An Evaluation of Finite Element Models of Stiffened Plates Subjected to Impulsive Loading

by

Omri Pedatzur

B.Sc., Mechanical Engineering, Tel Aviv University, 1995

Submitted to the Department of Ocean Engineering and the Department of Mechanical Engineering in Partial Fulfillment of the Requirements for the Degrees of

Master of Science in Naval Architecture and Marine Engineering

and

Master of Science in Mechanical Engineering

at the

Massachusetts Institute of Technology

February 2004

© Omri Pedatzur, MMIII. All rights reserved.
The author hereby grants MIT permission to reproduce and to distribute publicly paper and electronic copies of this thesis document in whole or in part.

Signature of Author.

10 December, 2003

Department of Ocean Engineering and the
Department of Mechanical Engineering

Certified by.

Klaus-Jürgen Bathe
Professor of Mechanical Engineering
Thesis Advisor

Certifieby.

David V. Burke
Senior Lecturer, Department of Ocean Engineering
Thesis Reader

Accepted by.

Michael S. Triantafillou
Professor of Ocean Engineering
Chairman, Department Committee on Graduate Students
Department of Ocean Engineering

Accepted by.

Ain A. Sonin
Professor of Mechanical Engineering
Chairman, Department Committee on Graduate Students
Department of Mechanical Engineering
An Evaluation of Finite Element Models of Stiffened Plates Subjected to Impulsive Loading

by

Omri Pedatzur

B.Sc. Mechanical Engineering, Tel Aviv University, 1995

Submitted to the Department of Ocean Engineering and the Department of Mechanical Engineering in Partial Fulfillment of the Requirements for the Degrees of

Master of Science in Naval Architecture and Marine Engineering

and

Master of Science in Mechanical Engineering

Abstract

Different finite element models are evaluated for two very common structures, a cantilever beam and a stiffened plate, subjected to impulsive loading.

For the cantilever beam case, the finite element models are one, two or three dimensional models. Various results from the finite element analyses are compared including with analytical solution and a closed-form approximate solution.

For the stiffened plate, the models differ from each other by the way the plate and the stiffeners are modeled. Some of the models are very accurate but require much computational resource, while other models are considerably more economic. The purpose of this study was to decide which model is most appropriate for analyzing a ship deck under slamming conditions. The plate modeled with 4-node shell elements and the stiffeners modeled with 2-node iso-beam elements are shown to yield excellent results while requiring reasonable computational resources.

In addition to the evaluation of the finite element models, the thesis presents closed-form approximate solutions for both the cantilever beam and the stiffened panel. These simplified solutions can be used to check and validate finite element analyses of similar structures. Furthermore, the analytical solutions can be very useful in understanding the basic physical behavior and the main parameters governing the dynamic response of these structures.

Thesis Advisor: Klaus-Jürgen Bathe.
Title: Professor of Mechanical Engineering.

Thesis Reader: David V. Burke.
Title: Senior Lecturer, Department of Ocean Engineering.
Acknowledgements

A few people have made this thesis possible and it gives me great pleasure to thank them all. First and foremost, I would like express my deep gratitude to my thesis advisor, Prof. Klaus-Jürgen Bathe, for his guidance, excellent suggestions and constructive criticism. His endless knowledge of finite element methods and his enthusiasm for teaching are unparalleled. He is an inspiration to all his students and I’m very proud to have worked with him.

I would also like to thank my thesis reader, Dr. David V. Burke, who used to say in class that “every day I’m getting a little bit older and a little bit smarter”. I hope that by the time I am his age, I will be as smart as he his.

Special thanks go out to ADINA R&D for allowing me to use their proprietary finite element code ADINA® for my research. All the finite element analyses results presented in this paper were generated by their software.

I am most grateful to the Israeli Navy for giving me the opportunity to take part in the MIT experience. I never knew it was my dream until I got here and I’m both pleased and honored to make my dream come true. I’m sure that the knowledge and skills I have gained through my education at MIT will be beneficial to the Israeli Navy in years to come.

Last but not least I would like to express my deepest gratitude toward my wife Einat and my son Elad for their unconditional love and support.

This thesis is dedicated to my beloved parents, Shosh and Avi. I owe them more than I can express.
Table of Contents

Abstract ........................................................................................................................................ 2
Acknowledgements .................................................................................................................. 3
Table of Contents .................................................................................................................. 4
List of Figures ........................................................................................................................ 6
List of Tables .......................................................................................................................... 8
Nomenclature ........................................................................................................................ 9
Chapter 1 – Introduction ........................................................................................................ 10
  1.1 Ship Structural Design ................................................................................................. 10
  1.2 Stiffened Plate Panel and Ship’s Structure ............................................................... 13
  1.3 Dynamic forces on the Ship’s Structure and Slamming ............................................ 15
  1.4 Objective .................................................................................................................... 17
Chapter 2 – Literature ......................................................................................................... 18
  2.1 Slamming of Ships ....................................................................................................... 18
  2.2 Finite Element Modeling of a Stiffened Plate Panel .................................................... 20
Chapter 3 – Finite Elements Formulation ......................................................................... 22
  3.1 The Process of Finite Element Analysis .................................................................... 22
  3.2 The Principle of Virtual Displacement ...................................................................... 23
  3.3 Equilibrium Equations in Static Analysis .................................................................. 24
  3.4 Equilibrium Equations in Dynamic Analysis ............................................................ 25
Chapter 4 – Element Descriptions and Assumptions ....................................................... 26
  4.1 Hermitian Beam Elements ......................................................................................... 26
  4.2 Iso-Beam Elements ..................................................................................................... 27
  4.3 2D Plane Stress Solid Elements ................................................................................ 28
  4.4 Shell Elements ............................................................................................................ 29
  4.5 3D Solid Elements ..................................................................................................... 30
Chapter 5 – A Study of Element Modeling Techniques: Dynamic Analysis of Cantilever
          Beam ............................................................................................................................... 31
  5.1 Geometry, Loading and Boundary Conditions .......................................................... 31
  5.2 Hierarchic Modeling of a Cantilever Beam ............................................................... 32
  5.3 Time Discretization Study ........................................................................................ 33
  5.4 Mesh Density Study .................................................................................................... 35
  5.5 Material Properties Study ........................................................................................ 39
  5.6 Element Study ............................................................................................................ 40
  5.7 Calculating the Exact Solution for a Cantilever Beam ............................................... 41
### List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Stiffened Plate Panel</td>
<td>13</td>
</tr>
<tr>
<td>2</td>
<td>Sample of High Speed Slam, Daidola et al [6]</td>
<td>18</td>
</tr>
<tr>
<td>3</td>
<td>Conventions Used for 2-node Hermitian Beam Elements, ADINA R&amp;D [1]</td>
<td>26</td>
</tr>
<tr>
<td>4</td>
<td>Iso-Beam Elements, ADINA R&amp;D [1]</td>
<td>27</td>
</tr>
<tr>
<td>5</td>
<td>Typical 2D Solid Elements</td>
<td>28</td>
</tr>
<tr>
<td>6</td>
<td>Typical Plane Stress Idealizations, ADINA R&amp;D [1]</td>
<td>28</td>
</tr>
<tr>
<td>7</td>
<td>Shell Elements, ADINA R&amp;D [1]</td>
<td>29</td>
</tr>
<tr>
<td>8</td>
<td>Typical 3D Typical Elements</td>
<td>30</td>
</tr>
<tr>
<td>9</td>
<td>Illustration of Modeling a Cylinder Using 3D Element, ADINA R&amp;D [1]</td>
<td>30</td>
</tr>
<tr>
<td>10</td>
<td>Cantilever Beam</td>
<td>31</td>
</tr>
<tr>
<td>11</td>
<td>Hierarchic Modeling of a Cantilever Beam</td>
<td>32</td>
</tr>
<tr>
<td>12</td>
<td>Time Discretization Study</td>
<td>33</td>
</tr>
<tr>
<td>13</td>
<td>Time Discretization Study – Small Time Steps</td>
<td>34</td>
</tr>
<tr>
<td>14</td>
<td>One Dimensional Mesh Study</td>
<td>35</td>
</tr>
<tr>
<td>15</td>
<td>Beam Mesh Study – 2D Plane Stress Elements (L=0.1m)</td>
<td>36</td>
</tr>
<tr>
<td>16</td>
<td>Beam Mesh Study – 2D Plane Stress Elements (L=1m)</td>
<td>36</td>
</tr>
<tr>
<td>17</td>
<td>Beam Mesh Study – 2D Plane Stress Elements (L=5m)</td>
<td>37</td>
</tr>
<tr>
<td>18</td>
<td>Beam Mesh Study – 2D Plane Stress Elements (L=10m)</td>
<td>37</td>
</tr>
<tr>
<td>19</td>
<td>Three Dimensional Mesh Study</td>
<td>38</td>
</tr>
<tr>
<td>20</td>
<td>Material Density Study</td>
<td>39</td>
</tr>
<tr>
<td>21</td>
<td>Young’s Modulus Study</td>
<td>39</td>
</tr>
<tr>
<td>22</td>
<td>Element Type Study</td>
<td>40</td>
</tr>
<tr>
<td>23</td>
<td>Forces Acting on Beam Element</td>
<td>41</td>
</tr>
<tr>
<td>24</td>
<td>First 8 Modes of Cantilever Beam</td>
<td>45</td>
</tr>
<tr>
<td>25</td>
<td>One DOF System</td>
<td>47</td>
</tr>
<tr>
<td>26</td>
<td>FEA vs. Analytical Solution, L=10m (First Mode)</td>
<td>50</td>
</tr>
<tr>
<td>27</td>
<td>FEA vs. Analytical Solution, L=10m (First 2 Modes)</td>
<td>50</td>
</tr>
<tr>
<td>28</td>
<td>FEA vs. Analytical Solution, L=10m (First 3 Modes)</td>
<td>51</td>
</tr>
<tr>
<td>29</td>
<td>FEA vs. Analytical Solution, L=10m (First 4 Modes)</td>
<td>51</td>
</tr>
<tr>
<td>30</td>
<td>FEA vs. Analytical Solution, L=10m (First 5 Modes)</td>
<td>52</td>
</tr>
<tr>
<td>31</td>
<td>FEA vs. Analytical Solution, L=10m (First 6 Modes)</td>
<td>52</td>
</tr>
<tr>
<td>32</td>
<td>FEA vs. Analytical Solution, L=10m (First 7 Modes)</td>
<td>53</td>
</tr>
<tr>
<td>33</td>
<td>FEA vs. Analytical Solution, L=10m (First 8 Modes)</td>
<td>53</td>
</tr>
<tr>
<td>34</td>
<td>FEA vs. Analytical Solution, L=5m (First 8 Modes)</td>
<td>54</td>
</tr>
<tr>
<td>35</td>
<td>Cantilever Beam</td>
<td>55</td>
</tr>
<tr>
<td>36</td>
<td>Simplified Model for the Cantilever Beam</td>
<td>57</td>
</tr>
<tr>
<td>37</td>
<td>Approximate Solution Scheme for a Cantilever Beam</td>
<td>59</td>
</tr>
<tr>
<td>38</td>
<td>Beam Length Study – FEA vs. Analytical Solutions</td>
<td>61</td>
</tr>
<tr>
<td>39</td>
<td>FEA vs. Approximate Solution – Young’s Modulus Study</td>
<td>62</td>
</tr>
<tr>
<td>40</td>
<td>FEA vs. Approximate Solution – Material Density Study</td>
<td>62</td>
</tr>
<tr>
<td>41</td>
<td>Stiffened Plate Geometry and Principal Dimensions</td>
<td>63</td>
</tr>
<tr>
<td>42</td>
<td>Stiffened Plate Boundary Conditions</td>
<td>64</td>
</tr>
<tr>
<td>43</td>
<td>Applied Load on the Stiffened Panel</td>
<td>64</td>
</tr>
</tbody>
</table>
Figure 44: Buckling Modes of a Stiffened Panel, Hughes [12].............................................. 65
Figure 45: Stiffened Plate Models .......................................................................................... 67
Figure 46: Plate Thickness Study – Clamped Boundary Conditions.................................. 68
Figure 47: Plate Thickness Study – Hinged Boundary Conditions ........................................ 68
Figure 48: Stiffener Width Study – Clamped Boundary Conditions................................. 69
Figure 49: Stiffener Width Study – Hinged Boundary Conditions ........................................ 69
Figure 50: Stiffener Height Study – Clamped Boundary Conditions................................. 70
Figure 51: Stiffener Height Study – Hinged Boundary Conditions ........................................ 70
Figure 52: Effect of Weight Increase on Maximum Displacement ....................................... 72
Figure 53: Stiffened Plate Geometry for the Approximate Solution ................................... 73
Figure 54: Stress Distribution in Stiffened Plate ................................................................. 76
Figure 55: Effective Breadth of Stiffened Panels ............................................................... 77
Figure 56: Deflection at the Center of a Stiffened Plate, Schade [22]............................... 80
Figure 57: Dynamic Magnification Ratio as a Function of the Panel’s Length .................... 83
Figure 58: Approximate solution for the stiffened panel dynamic response ..................... 84
Figure 59: FEA Displacement Solution for the Detailed Example ...................................... 92
List of Tables

