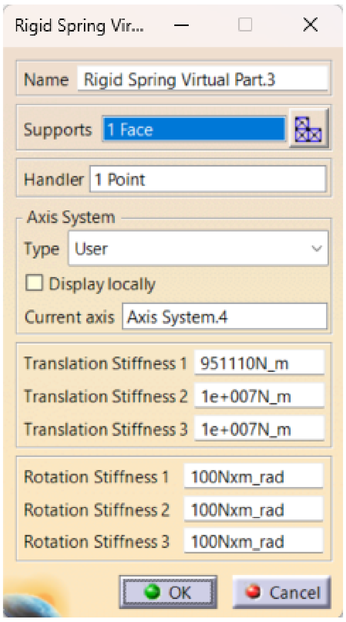

Figure 3.5: Rigid Spring Virtual Part Property assigned to the base line model.

The COG of the engine must be assigned the weight of the engine and an acceleration of 1g is introduced into the Rigid Virtual Part Connection in Fig:3.6. Finally, the clamped restraints are introduced to the bottom face of the mount. All these

(a) Rigid Virtual Part connecting the COG and the blocks.

(b) Mount Restraints assigned at the base of the mounts.

Figure 3.6: Boundary conditions applied to the finite element model.

are translated to all the four mounts in the setup and the simulation is run. The simulation setup is translated to two new pairs of brackets already used by Scania Power Solutions.

For Dynamic Loading case study, Scania has a dataset of acceleration experienced by a marine drivetrain at several coordinates. These dataset is shown in Fig 3.7 and is used as an input modulation in vertical direction at the Mount Restraints in Fig Fig 3.6(b). The distributed mass is assigned at the COG of the block as referred in Fig 3.6(a). The entire model is assigned a damping factor of 0.02 to account for energy dissipation during the simulation. This represents a 2% loss of vibrational energy, with the remaining 98% retained in the system, thereby providing a more realistic representation of the damping behaviour of the drivetrain during dynamic loading.

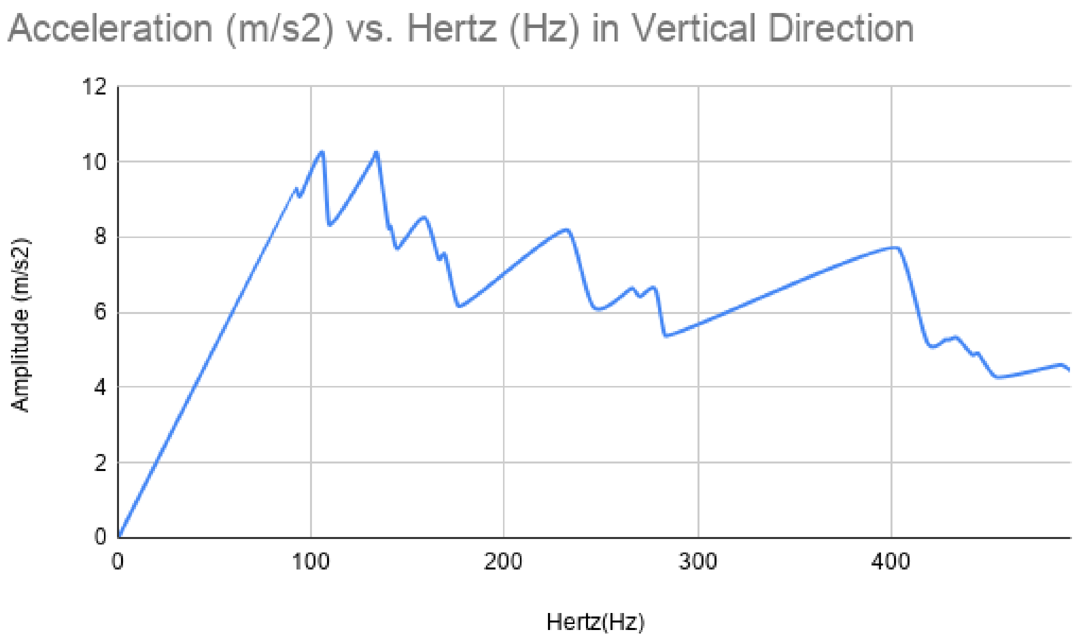

Figure 3.7: Experimental data for frequency modulation of the marine drivetrain.

4
Results

4.1 Comparison between analytical model case and optimizer cases

When the optimizer is run for the initial guess, the code outputs the best mount configuration for the given set of physical parameters. The physical parameters include the masses of the components, stiffness of the bracket and mount, length of each component, and the centre of gravity of each block. An initial guess

$$x_0 = [0.001, 0.6, 1.5, 2.3], \tag{4.1}$$

which is close to the analytical model mount positions, is used. Based on this initial guess, the optimizer iteratively varies the mount positions until convergence is reached. The outputs are then recorded.
The objective is to study how different mount positions affect the load distribution in the drivetrain for three different cases:

1. Analytical Model case
2. Minimize the maximum reaction force (maxRF)
3. Minimize the maximum moment (maxM)

4.1.1 Optimized Mount Position for Different Cases

The optimization algorithm iteratively evaluates the objective function for each set of candidate mount positions and checks the resulting solution against the specified constraints and convergence tolerances set in Section 3.2. The iterations continue until the convergence criteria are satisfied, after which the optimized mount positions are obtained for each objective function. All mount positions are measured from the front end of the engine, as shown in Fig 2.1. The resulting optimized mount positions for each case are presented in Table 4.1.

Table 4.1: Final results of optimized mount positions for three cases.

Configuration	x_1 **(m)**	x_2 **(m)**	x_3 **(m)**	x_4 **(m)**
analytical model	0.149	1.252	1.642	2.247
maxRF	0.004	1.117	1.616	1.820
maxM	0.462	1.050	1.478	1.964

4.1.2 Reaction Force Data for Different Cases

The optimized mount positions are substituted into the analytical model to compute the reaction forces at each mount. The results are presented in Table 4.2.

Table 4.2: Comparison of Reaction Forces at Mount Locations.

Case	Mount 1 (kN)	Mount 2 (kN)	Mount 3 (kN)	Mount 4 (kN)
analytical model	6.2485	4.5824	4.0678	3.2694
maxRF	4.5612	4.5017	4.5439	4.5612
maxM	5.7397	4.7488	4.1686	3.5110

4.1.3 Moments at the Flanges for Different Cases

The flange moments corresponding to the optimized mount positions are calculated and presented in Table 4.3.

Table 4.3: Comparison of Flange Moments.

Case	Flange 1 (kN·m)	Flange 2 (kN·m)
analytical model	-1.4211	-0.8908
maxRF	-0.7045	0.4988
maxM	0.0000017	0.0000014

A significant improvement is observed across the optimized cases. The maxRF configuration achieves near-uniform load distribution across mounts, while the maxM configuration effectively minimizes flange moments, resulting in near-zero bending. These observations will be discussed further in the Conclusion section.

4.2 Base model and Hand Calculation

A base model with a simple geometry was created to check and verify the connection properties against hand calculation. In this section the results for static, frequency analysis and hand calculations are shown and a lot of co-relation between each other suggests that this method is reliable.

4.2.1 Results for Static Deflection

These results in Fig 4.1 focus on the simulation results conducted for static deflection of the mount under loading of the engine at 1g acceleration.

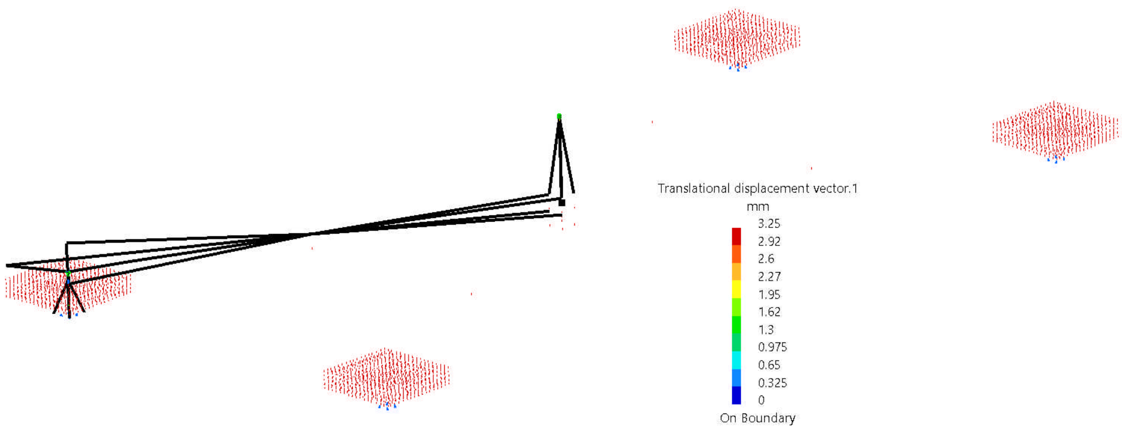

Figure 4.1: Simulation results for static deflection of Baseline model in CATIA.

4.2.2 Results for frequency analysis

The results in Fig 4.2 for the frequency analysis case is really close to results from hand calculation and the formula is discussed in Section 4.2.3.

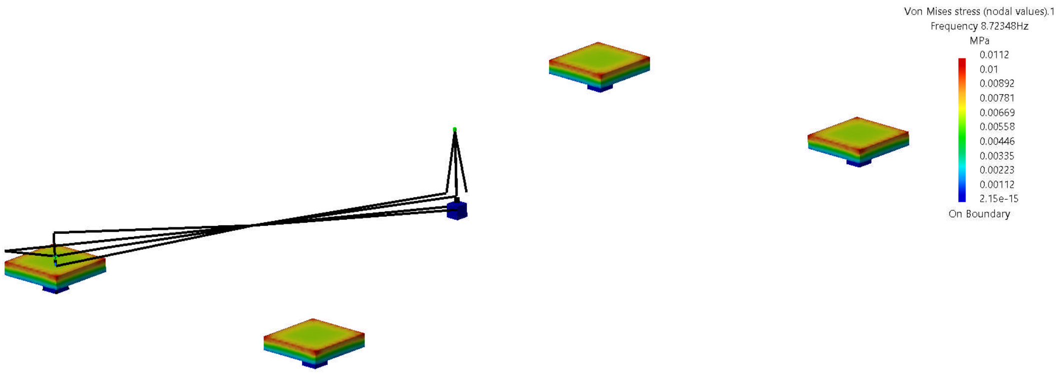

Figure 4.2: Frequency analysis of Baseline model in CATIA.

4.2.3 Hand Calculation for Static Deflection and Frequency

Given,

$$k = 951110 \text{ N/m}, \tag{4.2}$$
$$m = 1260 \text{ kg}, \tag{4.3}$$
$$g = 9.81 \text{ m/s}^2, \tag{4.4}$$
$$n = 4 \tag{4.5}$$

the equivalent stiffness of the mounting system is

$$k_{eq} = nk = 4 \times 951110 = 3\,804\,440 \text{ N/m.} \tag{4.6}$$

The natural circular frequency is calculated as

$$\omega_n = \sqrt{\frac{k_{eq}}{m}} = \sqrt{\frac{3\,804\,440}{1260}} = 54.95 \text{ rad/s,} \tag{4.7}$$

The corresponding natural frequency is

$$f_n = \frac{\omega_n}{2\pi} = \frac{54.95}{2\pi} = 8.75 \text{ Hz,} \tag{4.8}$$

The gravitational force acting on the drivetrain is

$$F = mg = 1260 \times 9.81 = 12\,360.6 \text{ N,} \tag{4.9}$$

Finally, the static deflection is

$$\delta = \frac{F}{k_{eq}} = \frac{12\,360.6}{3\,804\,440} = 0.00325 \text{ m} = 3.25 \text{ mm.} \tag{4.10}$$

4.3 Study on Engine Brackets and Mounts

The results above show extreme similarities with simulation and hand calculation , hence it is a good simulation method to follow for the future studies.

This section presents the results obtained for the different bracket configurations used by Scania in marine drivetrain applications. The performance of each bracket configuration is evaluated based on the following criteria:

- Static deflection,
- Natural frequency,
- Dynamic response of the Support System,

To obtain the dynamic response, the modulation Fig 3.7 is applied on the restraints. Restraints here are defined at the base of the Mounts as in Fig 3.6(b).

4.3.1 Study on Bracket Set 3038127

The Fig 4.3 represents the simulation results of the bracket set 3038127 in case of static deflection of the marine drivetrain. The Fig 4.4 represents the simulation results of the bracket set 3038127 visualising the natural frequency of the marine drivetrain.

The Fig 4.5 represents the simulation results of the bracket set 3038127 visualising the dynamic response for the frequency modulation in Fig 3.7 of the marine drivetrain.

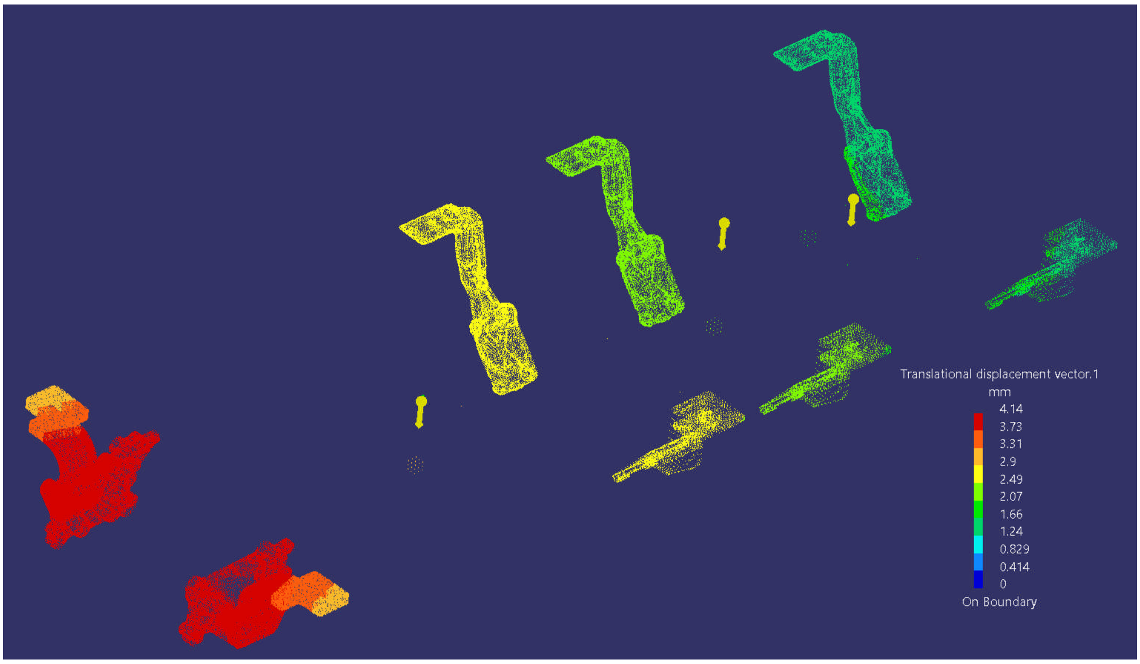

Figure 4.3: Simulation results of static deflection of bracket set 3038127.