Table 1: Various Finite Elements models for the Cantilever Beam Analysis .......................... 32
Table 2: Correction Factors Calculations of the Cantilever Beam Mass ................................ 48
Table 3: Mass Correction factors .................................................................................. 48
Table 4: Modifying the Approximate Solution for different Beam Boundary Conditions ... 59
Table 5: Beam Approximate Solution Test Cases .......................................................... 60
Table 6: Stiffened Plate Geometry ................................................................................ 63
Table 7: Hierarchic Modeling of a Stiffened Plate .......................................................... 66
Table 8: Plate Thickness Study - Comparison with Static Results ................................. 71
Table 9: Stiffener Width Study - Comparison with Static Results ..................................... 71
Table 10: Stiffener Height Study - Comparison with Static Results ................................. 71
Table 11: Effective Breadth of Stiffened Panel ................................................................. 77
Table 12: Types of Stiffening with Applicable Formulas for Parameters ......................... 79
Table 13: K Values for Case 1 – All Four Edges Simply Supported ................................. 81
Table 14: K Values for Case 2 – Long Edges Supported, Short Edges Fixed ..................... 81
Table 15: K Values for Case 3 – Long edges fixed, Short Edges Supported ......................... 81
Table 16: K Values for Case 4 – All Edges Fixed .............................................................. 82
Table 17: Testing the Stiffened Panel Approximate Solution .............................................. 86
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>Length of panel</td>
</tr>
<tr>
<td>A</td>
<td>Area</td>
</tr>
<tr>
<td>b</td>
<td>Breadth of panel</td>
</tr>
<tr>
<td>B</td>
<td>Strain displacement matrix</td>
</tr>
<tr>
<td>C</td>
<td>Damping matrix</td>
</tr>
<tr>
<td>E</td>
<td>Young’s modulus</td>
</tr>
<tr>
<td>f</td>
<td>Force per unit surface</td>
</tr>
<tr>
<td>F</td>
<td>Force</td>
</tr>
<tr>
<td>F₀</td>
<td>Peak force in impulse loading</td>
</tr>
<tr>
<td>h</td>
<td>Stiffener height</td>
</tr>
<tr>
<td>H</td>
<td>Displacement interpolation matrix</td>
</tr>
<tr>
<td>i</td>
<td>Unit stiffness</td>
</tr>
<tr>
<td>I</td>
<td>Moment of inertia (of a cross section), impulse</td>
</tr>
<tr>
<td>k</td>
<td>Structure stiffness</td>
</tr>
<tr>
<td>K</td>
<td>Stiffness matrix</td>
</tr>
<tr>
<td>L</td>
<td>Length</td>
</tr>
<tr>
<td>m</td>
<td>Mass</td>
</tr>
<tr>
<td>m̅</td>
<td>Mass per unit length</td>
</tr>
<tr>
<td>M</td>
<td>Mass matrix, Moment</td>
</tr>
<tr>
<td>P</td>
<td>Pressure</td>
</tr>
<tr>
<td>P₀</td>
<td>Peak pressure in impulse loading</td>
</tr>
<tr>
<td>R</td>
<td>Load vector</td>
</tr>
<tr>
<td>Rₐ</td>
<td>Body forces</td>
</tr>
<tr>
<td>Rₛ</td>
<td>Surface forces</td>
</tr>
<tr>
<td>R₁</td>
<td>Initial stresses</td>
</tr>
<tr>
<td>Rₐc</td>
<td>Concentrated forces</td>
</tr>
<tr>
<td>S</td>
<td>Space between stiffeners</td>
</tr>
<tr>
<td>t</td>
<td>Time, Thickness of plate</td>
</tr>
<tr>
<td>Tₐ</td>
<td>Applied load duration</td>
</tr>
<tr>
<td>U</td>
<td>Displacement</td>
</tr>
<tr>
<td>Ẋ</td>
<td>Velocity</td>
</tr>
<tr>
<td>Ẋ</td>
<td>Acceleration</td>
</tr>
<tr>
<td>Ẋ</td>
<td>Virtual displacement</td>
</tr>
<tr>
<td>V</td>
<td>Shear force</td>
</tr>
<tr>
<td>v</td>
<td>Volume</td>
</tr>
<tr>
<td>w</td>
<td>Stiffener width</td>
</tr>
<tr>
<td>w(tip)ₘₐₓ</td>
<td>Maximum tip displacement</td>
</tr>
<tr>
<td>W</td>
<td>Deflection at the center of the panel</td>
</tr>
<tr>
<td>β</td>
<td>Frequency ratio</td>
</tr>
<tr>
<td>λ</td>
<td>Effective breadth</td>
</tr>
<tr>
<td>ν</td>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>ε</td>
<td>Strain</td>
</tr>
<tr>
<td>Ɛ</td>
<td>Virtual strain</td>
</tr>
<tr>
<td>η</td>
<td>Torsion coefficient</td>
</tr>
<tr>
<td>ω</td>
<td>Natural free-vibration frequency</td>
</tr>
<tr>
<td>ωₐ</td>
<td>Applied load frequency</td>
</tr>
<tr>
<td>ρ</td>
<td>Density, virtual side ratio</td>
</tr>
<tr>
<td>σ</td>
<td>Stress</td>
</tr>
</tbody>
</table>
Chapter 1 – Introduction

1.1 Ship Structural Design

The size and principal characteristics of a ship are determined primarily by its mission, intended service, and cost. In addition to basic functional considerations, there are requirements such as stability, low resistance, high propulsive efficiency, good seakeeping, and various navigational restrictions on draft or beam, all of which influence the choice of dimensions and form. The ship's structure must be designed within these and other basic constraints, to sustain all the loads expected to arise in its seagoing environment. In contrast to land structures, the ship does not rest on a fixed foundation, but derives its entire support from buoyant pressures exerted by a dynamic and ever changing ocean environment which plays the roles of both friend and foe for the ship.