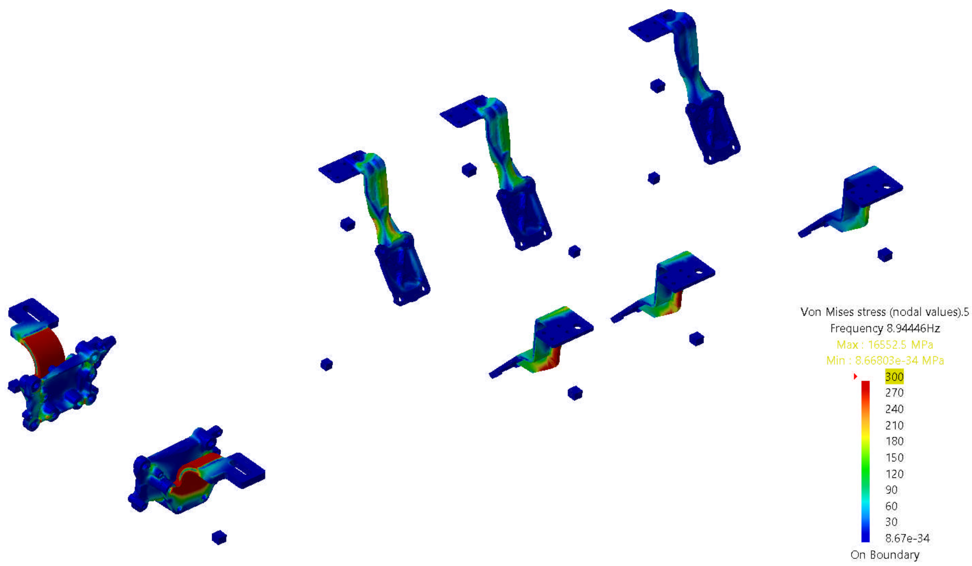

Figure 4.4: Simulation results of natural frequency of bracket set 3038127.

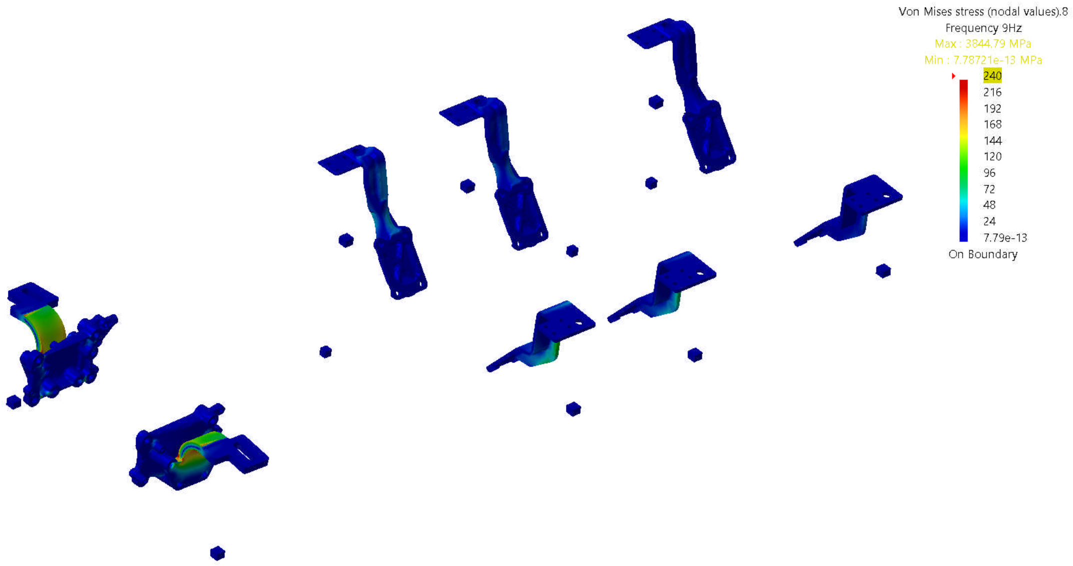

Figure 4.5: Simulation results of dynamic response of bracket set 3038127.

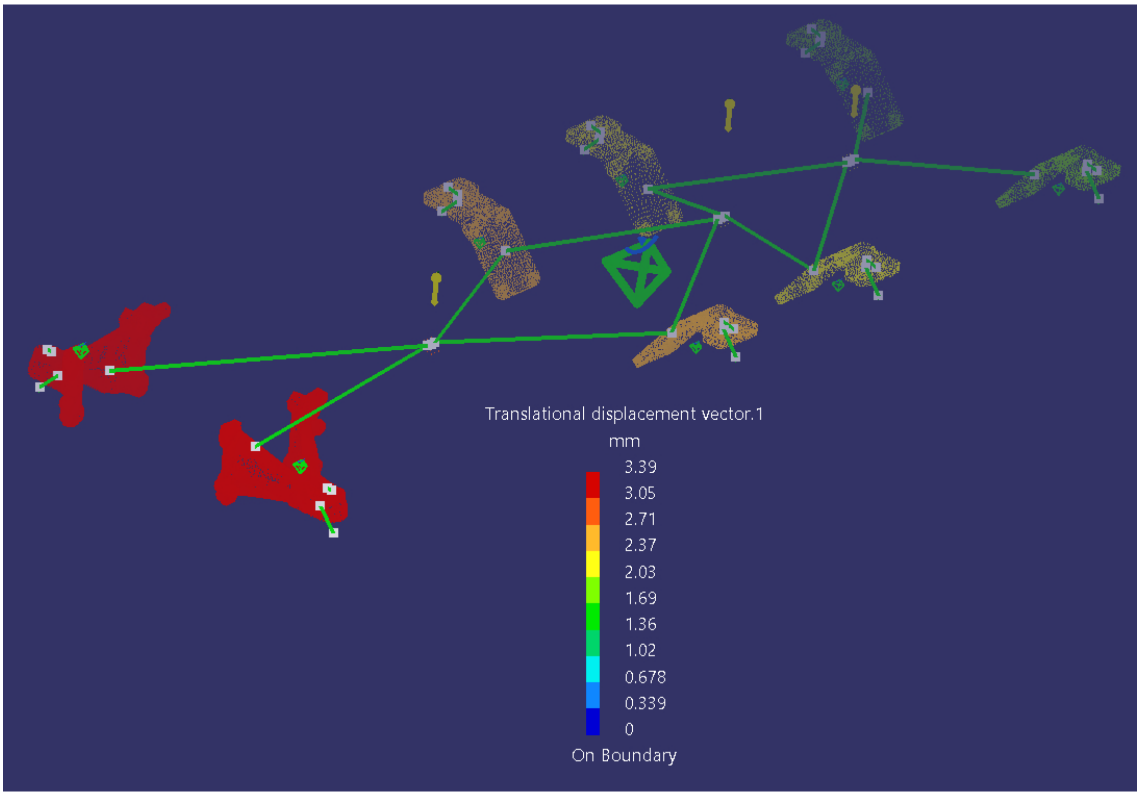

Figure 4.6: Simulation results of static deflection of bracket set 2520240.