The structural components of a ship are frequently designed to perform a multiplicity of functions in addition to that of providing the structural integrity of the ship. Furthermore, many strength members serve dual functions. For example, bulkheads that contribute substantially to the strength of the hull may also serve as watertight boundaries of internal compartments.

The loads that the ship structure must be designed to withstand have many sources. There are static components which consist principally of the weight and buoyancy of the ship in calm waters. There are dynamic components caused by wave induced motions of the ship, and by slamming, as well as vibratory loads by the propeller and machinery, all of which are of different frequency ranges. Furthermore, the loads imparted by the sea are random in nature, and therefore, the ship's structural behavior can be best expressed in probabilistic terms.
Four principal mechanisms are recognized as causing most of the cases of ship structural failure, aside from collision or grounding. These modes of failure are as follows:

- Excessive tensile or compressive loads.
- Buckling due to compressive or shear instability.
- Local concentrated, thermal, or impact loads.
- Fatigue cracking.

The general problem of ship structural design consists of the selection of material types, frame spacing, frame and stiffener sizes and plate thicknesses. This becomes an integrated part of the design spiral. It is convenient to divide the loads acting on the ship structure into four main categories, based partly upon the nature of the load and partly upon the ship's response:

1. Static loads are those that change only when the weight of the ship or its weight distribution changes. These include:

   - Weight of the ship and its contents.
   - Static buoyancy of the ship at rest or moving.
   - Thermal loads resulting from temperature gradients within the hull.
   - Concentrated loads caused by dry-docking or grounding.

2. Low frequency dynamic loads are those that vary in time with periods ranging from a few seconds to several minutes. They do not result in any significant resonant amplification of the stress induced in the structure. These can be separated into the following components:

   - Wave induced hull pressure variations.
   - Hull pressure variations caused by transient ship motion.
   - Inertial reactions resulting from the acceleration of the mass of the ship and its content.
3. High frequency dynamic loads are time varying loads of sufficiently high frequency that may induce vibratory response of the ship structure. Some of the existing loads may be quite small in magnitude, but as a result of resonant amplification, they may give rise to excessive stress and deflections. Examples of such dynamic loads include the following:

- Hydrodynamic loads induced by propulsive devices.
- Loads imparted to the hull by reciprocating or unbalanced machinery.
- Hydro-elastic load resulting from interaction of appendages with the flow passing the ship.
- Wave induced loads due primarily to short waves whose frequency of encounter overleaps the lower natural frequencies of hull vibration, called springing.

4. Impact loads are those resulting from slamming or wave impact on the bow, including the effects of green water on deck. In a naval ship, weapon recoil constitutes a very significant source of impact loads. Impact loads may induce transient hull vibration called whipping.

5. Specialized operational loads may be the dominant one for certain ship types. Examples of such loads, which may be either static or dynamic, are:

- Ice loads in the case of a vessel intended for icebreaking or arctic navigation.
- Loads caused by impact with other vessels, as in the case of tugs and barges.
- Impact of cargo handling equipment.
- Structural thermal loads imposed by special cargo carried at extreme temperature and/or pressure.
- Sloshing and impact loads on internal structures caused by movements of liquids in tanks.
- Aircraft or helicopter landing.
1.2 Stiffened Plate Panel and Ship’s Structure

Historically, the development of the stiffened structural form is one of slow growth of experiments by anonymous builders. It is known that the Egyptians, at least 5,000 years ago, developed a craft made of planks fastened around a wooden framework using much the same principles as are employed today. Also, ancient Viking ships were made of planks which were tied on the inside to ribs.

The stiffened plate panel (or simply “stiffened plate”) forms the backbone of most of a ship’s structure. It is by far the most commonly used structural element in a ship; appearing in decks, bottoms, bulkheads and side shell. Due to their simplicity of fabrication and excellent strength to weight ratio, stiffened plates are also widely used for construction of offshore structures, rail/road bridges, aircraft structures and many other applications.

The definition of a stiffened plate (also called stiffened panel or gross panel), for the purpose of this thesis, is a plate which has stiffeners running in two orthogonal directions. This panel is bounded by other structural members, which have significantly greater stiffness in the planes of the loads when compared to the plate and its stiffeners. These boundaries would be provided by a structure, such as transverse bulkheads, longitudinal bulkheads, side shells or large longitudinal girders (e.g., the keel).

Figure 1: Stiffened Plate Panel
The stiffened plate, which is intended to provide water-tightness and contribute significantly to the hull girder longitudinal and transverse strength, must be designed to withstand primary stress caused by hull girder bending, secondary stress caused by bending from local loading of the plate-stiffener combination and tertiary stresses caused by bending of the plate itself due to local lateral load.

The primary purpose of the panel is to absorb out of plane (or lateral) loads and distribute those loads to the ship’s primary structure. It also serves to carry part of the longitudinal bending stress because of the orientation of the stiffeners. Deck panels tend to experience large in-plane compression and minor lateral pressure. Bottom panels experience high in-plane compression and tension, but usually with very significant lateral pressure. The amount of in-plane compression or tension experienced depends on the location of the panel.

Since the beginning of the ship building history, the structural design of commercial ships was primarily rule based, where rules were formulated on the basis of practical experience and empirical studies. The US Navy used analyzed the ship’s structure using "first principles", but with very simplistic physical models. The modern computer enables us to use finite element codes to analyze the ship’s structure using “first principles” with more realistic physical models and since recent years the finite element method is widely used for ship structural analysis.

The actual ship structure is very complicated with numerous details and various structural members. Therefore, a wide range of modeling techniques are available for the analysis of stiffened plates; each model is based upon different assumptions in order to simplify the mathematical model and to consume less computational resources.
1.3 Dynamic forces on the Ship's Structure and Slamming

As discussed before, a ship traveling in sea undergoes various load conditions and dynamic forces. Although the various forces change significantly in time, most of them can be considered to be quasi-static because of their very low frequency. One exceptional phenomenon is the slamming, which is considered to be the most violent attack of the sea on a vessel. If the bow of the ship hits the water surface with sufficient high relative velocity, then a high impulsive load, a slam, will be experienced. Slamming cannot be modeled as a quasi-static phenomenon as wave bending. It is an impulsive phenomenon involving extreme pressure acting over a body surface for a very short time period.

The dynamic response of a structure to extreme hydrodynamic loads, such as slamming, is a highly transient and non-linear process. The damage sustained by a vessel due to slamming can manifest itself in many forms, from deformed shell plating, distorted and buckled longitudinal girders and frames, to fatigue cracking. For the most part, damage is sustained by the vessel's tertiary structure at the location of the impact, but the secondary and primary structures are affected as well. The secondary structure can be damaged by the direct action of the impulse forces or by the high frequency wave propagation that accompany the slamming (whipping phenomena). The primary structure is usually only affected by the whipping effects.
Slamming is associated with a sudden change of the acceleration of the ship. The acceleration, as well as the motion, is largest at the bow, and the most violent sudden change of the acceleration will occur at this place. Three types of stress are generated by slamming:

- The most obvious one and the most frequently described is that caused by high pressure on the plates under the forefoot. According to Szebejely [24], the part most susceptible to damage is the area of the bottom from 10% to 25% of the ship’s length. In the transverse direction, the keel to 25% of the beam is the most vulnerable part.

- Besides the sudden deceleration of the bow, there is also an elastic vibration which is generated by the sudden build-up of pressure (generally called “blow”). This vibration might also damage the superstructure; severe stress in light superstructure may result in cracked plates and loose rivets.

- The third type of stress generated by slamming increases the sagging stress produced amidships by normal wave action by some 30% (Szebejely [24]).

The damage on the bottom stiffened plates is caused by the pressure developed when slamming takes place, while the damage on the deck stiffened plates is caused by the weight of the payload and the deceleration this payload undergoes during slamming.

These extreme environmental forces drive structural design in one direction, towards more substantial and heavier structures. The unfortunate consequences to a vessel include the effects of weight addition, reduced payload, increased construction costs, and reduced vessel speed. This is a particularly acute problem in high-speed combatants and patrol crafts, which are highly weight critical.
1.4 Objective

The objective of this research is to analyze a stiffened panel's response under dynamic impulsive loading conditions with various element types and to evaluate and compare the results obtained in the different finite element models. More specifically, we want to obtain the maximum displacement in the center of the stiffened plate panel caused by the sudden pressure raise during slamming and impact loads.

The various results, obtained using the different simplified models, will be compared to the results obtained using a more comprehensive model. The goal is to conclude whether certain simplified finite element models can provide satisfactory results despite the assumptions involved.

In order to reach our goal, a two step approach will be adopted; firstly, we will analyze a simple cantilever beam subjected to a dynamic tip load to get familiar with the various finite element modeling techniques. Secondly, we will analyze a stiffened plate panel subjected to a dynamic lateral pressure.

In addition to the finite element analyses, we will compare the numerical solutions to analytical solutions. For the cantilever beam we will derive two analytical solutions:

- A detailed exact solution that must be calculated using a computer.
- A closed-form approximate solution that can be obtained very quickly using a calculator only.

For the stiffened plate we will derive only a closed-form approximate solution. This kind of approximation can be used to check and verify a detailed finite element solution, performed by a third party, in some quick steps without repeating the whole detailed analysis again.
Chapter 2 – Literature

2.1 Slamming of Ships

According to Daidola and Mishkevich [6], the most significant factors which govern or influence slamming conditions are the length of the ship, sea severity, ship speed course angle relative to predominant sea, ship loading condition, overall ship form as it affects ship motion and also fullness or flatness of bottom forward.

Because of the several random variables involved in both wave motion and wave-induced motion of ships, it is necessary to acquire actual data from shipboard measurements. Several attempts to measure slamming effects on ships are described in the literature. Various pressure gages, accelerometers, and stain gages were placed throughout ships to record external pressure on the hull, accelerations related to bow pitching and heaving forces. A typical example of these types of measurements is presented in Figure 2:

Figure 2: Sample of High Speed Slam, Daidola et al [6]
Note that the flat portions in the pressure transducer records (labeled W.P) indicate that the bow has lifted completely out of the water and that the pressure gages are sensing atmospheric pressure. At the ends of these flat portions, there is a sharp discontinuity in pressure results from a slam.

Szebejely [24] and Ochi [16] have found that in regular waves, slamming generally occurred when the ship model and the impact surface were nearly parallel. Szebejely [24] showed that three conditions must exist for a slam to occur:

- Bow emergence.
- A certain magnitude of relative velocity between the bow and the wave surface.
- Unfavorable phase between bow motion and wave motion.

A fourth criterion mentioned by Szebejely [24] affecting the severity of slamming was the angle between the wave surface and keel.

Ochi [16] examined the condition leading to slamming from tests in irregular waves and found that bow emergence was a prerequisite for bottom slamming. However, bow emergence was not a sufficient cause for slamming and it appears that a critical relative vertical velocity exists between bow and wave, below which slamming does not occur. This critical relative vertical velocity equals to $0.096(g/L)^{0.5}$, where $L$ is the length of the ship in meters and $g=9.81$ m/sec$^2$ is the gravitational acceleration. Ochi has also shown that slamming severity increases with wave severity, if other conditions remain unchanged.

Based on laboratory and full-scale tests (Henry at el [10]), it appears that:

- The pulse width during a slam varies from a small fraction of a millisecond (0.05 msec.) to 30 milliseconds. The most common slams are 20msec long.
- The peak pressure of significant slams ranges from 300 psi to 1000 psi (20-68 Atm).
- The pressure rise times measured are in the range of 50 to several hundred microseconds, depending on the frequency response characteristic to the recording equipment.
- Results of laboratory drop tests cannot be scaled for analytical application to full-scale ships.
2.2 Finite Element Modeling of a Stiffened Plate Panel

In the modeling and analysis of a stiffened plate we can distinguish between two different research areas; one field is dedicated toward modeling the whole stiffened plate panel as one structural member in the overall ship structural analysis, mainly to analyze the ship’s primary stresses. The other research field concentrates on the panel itself and obtains the stresses and deformations within the panel itself, mainly due to secondary/tertiary stresses and local loads.

Elbatouti et al. [7] have used the orthotropic model of the stiffened panel in which the panel is substituted with an equivalent rigid plate. Similar to the orthotropic model, Paik et al [18] have developed a new stiffened panel model in which the stiffened panel is replaced by a plate of optimal thickness providing the same buckling or collapse strength as the parent stiffened plate.

Satish et al [21] have developed a stiffened plate element for the three-dimensional finite element analysis of the complete ship structure. Their element is a combination of Allman’s plane stress element and Discrete Kirchhoff-Mindlin Triangular plate bending element.

Gangadhara et al [8] obtained the transient dynamic response of composite stiffened plates and shells by using an 8-noded curved quadratic isoparametric element for the shell and a 3-noded curved beam element for the stiffener.

Chen et al [4] studied the bucking behavior of stiffened steel plates and compared it to laboratory measurements. All the elements in the panel's geometry were modeled using 4-node plate bending elements and the panel material behavior was modeled by an elastic-plastic von Mises kinematic strain-hardening constitutive model.
Salomon [20] evaluated the following finite element models for a stiffened plate subjected to static load:

- Plate modeled using shell elements and stiffeners modeled using Hermitian beam elements.
- Plate modeled using shell elements and stiffeners modeled using iso-beam elements.
- Both the plate and the stiffeners are modeled using shell elements.
- Both the plate and the stiffeners are modeled using 3D brick elements.

His research points out that the usage of the classical Hermitian beam to model the stiffeners yield excellent results.

Hu et al [11] conducted non-linear analyses for various stiffened panels to simulate a bucking test procedure. Their stiffened panels were expected to experience large displacements and plastic deformations and they used ADINA® to perform their analyses. The stiffened panels were modeled using 4-node shell elements for both the plate and the stiffener. The material property of the panel was idealized as bilinear elastic-perfectly plastic with von Mises yield condition.

Prusty et al [19] developed a method to predict the failure load on laminated composite stiffened panels under various loading conditions. For their research, the plate was modeled using 8-node isoparametric shell elements and the stiffeners were modeled using 3-node curved beam elements.

Louca et al [14+15] carried out dynamic analyses on both stiffened and unstiffened plates subjected to impulsive loading. Their load profile was symmetric triangular with peak value at $t = 25$ msec. In order to model the stiffened panel they used 4-node shell elements for the plate and beam elements for the stiffeners.

As presented above, the same physical geometry can be modeled using various elements and modeling techniques. In our FEA we will explore the difference between some of the most common modeling techniques.
3.1 The Process of Finite Element Analysis

The finite element method is used to solve physical problems in engineering analysis and design. The finite element method is the most widely applied computer simulation in engineering and it is integrated with the modern CAD/CAM software as part of the computer aided engineering (CAE) concept.

The physical problem in engineering analysis usually involves an actual structure, with known material properties, subjected to certain loads and boundary conditions. The idealization of the physical problem to a mathematical model requires certain assumptions that together lead to differential equations governing the mathematical model. The finite element analysis solves this mathematical model. Since the finite element solution is obtained numerically, some error is produced but this error can be reduced by refining the solution parameters (finer mesh, smaller time steps in transient or non-linear analysis, etc).

Refining the solution parameters requires more computational resources but even with endless computational power, the finite element solution will solve only the defined mathematical model and we cannot expect any more information in the prediction of the physical phenomena than the information contained in the mathematical model.

It is impossible to reproduce, even in the most refined finite elements analysis, the exact physical solution, but with an appropriate mathematical model and reasonable mesh we can obtain a very good solution that will be accurate enough for the engineering application.

The modern finite elements codes, as well as the CAD software that integrates finite element capabilities within its features, have a nice user-friendly interface, but this colorful look might be deceiving. These powerful tools, used by unqualified personnel, might produce very poor or false results that might lead to wrong engineering decisions.
3.2 The Principle of Virtual Displacement

The principle of virtual displacement (also known as the principle of virtual work) is used as the basis of the finite element solution for solids. The derivation of this principle appears in numerous finite elements textbooks and it states the following:

Given a 3D body with volume $V$, total surface area $S$, prescribed displacements on part of the area $S_u$ and subjected to surface tractions $t^s$ on the surface area $S_f$ (such that $S_u \cap S_f = 0$ and $S_u \cup S_f = S$). In addition, the body is subjected to externally applied body forces $f_b$ per unit volume and concentrated load $R_c$ at a number of points. Then, for this given body, the following equation is valid:

\[ \int_V \varepsilon^T \, dV = \int_V \bar{U}^T f^b \, dv + \int_{S_f} U^{s_f} f^{s_f} \, ds + \sum_i \bar{U}^{i^T} R^{i} \]  

(1)

where $\bar{U}$ are virtual displacements and $\varepsilon$ are the corresponding virtual strains.

The virtual displacements $\bar{U}$ are a continuous virtual displacement field such that $\bar{U} = 0$ on surface $S_u$ and the virtual strains $\varepsilon$ are calculated by differentiating the assumed virtual displacements.

The left hand side of equation (1) expresses the total internal virtual work while the right hand side of this equation represents the total external virtual work.

We can calculate a closed-form analytical solution to very simple geometries only by integrating the various terms in equation (1). When we consider more complex geometries, then we have to divide the body into a finite number of elements interconnected at nodal points on the elements boundaries. In practical problems, the body is divided into many thousands of elements and it is practically impossible to solve the resulting equations without a computer.
3.3 Equilibrium Equations in Static Analysis

In the finite element analysis we approximate the structure as an assemblage of \( m \) discrete finite elements interconnected at nodal points on the element boundaries. For every element \( m \) we can construct a displacement interpolation matrix \( H \) such that

\[
\mathbf{u}^{(m)}(x, y, z) = H^{(m)}(x, y, z)\hat{\mathbf{U}}
\]

(2)

where \( \hat{\mathbf{U}} \) is a vector of the three global displacement components.

The corresponding element strains can be obtained by the following equation:

\[
\varepsilon^{(m)}(x, y, z) = B^{(m)}(x, y, z)\hat{\mathbf{U}}
\]

(3)

where \( B \) is the strain-displacement matrix (the rows of \( B^{(m)} \) are obtained by appropriately differentiating and combining rows of the matrix \( H^{(m)} \)).

The stiffness matrix \( K \) is obtained by the following term:

\[
K = \sum_{m} K^{(m)} = \sum_{m} \int B^{(m)T} C^{(m)} B^{(m)} dV^{(m)}
\]

(4)

where \( C \) is the stress-strain law of the material.