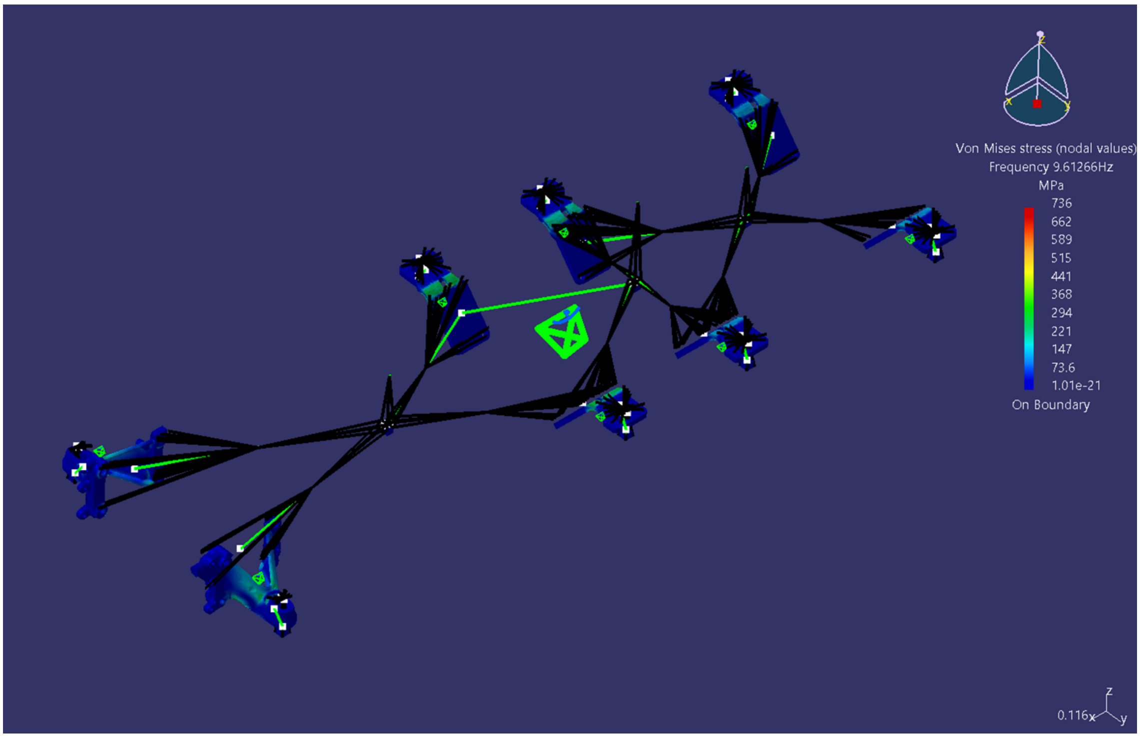

Figure 4.7: Simulation results of natural frequency of bracket set 2520240.

4.3.2 Study on Bracket Set 2520240

The Fig 4.6 represents the simulation results of the bracket set 2520240 in case of static deflection of the marine drivetrain.
The Fig 4.7 represents the simulation results of the bracket set 2520240 visualising the natural frequency of the marine drivetrain.
The Fig 4.8 represents the simulation results of the bracket set 2520240 visualising the dynamic response for the frequency modulation in Fig 3.7 of the marine drivetrain.

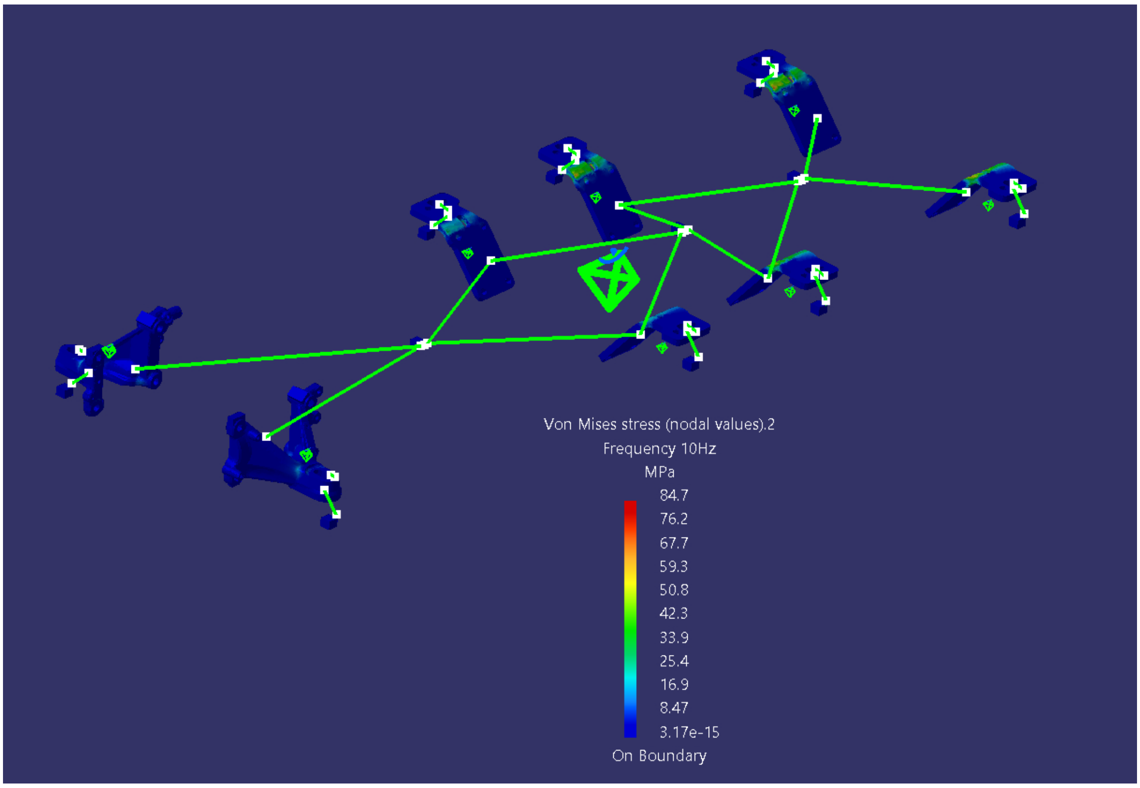

Figure 4.8: Simulation results of dynamic response of bracket set 2520240.

5

Conclusion

5.1 Optimized Results

5.1.1 Displacements of each mounts

The displacement of each mounts are plotted in Fig 5.1 The displacements of each mount give us a clear idea of how the load is distributed along the length of the marine drivetrain under different cases. The displacement seen in the analytical model has a very high deflection at the front face of the engine or in this case $Mount1$ and gradually decreases as we move towards the other mounts. This indicates high uneven load distribution and the front mount takes most of the load.

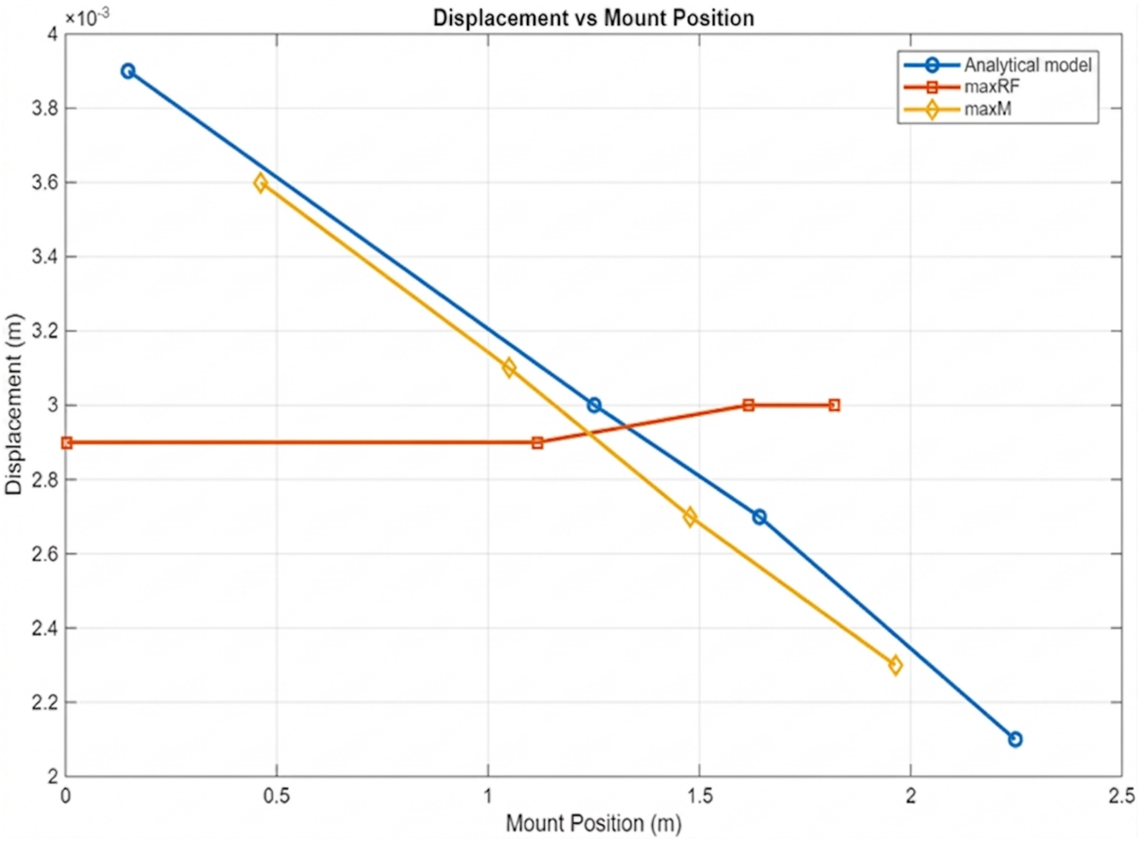

Figure 5.1: Comparison of displacements v/s mount position along the drivetrain for all three cases.

In contrast, the optimized configuration from the $maxRF$ case demonstrates a more uniform loading and all the mounts are displaced evenly. This indicates that the there is even distribution of the loads across the mounts. Such a distribution is desirable from a structural point of view, as it reduces any failures or excessive deformation at any single support and resulting in prolonged support life.

The $maxM$ case is a representation of something in-between of the two cases. There is some high deflections in the $Mount1$ but it is much lesser than the analytical model. This case also shows has a decrease in the deflection along the drivetrain.

5.1.2 Reaction forces on each mounts

The results corresponding to the reaction forces in Fig 5.2 have a similar story as discussed in Section 5.1.1. In the analytical model there is decrease in the loading as we move along the drivetrain. There is high localized loading in $Mount1$ similar to case of displacement, seen in Fig 5.1. This is also because the force is directly proportional to the displacement $F \propto \delta$, where F is the reaction force and δ is the displacement by the mount. This must be avoided to prolong the longevity of the support system

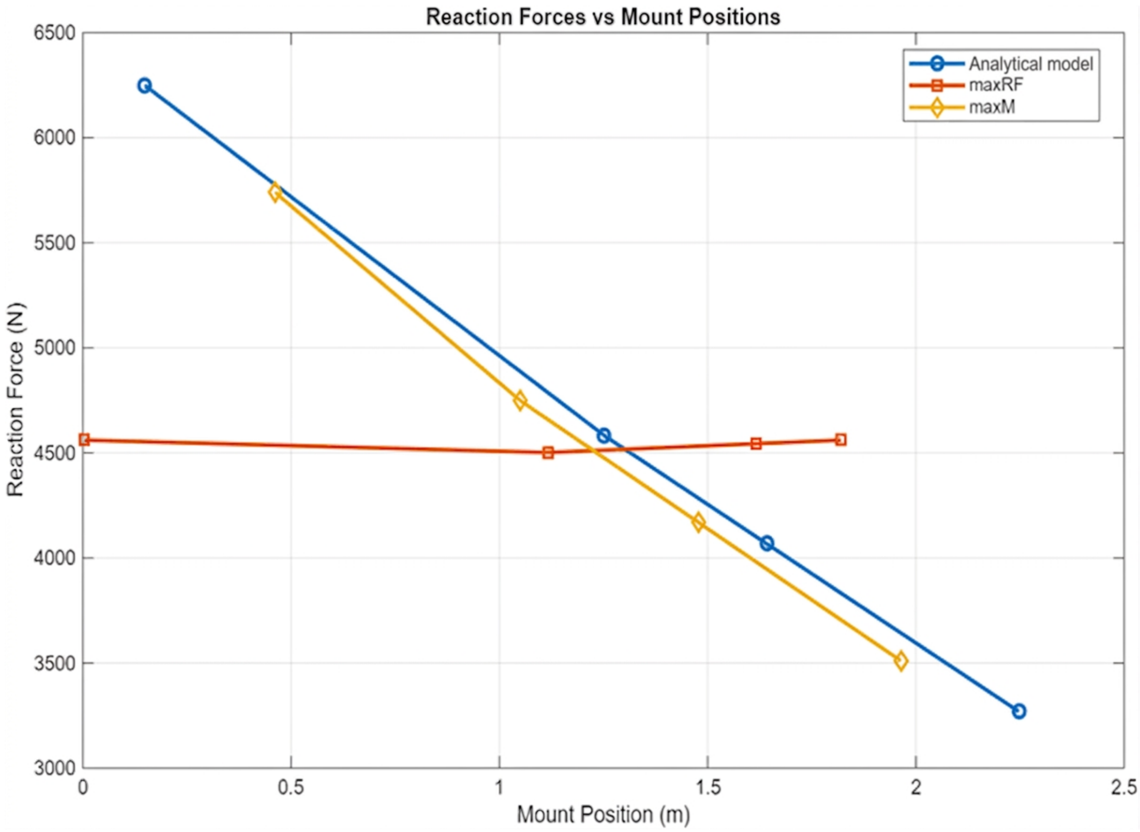

Figure 5.2: Comparison of reaction force v/s mount position along the drivetrain for all three cases.