The fundamental relationship that links the displacements, the stiffnesses and the loads is:

\[
K\mathbf{U} = \mathbf{R}
\]

(5)

The load vector \( \mathbf{R} \) includes the effects of the element body forces, \( R_B \), surface forces, \( R_S \), initial stress forces, \( R_I \) and nodal concentrated load \( R_C \):

\[
\mathbf{R} = R_B + R_S - R_I + R_C
\]

(6)

\[
R_B = \sum_{m} R_B^{(m)} = \sum_{m} \int H^{(m)} f^{B(m)} dV^{(m)}
\]

(7)

\[
R_S = \sum_{m} R_S^{(m)} = \sum_{m} \int H^{S(m)T} f^{S(m)} dS^{(m)}
\]

(8)

\[
R_I = \sum_{m} R_I^{(m)} = \sum_{m} \int B^{(m)T} t^{I(m)} dV^{(m)}
\]

(9)
3.4 Equilibrium Equations in Dynamic Analysis

Unlike the linear-static FEA, the transient-dynamic FEA has to analyze the stresses and the kinematics of the body over the analyzed time period. Therefore, we have to divide the time span into discrete time steps and to obtain the solution for each time step.

If the loads are applied rapidly, relative to the natural frequencies of the body, then we have to consider both the mass properties of the body and its damping properties.

The equilibrium equation can be written in the following form:

\[ M \cdot \ddot{U} + C \cdot \dot{U} + K \cdot U = R \]  \hspace{1cm} (10)

where \( M \) is the mass matrix, \( C \) is the damping matrix and \( K \) is the stiffness matrix of the structure:

\[ M = \sum_m M^{(m)} = \sum_m \int \rho^{(m)} H^{(m)\top} H^{(m)} dv^{(m)} \]  \hspace{1cm} (11)

\[ C = \sum_m C^{(m)} = \sum_m \int k^{(m)} H^{(m)\top} H^{(m)} dv^{(m)} \]  \hspace{1cm} (12)

\[ K = \sum_m K^{(m)} = \sum_m \int B^{(m)\top} C^{(m)} B^{(m)} dv^{(m)} \]  \hspace{1cm} (13)

\( U \), \( \dot{U} \) and \( \ddot{U} \) are the displacement, the velocity and the acceleration vectors of the finite element assemblage and \( R \) is the vector of externally applied loads.

Solving the above governing differential equation can be very expensive in terms of computational resources; therefore, in practice we are choosing from special methods to obtain the solution such as direct integration and mode superposition.

There are various direct integration methods (central difference method, Houbolt method, Wilson θ method, Newmark method, etc) but all the methods are based on the same basic concept: The equations in (10) are integrated using a numerical step-by-step procedure where the solution at time \( t \) is used to obtain the solution at time \( t+\Delta t \) where \( \Delta t \) represents the time step interval.

Instead of trying to satisfy equation (10) at any time along the analyzed time span, we are satisfying the equation only at a finite number of time intervals.
Chapter 4 – Element Descriptions and Assumptions

4.1 Hermitian Beam Elements

The Hermitian beam element is a 2-node element with a constant cross section and 6 degrees of freedom at each node. Using Hermitian beam elements is the most economic method to model a beam.

![Diagram of 2-node Hermitian Beam Elements]

Figure 3: Conventions Used for 2-node Hermitian Beam Elements, ADINA R&D [1]

The displacements modeled by the Hermitian beam elements are:
- $\bar{v}$ - Cubic transverse displacement in s-direction.
- $\bar{u}$ - Linear longitudinal displacement in r-direction.
- $\bar{w}$ - Cubic transverse displacement in t-direction.
- $\bar{\theta}_r$ - Linear torsional displacement in r-direction.
- $\bar{\theta}_t$ and $\bar{\theta}_i$ - Rotations.

The forces modeled by the Hermitian beam element are:
- $S_1$ and $S_7$ – r-direction forces at node 1 and node 2, respectively.
- $S_2$ and $S_8$ – s-direction forces at node 1 and node 2, respectively.
- $S_3$ and $S_9$ – t-direction forces at node 1 and node 2, respectively.

The moments modeled by the Hermitian beam element are:
- $S_4$ and $S_{10}$ – r-direction moments at node 1 and node 2, respectively.
- $S_5$ and $S_{11}$ – s-direction bending moments at node 1 and node 2, respectively.
- $S_6$ and $S_{12}$ – t-direction bending moments at node 1 and node 2, respectively.

The Hermitian beam element is formulated based on the Bernoulli-Euler beam theory, corrected for shear deformation effects if requested and it is very economic.
4.2 Iso-Beam Elements

The general 3D iso-beam element is a 2, 3 or 4-node element with a constant rectangular cross section and 6 degrees of freedom at each node. The 3-node and 4-node elements can be curved, but the element nodes must initially lie in one plane (r-s plane).

![Iso-Beam Elements](image)

(a) 2-Node Iso-Beam  (b) 3-Node Iso-Beam  (c) 4-Node Iso-Beam

Figure 4: Iso-Beam Elements, ADINA R&D [1]

The element matrices and vectors are formulated using isoparametric interpolation and Gauss or Newton-cotes numerical integration is used to evaluate these matrices in all analyses.

In order to reduce the number of degrees of freedom, few 2D simple elements can be derived from the general 3D iso-beam element:

- Plane stress 2D beam element – Similar to the general 3D element but constrained to act only in YZ plane (out-of-plane stress $\sigma_{xx}$ is equal to zero).
- Plane strain 2D beam element – Similar to the general 3D element with the assumption that the out-of-plane strain $\varepsilon_{xx}$ is equal to zero.

Both the plane stress and plane strain beam elements have three degrees of freedom per node instead of six in the general 3D element, which makes them more efficient.

In the case of linear analysis of a straight beam, the Hermitian beam elements are more cost effective since they can describe cubic displacements with half the degrees of freedom. The Hermitian beam element can be used with various cross sections, while the iso-beam elements can be used only for rectangular cross sections. However, the iso-beam elements predict the shear deformation more accurately. Furthermore, we can represent curved beams by using 3-node and 4-node iso-beam elements.
4.3 2D Plane Stress Solid Elements

The 2D plane stress solid element is an isoparametric displacement-based finite element with 4 to 9 nodes per element.

![4-Node 2D Element](a) 4-Node 2D Element ![8-Node 2D Element](b) 8-Node 2D Element ![9-Node 2D Element](c) 9-Node 2D Element

Figure 5: Typical 2D Solid Elements

The 4-node solid element is the most economic but it is not suitable when bending effects are significant. Usually the 8-node and 9-node elements are most effective if the element is rectangular.

If the 2D plane stress element is defined in the YZ plane, then the following basic assumptions are valid:

- $\sigma_{xx} = 0$
- $\sigma_{xz} = 0$
- $\sigma_{yz} = 0$

The 2D plane stress element can be used to model thin structures like membrane sheets and beams, as demonstrated in the following Figure:

![Membrane Sheet](a) Membrane Sheet ![Cantilever Beam](b) Cantilever Beam

Figure 6: Typical Plane Stress Idealizations, ADINA R&D [1]
4.4 Shell Elements

The shell element is a 4, 8, 9 or 16-node isoparametric element that can be employed to model thick and thin general shell structures (The number of nodes on the element must be chosen according to the application).

![Shell Elements](image)

(a) 4-Node Shell  (b) 9-Node Shell  (c) 16-Node Shell

Figure 7: Shell Elements, ADINA R&D [1]

The 4-node shell element is considered to be the most effective one for analysis of general shells. This element does not lock and has a high predictive capability for both thin and thick shells.
The shell element is formulated treating the shell as a 3D continuum with the following assumptions:

- Material particles that originally lie on a straight line normal to the mid surface of the structure remain on that straight line during the deformation.
- The stress in the direction normal to the mid surface is zero.

Either 5 or 6 degrees of freedom can be assigned at a shell mid surface node. In most cases we should specify only 5 degrees of freedom, while the usage of 6 degrees of freedom should be limited to the following cases:

- Shell elements intersecting at an angle.
- Coupling of shell elements with other types of structural elements such as beam elements (e.g., in the modeling of a stiffened panel using shell and beam elements).
- Coupling of rigid links to the shell mid surface nodes.
- Imposing specific boundary conditions.
4.5 3D Solid Elements

The 3D solid element is a variable 4 to 27-node element applicable to general 3D analysis when the 3D state of stress is required or in special stress/strain conditions.

![Typical 3D Typical Elements](image)

(a) 4-Node 3D element  (b) 8-Node 3D Element  (c) 27-Node 3D Element

Figure 8: Typical 3D Typical Elements

The 3D solid element is usually a tetrahedral or a brick, as presented above, but many other prismatic shapes like pyramids are available mainly for transition zones. The 3D solid elements usually used are isoparametric displacement-based finite elements.

The 27-node element is the most accurate among all available 3D solid elements. However, the use of this element requires a lot of computational resources and can be very costly. The 4-node tetrahedral and the 8-node brick elements are very economic but they are not suitable when bending effects are significant. The 20-node 3D solid element is usually the most effective if the element is rectangular (undistorted) and the 10-node tetrahedral element is the most cost-effective if the element is tetrahedral.