The $maxRF$ case is optimized to distribute the load evenly and results confirm this, eliminating the peak forces on the individual mounts. One more observation is that

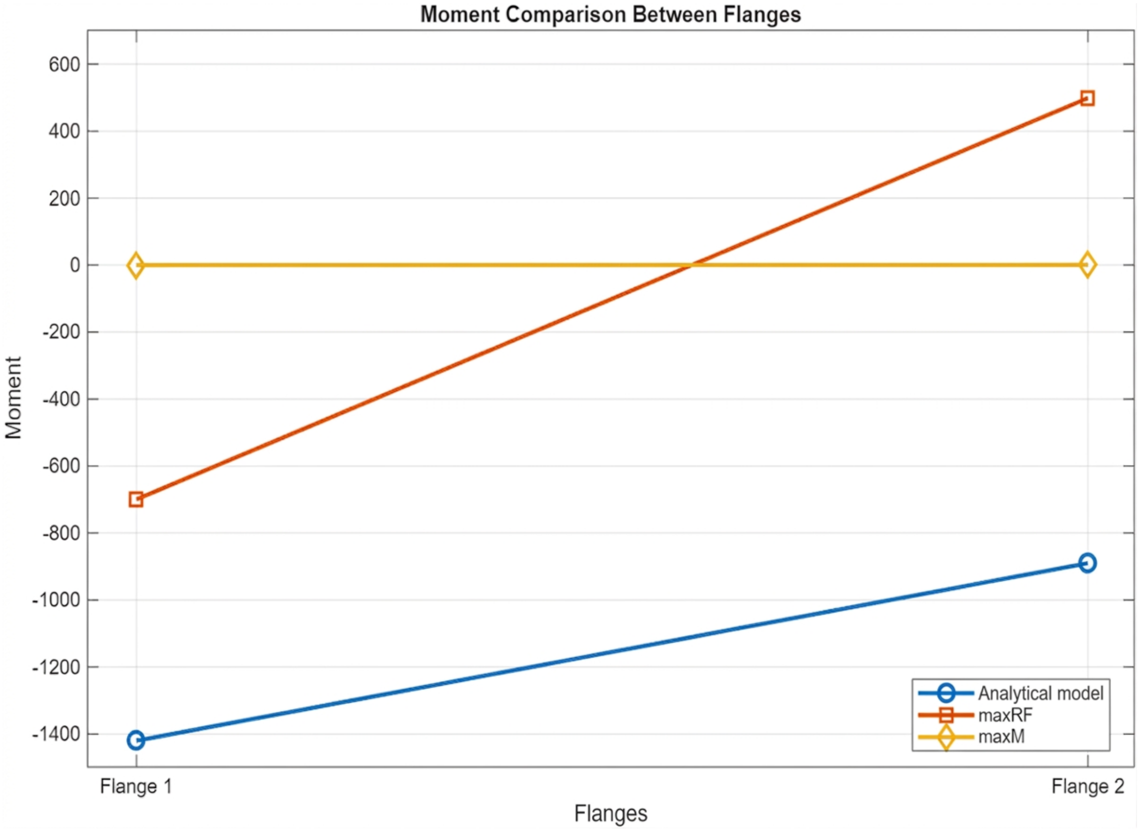

Figure 5.3: Comparison of moments at the flanges for all three cases

the mounts are equally spaced along the drivetrain allowing the load to distribute quite evenly. Such redistribution is advantageous for the enhancing the the durability of the system and minimize the mount failure.

The results for moments at each flanges are shown in Fig 5.3. In comparison to the $maxM$ shows a gradual reduction in the reaction forces to the analytical model. Although the $maxM$ reduces the flanges moments significantly, it does not fully equalize the load distribution.

5.2 Comparison between analytical model and Base model

The study done with the analytical model and the baseline finite element model results are shown in Table 5.1.

Table 5.1: Comparison of analytical model and the base model.

Parameter	analytical model	Baseline model	% Error
Deflection (mm)	3.25	3.25	0.00
Natural Frequency (Hz)	8.75	8.72	0.34

The results shows significant coherence with each other and the base simulation model is modelled well into a 2D model which is theoretically close to an ideal conditions. This gives us a good idea of how to use connection properties for marine drivetrain system.This serves as a foundation to use these connection properties for future bracket and mount sets.

5.3 Comparison between analytical model and bracket set

The Deflection of each mount of two bracket sets can be determined by the analytical model MATLAB Code from [1]. A comparison between the CATIA models in Section4.3 and analytical model is shown in Fig 5.4 and Fig 5.5

5.3.1 Displacement

There is a good coincidence with the existing model and the simulation setup There is considerably good accuracy for the bracket set in Fig 5.4 with The mean absolute percentage error (MAPE) of 20.47%. However there is better relation between each other with Fig 5.5 having an MAPE of 5.59%.

Such disagreement is expected as the analytical model is assumed to be perfectly elastic and all the motion are in a 2D plane compared to the simulation results to be more realistic. By this, we can conclude that the simulation setup is a good assumption to determine the displacements.

5.3.2 Natural Frequency Analysis

The base model uses uniform bracket stiffness, but for the bracket set, the stiffness of the brackets differs for the Left and Right-Hand side of the marine-drivetrain and the analytical model was derived in the Section 2.4. Using that derivation we can estimate a value for the natural frequency of the system. The difference in the stiffness is purely due to different design approach by Scania and is out of the scope of the project.

The equivalent stiffness of each support is first calculated using the bracket and mount stiffness values. The stiffness values of each bracket were calculated in Appendix A.3. One of the system equivalent stiffness for bracket set in Section 4.3.1 is calculated below for reference.

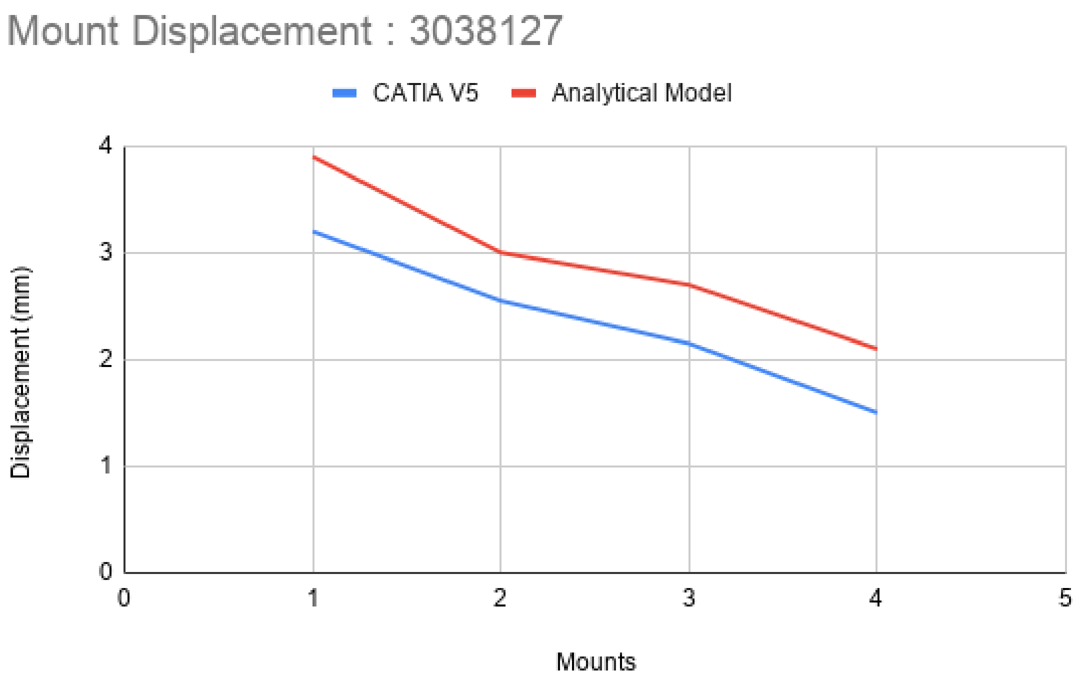

Figure 5.4: Comparison between CATIA and analytical model for Bracket Set 3038127.

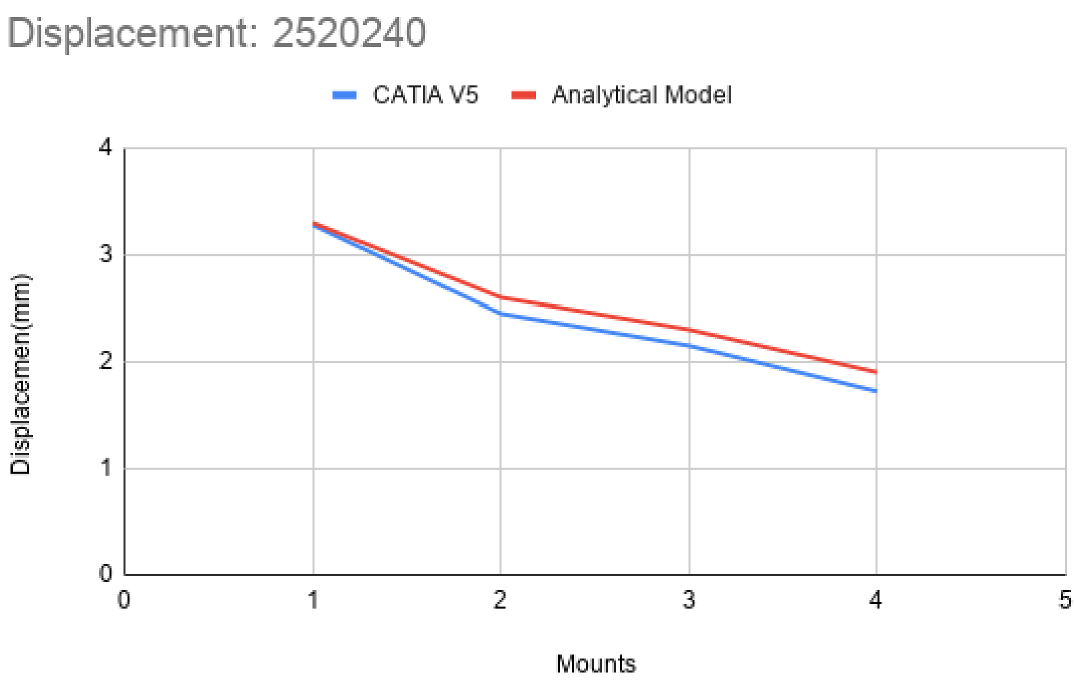

Figure 5.5: Comparison between CATIA and analytical model for Bracket Set 3038127.

To calculate the equivalent stiffness on the Left-Hand side of the drivetrain,using the formulae from the (2.7) we get,

$$k_{LH,1} = \frac{4.59 \times 10^6 \times 951110}{4.59 \times 10^6 + 951110} = 787856 \text{ N/m}, \tag{5.1}$$

$$k_{LH,2} = k_{LH,3} = k_{LH,4} = \frac{3.60 \times 10^6 \times 951110}{3.60 \times 10^6 + 951110} = 752343 \text{ N/m}. \tag{5.2}$$

Similarly, for the right-hand supports

$$k_{RH,1} = \frac{4.59 \times 10^6 \times 951110}{4.59 \times 10^6 + 951110} = 787856 \text{ N/m}, \tag{5.3}$$

$$k_{RH,2} = k_{RH,3} = k_{RH,4} = \frac{3.82 \times 10^6 \times 951110}{3.82 \times 10^6 + 951110} = 761508 \text{ N/m}. \tag{5.4}$$

Hence, the equivalent stiffness of the complete support system is

$$\begin{aligned} k_{eq,\text{system}} &= \sum_{i=1}^{4} k_{LH,i} + \sum_{i=1}^{4} k_{RH,i} \\ &= (787856 + 3 \times 752343) + (787856 + 3 \times 761508) \\ &= 6\,117\,266 \text{ N/m} \end{aligned} \tag{5.5}$$

The total mass including bracket mass is

$$m = 1260 + 242 + 350 + 20.171 = 1872.171 \text{ kg} \tag{5.6}$$

The natural circular frequency is therefore

$$\omega_n = \sqrt{\frac{6117266}{1872.171}} = 57.16 \text{ rad/s} \tag{5.7}$$

The corresponding natural frequency is

$$f_n = \frac{57.16}{2\pi} = 9.10 \text{ Hz} \tag{5.8}$$

This can be calculated with MATLAB and the code can be found in Appendix A.2.

Similarly, the natural frequency analysis for the Bracket Set 2520240 is conducted and the Table 5.2 below displays the results

Table 5.2: Comparison of analytically calculated and simulated natural frequencies for different bracket sets.

Bracket Set	**analytical (Hz)**	**CATIA (Hz)**	**Error (%)**
3038127	9.10	8.94	1.76
2520240	9.81	9.61	2.04

The results compared to analytical method with the CATIA model shows high correlation between each other. The slight discrepancy is due to the 3D nature of the CATIA model and other assumptions made before. The percentage error is less than 3% for both cases, demonstrating good agreement between the analytical formulation and the finite element model.

6

Discussion and Future Work

6.1 Optimizer Tool for Mount Position

The optimization tool *fmincon* developed in MATLAB, provides a systematic approach to solve the objective function either the maximum reaction force experienced by the mounts or the bending moments acting on the drivetrain flanges. Unlike the analytical method in previous work, where mount position were manually selected and evaluated, this tool automates the process on selecting the best mount position satisfying the geometric constraints and minimizing the objective function. The results indicate that no single mount arrangement simultaneously minimizes both reaction forces and flange moments, highlighting the inherent trade-off between the two design objectives.

The optimization was performed in MATLAB *fmincon* solver with the Active-Set algorithm. This algorithm was used in this case because it is computationally efficient for small non linear problems like this work, which involves four design variables. Consequently, the final solution depends on the initial guess supplied by the user and the solution obtained does not represent a global optimum. Although, the Active-Set Algorithm performed satisfactory with this work.

Future work could address this limitation by employing more robust optimization techniques such as Sequential Quadratic Programming (SQP) combined with a MultiStart strategy such as Genetic Algorithm or Particle Swarm Optimization Technique. One problem that was encountered is that the obtained mount position sometimes overlap the position of the flange interface. A new constraint to tackle such situation can be introduced.

Overall, the developed optimization tool provides a practical engineering tool that significantly reduces the time required to evaluate different mounting arrangements while enabling application engineers to investigate multiple design alternatives in an automated manner.

6.2 Finite Element model

The second contribution of this work was the development of a finite element model in CATIA V5 to investigate the dynamic behaviour of the complete marine drivetrain assembly. A comparison between analytical and FEM showed close agreement with each other. The natural frequency predicted by finite element differed by 2% from analytical method.

Following this, the FEM uses this to investigate the stress distribution within the brackets over the range of operating frequency. The stress values showed significantly less in other frequencies however increased significantly as the excitation frequency approached the system's natural frequency. This behaviour is consistent with the expected dynamic amplification associated with resonant excitation.

The evaluation of the von Mises stress concentration and the structural integrity in the bracket sets is not analysed and is beyond the scope of this work. Although some conclusions can be drawn that the system shows high stresses when the excitation frequency is equal to the natural frequency of the natural frequency.

Future work could involve a mesh convergence study. There are some high stress concentrations at boundaries and sharp edges. This can be solved by reducing the mesh size. However, it will require high computational power with such complex connections in a marine drivetrain. Based on the stress map developed, one can also make design changes according to the on-field requirements at Power Solutions. Overall, the developed CATIA methodology establishes a validated simulation framework for evaluating different bracket configurations and provides a foundation for future dynamic investigations of marine drivetrain support systems.