![Illustration of Modeling a Cylinder Using 3D Element](image)

(a) Physical Problem  (b) 3D Finite Elements Model

Figure 9: Illustration of Modeling a Cylinder Using 3D Element, ADINA R&D [1]
Chapter 5 – A Study of Element Modeling Techniques: Dynamic Analysis of Cantilever Beam

5.1 Geometry, Loading and Boundary Conditions

In order to understand and quantify the difference between the various finite element modeling techniques, a simple geometry is to be considered:

A cantilever beam with a rectangular cross section is fixed at one end and subjected to a dynamic tip force perpendicular to the axis of the beam, as presented in Figure 10. The dynamic force is the first half of a sine wave with 40 msec. time period and the L/H ratio of the beam ranges from 1/10 to 1/1000 (The beam thickness and height, H, is 1 cm. throughout the whole study and the various L/H ratios were obtained by changing the length of the beam, L, from 10 cm. to 10 meters). The force duration of 20 msec is a good approximation of a typical slamming force and the material of the beam is considered to be elastic-isotropic with the approximate properties of structural steel. In all the finite element analyses, we track the maximum displacement that the free edge of the beam reaches within 3 seconds (This time limit is mainly to keep the usage of computation resources at a reasonable level).
5.2 Hierarchic Modeling of a Cantilever Beam

As presented in the previous chapter, there are numerous elements available and there are a few different ways to represent the physical geometry of a cantilever beam of rectangular cross section.

![Hierarchic Modeling of a Cantilever Beam](image)

Figure 11: Hierarchic Modeling of a Cantilever Beam

As presented above, the finite element model can be one dimensional, two dimensional or 3D. Table 1 lists the 11 different models we used in our study. In addition to the various finite element models, a few convergence analyses and two sensitivity analyses were performed.

Table 1: Various Finite Elements models for the Cantilever Beam Analysis

<table>
<thead>
<tr>
<th>One Dimensional Models</th>
<th>2D Models</th>
<th>3D Models</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hermitian beam model</td>
<td>4-node 2D plane stress model (regular &amp; incompatible mode)</td>
<td>8-node 3D brick element</td>
</tr>
<tr>
<td>2-node iso-beam model</td>
<td>9-node 2D plane stress model</td>
<td>27-node 3D brick element</td>
</tr>
<tr>
<td>3-node iso-beam model</td>
<td>4-node shell model</td>
<td></td>
</tr>
<tr>
<td>4-node iso-beam model</td>
<td>9-node shell model</td>
<td></td>
</tr>
</tbody>
</table>
5.3 Time Discretization Study

Transient dynamic finite element analysis has a tendency to consume a lot of computational resources because of two main reasons:

- A very complicated model with very fine mesh and many degrees of freedom.
- Dividing the analyzed time period into many discrete time steps.

The first reason is common for both static-linear analysis and dynamic analysis, but the second reason is unique for transient-dynamic or non-linear analyses.

The purpose of this study is to figure out the appropriate time stepping for our analysis. The time interval $\Delta t$ is to be established by the engineer and should be an optimum considering the analysis error and the usage of computational resources. For this purpose, the beam was modeled using 50 Hermitian beam elements and various time steps per second.

![Time Discretization Study](image)

Figure 12: Time Discretization Study.
Here is the same graph with zoom in on the time step below 6 msec:

![Graph of Time Discretization Study](image)

Figure 13: Time Discretization Study – Small Time Steps.

The force duration in our case is 20 msec. so any attempt to run the analysis with more than 20 msec for each time step will fail because the finite element code cannot notice the impulse force at all. When the time step decreases below 20 msec, the beam starts to deflect, but only below 4 msec the maximum displacement of the beam starts to stabilize near the same value.

From Figure 13, we can conclude that there is almost no practical need to decrease the time steps to less than 4 msec. To be on the safe side, the time discretization for all the analyses from now on will be 2 msec. per time step.

Another practical method to obtain accurate solution with large $\Delta t$ is to calculate the initial velocity of the tip, as a result of an impulsive load, and let the beam vibrate with these initial conditions. This method can reduce the number of time steps for the analysis and it is suitable only for very long and slender beams ($L/H > 100$) where the maximum deflection does not occur when the load is still acting on the beam. In Section 5.7 we will present an analytical solution based on this assumption.
5.4 Mesh Density Study

The meshing has a large impact on the computational resources that the analysis requires, as the meshing gets finer and the element size gets smaller, more computational resources are needed in order to complete the analysis. The same geometry of cantilever beam can be modeled in one, two or three dimensions and each time various element types can be used. The mesh density study was performed using Hermitian beam elements, 2D plane stress elements and 3D brick elements.

![Mesh Study - Beam Elements](image)

*Figure 14: One Dimensional Mesh Study*

As presented above, for one dimensional Hermitian beam element, the result converges very quickly and from 25 elements and above, there are no significant changes in the results regardless of the beam length.
Figure 15: Beam Mesh Study – 2D Plane Stress Elements (L=0.1m)

Figure 16: Beam Mesh Study – 2D Plane Stress Elements (L=1m)
Figure 17: Beam Mesh Study – 2D Plane Stress Elements (L=5m)

Figure 18: Beam Mesh Study – 2D Plane Stress Elements (L=10m)
In the two dimensional analyses, we compared the plane stress 4-node element with the 9-node element for four different beam lengths. As presented in Figures 15 to 18, the 2D 9-node element converges very quickly while the 2D 4-node element requires a very fine mesh to reach convergent. The 4-node incompatible mode elements behave much like the 9-node elements but with less usage of computational resources. The difference in the performance of the 4-node elements and the 9-node elements will be emphasized once again in the element type study.

The 3D 8-node elements behave very much like the regular 4-node 2D elements and fail to converge even in a very fine mesh. The phenomenon we just observed with the 2D 4-node and 3D 8-node elements is called locking. In essence, the locking problem arises because the interpolation functions used for an element are not able to represent zero (or very small) shearing or membrane strain. If the element cannot represent zero shearing strain, but zero or very small shearing strain does exist in the physical problem, then the element becomes very stiff as its thickness over length ratio.
5.5 Material Properties Study

The material properties sensitivity analysis was performed using Hermitian beam elements. In the first analysis, nine different beam lengths were analyzed by keeping all the parameters unchanged except for the material density. In the second analysis, the same beam lengths were analyzed by keeping all the parameters unchanged except for the Young's modulus.

![Material Density Study](image)

**Figure 20: Material Density Study**

![Young's Modulus Study](image)

**Figure 21: Young's Modulus Study**
5.6 Element Study

In this study, our beam was modeled with ten different element types in one, two and three dimensions. As presented below, all the beam elements (both Hermitian and Iso-beam), the 9-node 2D elements and the 27-node 3D elements give us the same approximate results. Later on, the same results will be plotted against two analytical solutions, a detailed exact solution and a closed form approximate solution.

The 2D 4-node elements and the 3D 8-node elements are too stiff and they failed to predict the maximum deformation of the beam. These two elements gave the same low results.
5.7 Calculating the Exact Solution for a Cantilever Beam

Step 1 – Calculating the Natural Frequencies of the Beam:

In order to derive the necessary equations, the following relation is used:

$$EI \cdot \frac{d^2w}{dx^2} = M \tag{14}$$

This equation relates the curvature of the beam to the bending moment at each section of the beam. Equation (14) is based upon the assumptions that the material is homogeneous, isotropic, and obeys Hooke's law and that the beam is straight and of uniform cross section. This equation is valid for small deflections only and for beams that are long compared to the cross-section dimensions, since the effects of shear deflection are neglected.

![Figure 23: Forces Acting on Beam Element](image)

The equation of motion for lateral vibrations of the beam is found by considering the forces acting on the element, presented in Figure 23, which is formed by passing two parallel planes A-A and B-B through the beam normal longitudinal axis. The vertical elastic shear force acting on section A-A is \( V \), and that on section B-B is \( V + \frac{\partial V}{\partial x} \, dx \). Shear forces acting as shown are considered to be positive.

The total vertical elastic shear force at each section of the beam is composed of two parts: the one caused by the static load including the weight of the beam and the one caused by the vibration. The part of the shear force caused by the static load balances the load exactly, so that there is no need to consider them in the derivation of the equation of the beam.
The sum of the remaining vertical forces acting on the element must equal the product of the mass of the element and the acceleration in the lateral direction:

$$\frac{\partial V}{\partial x} = -\rho \cdot A_{\text{cross-section}} \frac{\partial^2 w}{\partial t^2}$$ \hspace{1cm} (15)

The moments are taken about point 0 of the small element presented in Figure 23:

$$V = \frac{\partial M}{\partial x}$$

Other terms contain differentials of higher order and can be neglected. Substituting this in equation (15) gives:

$$-\frac{\partial^2 M}{\partial x^2} = \rho \cdot A_{\text{cross-section}} \frac{\partial^2 w}{\partial t^2}$$ \hspace{1cm} (16)

Substituting equation (14) gives:

$$-\frac{\partial^2}{\partial x^2} \left( EI \frac{\partial^2 w}{\partial x^2} \right) = \rho \cdot A_{\text{cross-section}} \frac{\partial^2 w}{\partial t^2}$$ \hspace{1cm} (17)

This equation is the basic equation for the lateral vibration of beam. The solution of this equation, if EI is constant, is of the form:

$$w = X(x) \cdot \left[ \cos(\omega_n t + \theta) \right]$$

in which X is a function of x only. Substituting

$$k^4 = \frac{\omega_n^2 \rho \cdot A_{\text{cross-section}}}{EI}$$ \hspace{1cm} (18)

and dividing equation (17) by \(\cos(\omega_n t + \theta)\):

$$\frac{d^4 X}{dx^4} = k^4 X$$ \hspace{1cm} (19)

where X is a function who satisfy the boundary conditions of the beam.