Bibliography

[1] Rajesh Vasisth Puthenmadom, H. (2024). *An analytical method to estimate behaviour of engine mounts and flanges in marine drivetrains* (Dissertation). Retrieved from https://urn.kb.se/resolve?urn=urn:nbn:se:kth:diva-355866

[2] Park, Y. (2021). An Automatic Program of Generation of Equation of Motion and Dynamic Analysis for Multi-body Mechanical System using GNU Octave. *Journal of Applied and Computational Mechanics*, 7(3), 1687–1697. https://doi.org/10.22055/jacm.2020.33826.2293

[3] Trelleborg Antivibration Solutions. *Mount Finder Pro*. Available at: https://www.trelleborg.com/apps/mountfinder/index.html

[4] Dassault Systèmes. *CATIA V5*. Available at: https://www.3ds.com/products/catia

[5] MathWorks. *MATLAB*. Available at: https://www.mathworks.com/products/matlab.html

[6] GitHub. *GitHub*. Available at: https://github.com/

[7] Rao, S. S. (2017). *Mechanical Vibrations*. Pearson.

A

Appendix 1

A.1 Marine Drivetrain Simulator

A web-based marine drivetrain simulation tool was developed to support the analysis and evaluation of the drivetrain system considered in this thesis. The simulator provides an interactive environment for investigating the static behaviour of the marine drivetrain and enables the user to modify relevant system parameters and observe their influence on the drivetrain response.

The simulator is available through the following web application:

`https://marine-drivetrain-simulator.ai.studio`

The simulator was developed as a complementary tool to the numerical analyses presented in [1].

A.2 MATLAB Code

The MATLAB source code developed as part of this thesis is maintained in the following GitHub repository:

`https://github.com/brijeshn925-ops/master_thesis_appendix_2026`

A.3 Bracket Stiffness Evaluation

The bracket stiffness can be calculated by applying a constant force at the face of intersection and clamp the bracket at the mounting position of the engine.

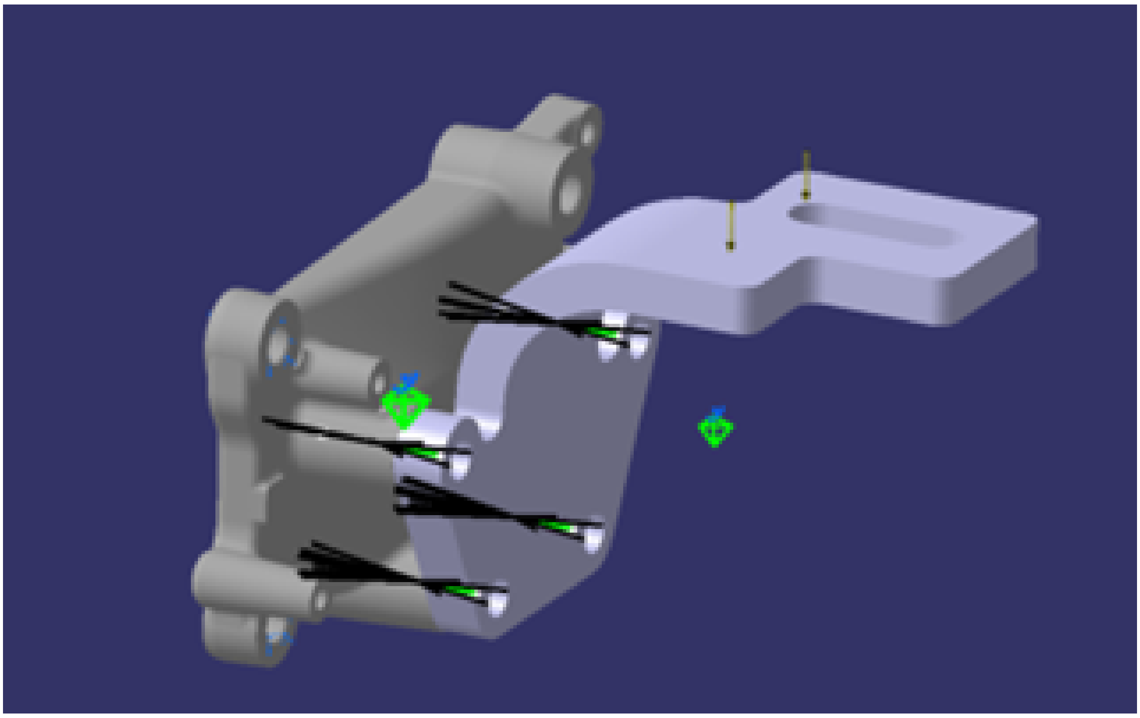

Figure A.1: Force acting on the face of the mount

The Fig A.2 depicts the behaviour of the bracket under loading condition ranging from $10000N$ to $15000N$, The stiffness of the bracket can be evaluated by 2.1 and is found to be $k = 4.59e6\ N/m$.

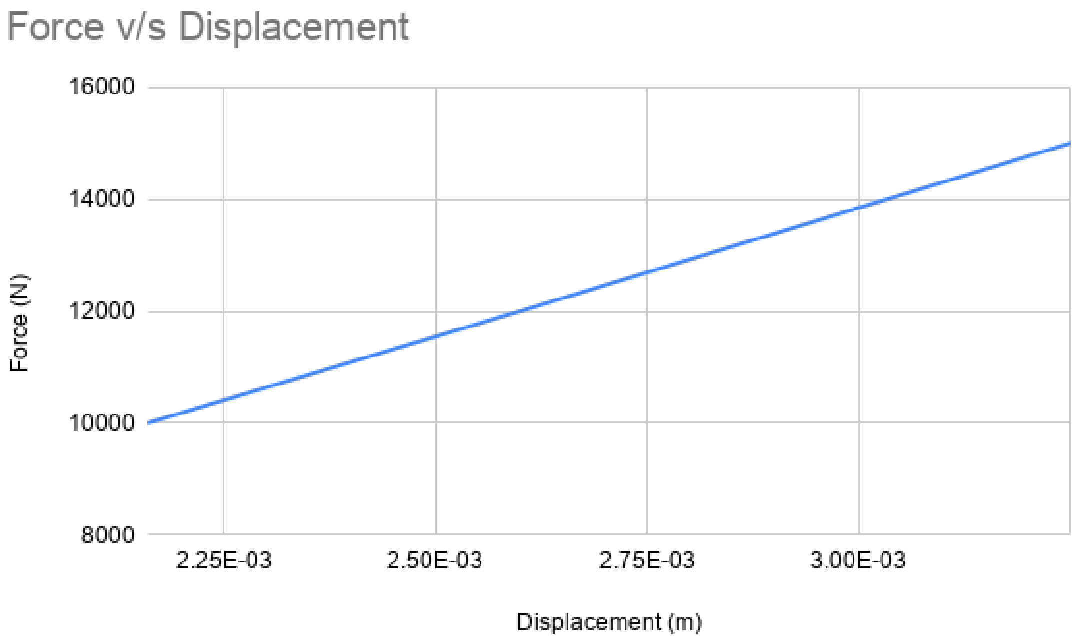

Figure A.2: Graph of Force acting on the face of the bracket v/s Displacement occurred by the bracket.

Similarly, the remaining brackets are evaluated under the same loading and boundary conditions. The calculated stiffness values for all brackets and mounts are presented in Table A.1.

Table A.1: Calculated stiffness values of brackets and mounts.

Part Number	**Stiffness (N/m)**
3038127	4.59×10^6
2788087	3.82×10^6
2784246	3.60×10^6
2520242	7.14×10^7

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026
www.chalmers.se

CHALMERS
UNIVERSITY OF TECHNOLOGY