The following functions satisfy the required conditions and represent the solution of the equation:

$$X = A_1 \sin(kx) + A_2 \cos(kx) + A_3 \sinh(kx) + A_4 \cosh(kx)$$ \hspace{1cm} (20)

The solution can also be expressed in terms of exponential functions, but the trigonometric and hyperbolic functions usually are more convenient to use.
For beams having various support conditions, the constants $A_1$, $A_2$, $A_3$ and $A_4$ are found from the boundary conditions. In finding the solutions, it is convenient to write the equation in the following form in which two of the constants are zero for each of the usual boundary conditions:

$$X = A [\cos(kx) + \cosh(kx)] + B [\cos(kx) - \cosh(kx)] + C [\sin(kx) + \sinh(kx)] + D [\sin(kx) - \sinh(kx)]$$

In applying the boundary conditions, the following relations are used where a prime indicates derivative with respect to $x$:

- The deflection is proportional to $X$ and is zero at any rigid support.
- The slope is proportional to $X'$ and is zero at any fixed end.
- The moment is proportional to $X''$ and is zero at any free or hinged end.
- The shear is proportional to $X'''$ and is zero at any free end.

The required derivatives are:

$$X' = k [A [-\sin(kx) + \sinh(kx)] + B [-\sin(kx) - \sinh(kx)] + C [\cos(kx) + \cosh(kx)] + D [\cos(kx) - \cosh(kx)]]$$

$$X'' = k^2 [A [-\cos(kx) + \cosh(kx)] + B [-\cos(kx) - \cosh(kx)] + C [-\sin(kx) + \sinh(kx)] + D [-\sin(kx) - \sinh(kx)]]$$

$$X''' = k^3 [A [\sin(kx) + \sinh(kx)] + B [\sin(kx) - \sinh(kx)] + C [-\cos(kx) + \cosh(kx)] + D [-\cos(kx) - \cosh(kx)]]$$

For the most common boundary conditions, two of the constants are zero, and we get two equations containing two constants. These can be combined to give an equation which contains only the natural frequency and an unknown constant.

For a cantilever beam, with one fixed end and one free end, we obtain the following boundary conditions:

- $X = 0$ at the fixed end ($x = 0$).
- $X' = 0$ at the fixed end ($x = 0$).
- $X'' = 0$ at the free end ($x = 1$)
- $X''' = 0$ at the free end ($x = 1$)

43
The first boundary condition requires that \( A = 0 \) since the other constants are multiplied by zero at \( x = 0 \). The second condition requires that \( C = 0 \). From the third and fourth boundary conditions, the following equations are obtained:

\[
0 = B[- \cos(\kappa l) - \cosh(\kappa l)] + D[- \sin(\kappa l) - \sinh(\kappa l)]
\]
\[
0 = B[\sin(\kappa l) - \sinh(\kappa l)] + D[- \cos(\kappa l) - \cosh(\kappa l)]
\]

Solving each of these for the ratio \( D/B \) and equating, or making use of the mathematical condition that for a solution the determinant of the two equations must vanish, the following equation results:

\[
\frac{D}{B} = \frac{\cos(\kappa l) + \cosh(\kappa l)}{\sin(\kappa l) + \sinh(\kappa l)} = \frac{\sin(\kappa l) - \sinh(\kappa l)}{\cos(\kappa l) + \cosh(\kappa l)}
\]

This equation reduces to:

\[
\cos(\kappa l) \cdot \cosh(\kappa l) = -1
\]

The first 8 values that satisfy this equation are:

\[ k_1 l = 1.8751 \quad k_2 l = 4.6941 \quad k_3 l = 7.8548 \quad k_4 l = 10.9955 \]

\[ k_5 l = 14.1372 \quad k_6 l = 17.279 \quad k_7 l = 20.42 \quad k_8 l = 23.562 \]

The corresponding frequencies of vibration are found by substituting the length of the beam to find each \( k \) and then solving equation (24) for \( \omega_n \). The first 8 natural frequencies are:

\[
\omega_{n1} = 1.8751^2 \cdot \frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}
\]
\[
\omega_{n2} = 4.6941^2 \cdot \frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}
\]
\[
\omega_{n3} = 7.8548^2 \cdot \frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}
\]
\[
\omega_{n4} = 10.9955^2 \cdot \frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}
\]
\[
\omega_{n5} = 14.1372^2 \cdot \frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}
\]
\[
\omega_{n6} = 17.279^2 \cdot \frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}
\]
\[
\omega_{n7} = 20.42^2 \cdot \frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}
\]
\[
\omega_{n8} = 23.562^2 \cdot \frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}
\]
We can find more natural frequencies than the ones listed above just by finding more solution to equation 24, but the contribution of higher frequencies to the final result can be neglected without losing the accuracy of the solution. Later on we will plot the sum of the first 7 modes and the sum of the first 8 modes to prove this assumption right.

The shapes of the different modes are given in the following equation:

\[
Y_n(x) = C \cdot \left\{ \sin(k_n x) - \sinh(k_n x) + \frac{\sin(k_n l) + \sinh(k_n l)}{\cos(k_n l) + \cosh(k_n l)} \left[ \cosh(k_n x) - \cos(k_n x) \right] \right\} \tag{25}
\]

where \( C \) is a constant equal to the vibration amplitude at this specific frequency.

Based on equation 25 we can draw the shapes of the first 8 modes:

Figure 24: First 8 Modes of Cantilever Beam
Step 2 – Calculating the Tip Displacement using the Natural Frequencies of the Beam:

For long-duration loading, the dynamic magnification factor depends principally on the rate of increase of the load to its maximum value. For short-duration load that starts at \( t=0 \) and ends at \( t=T \), the maximum displacement amplitude depends principally upon the magnitude of the applied impulse:

\[
I = \int_{0}^{T} F(t) dt \tag{26}
\]

A convenient procedure for evaluating the maximum response to a short-duration impulsive load may be derived as follows:

The impulse-momentum relationship for the mass \( m \) may be written

\[
m \cdot \Delta \dot{v} = \int_{0}^{T} \left[ F(t) - k \cdot v(t) \right] dt \tag{27}
\]

in which \( \Delta \dot{v} \) represents the change of velocity produced by the loading. In this expression it may be observed that for small values of \( T \) the displacement developed during the loading is of the order of \( T^2 \) while the velocity change \( \Delta \dot{v} \) is of the order of \( T \). Thus, since the impulse is also of the order of \( T \), the elastic force term \( k \cdot v(t) \) vanishes from the expression as \( T \) approaches zero and is negligibly small for short-duration loading.

On this basis, the following relationship may be used:

\[
m \cdot \Delta \dot{v} = \int_{0}^{T} F(t) dt \tag{28}
\]

The response after the termination of the loading is a free vibration:

\[
v(\tilde{t}) = \frac{\dot{v}(T)}{\omega} \sin(\omega \tilde{t}) + v(T) \cdot \cos(\omega \tilde{t}) \tag{29}
\]

in which \( \tilde{t} = t - T \). But since the displacement term \( v(T) \) is negligibly small and the velocity \( \dot{v}(T) = \Delta \dot{v} \), then the following relationship may be used:

\[
v(\tilde{t}) = \frac{1}{m \cdot \omega} \left( \int_{0}^{T} F(t) dt \right) \sin(\omega \tilde{t}) \tag{30}
\]
Equation (30) presents the displacement of the body due to a single frequency. In order to get a more accurate solution, we need to sum the contributions to the displacement made by the first \( n \) natural frequencies.

\[
\nu(t) = \sum_{i=1}^{n} \frac{1}{m \cdot \omega_i} \left( \int_{0}^{T} F(t) dt \right) \sin(\omega_i t)
\]

Both equations (30) and (31) regard the vibrating body as a rigid body with mass \( m \), like the simple one degree of freedom system presented in Figure 25:

![Figure 25: One DOF System](image)

In our case, the cantilever beam does not vibrate as a rigid body and we need to find a correction factor that will represent the mass of the beam as if it were concentrated at the free end of the beam. In order to obtain this correction factor, we will calculate the ratio of the tip displacement to the average displacement in each mode. For these calculations we will use equation (25).
Table 2 presents the calculations of the correction factors:

Table 2: Correction Factors Calculations of the Cantilever Beam Mass

<table>
<thead>
<tr>
<th>Mode</th>
<th>Tip Response</th>
<th>Average Response</th>
<th>$R = \frac{\omega_n^4}{\omega_1^4}$</th>
<th>$\frac{\omega_n^4}{\omega_1^4}$</th>
<th>$R \frac{\omega_n^4}{\omega_1^4}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.724</td>
<td>1.0695</td>
<td>2.54706</td>
<td>12.3623</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>-1.964</td>
<td>0.4121</td>
<td>-4.766</td>
<td>485.523</td>
<td>39.2746</td>
</tr>
<tr>
<td>3</td>
<td>2.002</td>
<td>0.2621</td>
<td>7.63689</td>
<td>3806.63</td>
<td>307.924</td>
</tr>
<tr>
<td>4</td>
<td>-2.001</td>
<td>0.1699</td>
<td>-11.776</td>
<td>14617.1</td>
<td>1182.39</td>
</tr>
<tr>
<td>5</td>
<td>-2</td>
<td>0.1501</td>
<td>13.3179</td>
<td>39944.2</td>
<td>3231.14</td>
</tr>
<tr>
<td>6</td>
<td>-2</td>
<td>0.1044</td>
<td>-19.160</td>
<td>89140.4</td>
<td>7210.69</td>
</tr>
<tr>
<td>7</td>
<td>1.999</td>
<td>0.1072</td>
<td>18.6523</td>
<td>173869</td>
<td>14064.5</td>
</tr>
<tr>
<td>8</td>
<td>-2</td>
<td>0.0738</td>
<td>-27.114</td>
<td>308211</td>
<td>24931.6</td>
</tr>
</tbody>
</table>

The mass correction factor CF can be calculated by summing the first n terms in the $R \frac{\omega_n^4}{\omega_1^4}$ column. The final mass correction factors are listed below.

Table 3: Mass Correction factors

<table>
<thead>
<tr>
<th>n</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>CF_n</td>
<td>2.547064</td>
<td>2.425715</td>
<td>2.450517</td>
<td>2.440557</td>
<td>2.444679</td>
<td>2.442022</td>
<td>2.443348</td>
<td>2.44226</td>
</tr>
</tbody>
</table>

Using the selected correction factor and equation (31), we can obtain the desired solution:

$$v(t) = \frac{1}{CF_n} \sum_{i=1}^{n} \frac{1}{m \cdot \omega_i} \left[ F(t) dt \right] \sin(\omega_i t)$$

(32)
The exact dynamic response for a cantilever beam subjected to an impulsive load can be summarized by the following steps:

- Decide how many modes \( n \) you want to consider in your exact calculation (\( 1 \leq n \leq 8 \)).
- Find the first \( n \) natural frequencies of the beam with the following equation:

\[
\omega_i = \sqrt{\frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}}
\]

where:
- \( \omega_1 = 1.8751^2 \cdot \sqrt{\frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}} \)
- \( \omega_2 = 4.6941^2 \cdot \sqrt{\frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}} \)
- \( \omega_3 = 7.8548^2 \cdot \sqrt{\frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}} \)
- \( \omega_4 = 10.9955^2 \cdot \sqrt{\frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}} \)
- \( \omega_5 = 14.1372^2 \cdot \sqrt{\frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}} \)
- \( \omega_6 = 17.2792^2 \cdot \sqrt{\frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}} \)
- \( \omega_7 = 20.422^2 \cdot \sqrt{\frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}} \)
- \( \omega_8 = 23.562^2 \cdot \sqrt{\frac{EI}{A_{\text{cross-section}} \cdot \rho \cdot L^4}} \)

- Select the appropriate correction factor according to \( n \):

<table>
<thead>
<tr>
<th>( n )</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>( CF_n )</td>
<td>2.547064</td>
<td>2.425715</td>
<td>2.450517</td>
<td>2.440557</td>
<td>2.444679</td>
<td>2.442022</td>
<td>2.443348</td>
<td>2.44226</td>
</tr>
</tbody>
</table>

- Use MathCAD® or other mathematical software to draw the tip displacement of the beam as a function of time:

\[
v(t) = \frac{1}{CF_n} \sum_{i=1}^{n} \frac{1}{m \cdot \omega_i} \left[ \int_0^t F(t)dt \right] \sin(\omega_i t)
\]

- Use the same software to obtain the maximum response of the beam.

A comparison between analytical solutions, including from 1 to 8 modes, and the FEA solution of a 10 meter beam is presented in Figures 26 to 33. In order to compare the analytical solution to a very accurate FEA solution, we used a very fine mesh of 100 Hermitian beam elements and very small time steps of 1 msec.

The eight analytical solutions were obtained using the method described above with the first mode only, the first two modes, the first three modes and so on until the most accurate solution with the first eight modes.

49
Figure 26: FEA vs. Analytical Solution, L=10m (First Mode)

Figure 27: FEA vs. Analytical Solution, L=10m (First 2 Modes)
Figure 28: FEA vs. Analytical Solution, $L=10m$ (First 3 Modes).

Figure 29: FEA vs. Analytical Solution, $L=10m$ (First 4 Modes)
Figure 30: FEA vs. Analytical Solution, L=10m (First 5 Modes)

Figure 31: FEA vs. Analytical Solution, L=10m (First 6 Modes)
FEA vs. Analytical Solution - L=10m.  
(First 7 Modes)

Figure 32: FEA vs. Analytical Solution, L=10m (First 7 Modes)

FEA vs. Analytical Solution - L=10m.  
(First 8 Modes)

Figure 33: FEA vs. Analytical Solution, L=10m (First 8 Modes)
We can definitely observe, in the previous figures, how the analytical solutions approach the FEA solution as we add more modes to the calculation. The improvement in the analytical solution accuracy is quite significant in the first steps but as we add higher and higher modes to the calculation, the changes in the analytical solution can hardly be seen.

We can repeat this exercise for different beam lengths, different forces and different material properties. Just to illustrate the accuracy of the solution for a different beam length, here is a comparison between the FEA solution and the analytical solution for a beam length of 5 meters:

![Graph](image)

**Figure 34: FEA vs. Analytical Solution, L=5m (First 8 Modes)**

For the 10m beam we plotted the displacement over the first 3 seconds and for the 5m beam we plotted the displacement over the first 1.5 seconds. In both cases the time period was chosen to include about \( \frac{1}{4} \) to \( \frac{1}{2} \) of the first mode time period. In order to observe the full vibration of the beam we need to examine larger time periods or simply analyze shorter (stiffer) beams.
5.8 Approximate Solution for the Problem Considered

As presented before, the problem of a cantilever beam subjected to sine-wave impulse is quite complicated and in order to obtain the exact solution or finite element solution which is close enough to the exact one, we need a computer with a finite element code, like ADINA®, or math software like Matlab® or MathCad®.

In addition to the various ways to obtain a good solution, we also need a quick and simple method to obtain an approximate solution. We are looking for some rule of thumb or "back of an envelope calculation" that can give us approximate results without the usage of a computer or specific software.

The general problem we want to solve is the following:

A cantilever beam of length $L$ has a rectangular cross section $a \times b$ and is subjected to half a sine-wave impulse load at the free edge, $T_a$ second long, with a peak value of $F_0$. The material properties of the beam, $E$ and $\rho$, are given. We need to find an approximate solution to evaluate the maximum displacement response for such a beam.

![Cantilever Beam Diagram](image)
Let \( \omega_a \) and \( \omega \) be the applied load frequency and the natural free-vibration frequency of the beam, respectively. The frequency ratio \( (\beta) \) is defined as follows:

\[
\beta \equiv \frac{\omega_a}{\omega}
\]

The structural response will be divided into two phases, as presented in Figure 35, corresponding to the interval during which the load acts, followed by the free-vibration phase.

The undamped response of a structure with stiffness \( k \), including the transient as well as the steady-state term, to harmonic loading is given as:

\[
U(t) = \frac{F_0}{k} \cdot \frac{1}{1 - \beta^2} \cdot (\sin \omega_a t - \beta \cdot \sin \omega t)
\]  
(33)

During phase I, the structure is subjected to harmonic loading starting from rest. Therefore, we can say that during phase I the above term is valid. The free-vibration motion which occurs during phase II depends on the displacement \( U(t) \) and velocity \( \dot{U}(t) \) at \( t = T \) (exiting the end of phase I). This motion may be expressed as follows:

For \( \bar{t} = t - T \geq 0 \):

\[
U(t) = \frac{\dot{U}(t = T)}{\omega} \cdot \sin(\omega \bar{t}) + U(t = T) \cdot \cos(\omega \bar{t})
\]  
(34)

in which the new variable \( \bar{t} = t - T \) has been introduced for convenience.

We are interested in the maximum response produced by the impulsive load rather than the complete history. Therefore, we need to calculate the time when the maximum response occurs. This can be determined by differentiating equation (34) with respect to time and equating to zero:

\[
\frac{dU(t)}{dt} = 0 = \frac{F_0}{k} \cdot \frac{1}{1 - \beta^2} \cdot (\omega_a \cdot \cos \omega t - \omega_a \cdot \cos \omega t)
\]  
(35)

From which

\[
\cos(\omega_a t) = \cos(\omega t)
\]  
(36)

and hence

\[
\omega_a t = 2\pi n \pm \omega t \quad n = 0, \pm 1, 2, 3, \ldots
\]  
(37)
This expression is valid only as long as $\omega_a t \leq \pi$, that is, if the maximum response occurs while the impulsive load is acting.

For $\beta > 1$ ($\omega_a > \omega$) the maximum response occurs during the free-vibration phase (phase II). The initial displacement and velocity for this phase are given by introducing $\omega_a t = \pi$ into equation (33):

$$U(t = T) = \frac{F_0}{k} \cdot \frac{1}{1 - \beta^2} \cdot \left(0 - \beta \cdot \sin \frac{\pi}{\beta}\right)$$  

(38)

$$\dot{U}(t = T) = \frac{F_0}{k} \cdot \frac{\omega_a}{1 - \beta^2} \cdot \left(-1 - \cos \frac{\pi}{\beta}\right)$$  

(39)

The amplitude of this free-vibration motion is given by the following equation:

$$A = \sqrt{\left[U(0)\right]^2 + \left[\dot{U}(0)\right]^2} = \frac{F_0}{k} \cdot \beta \cdot \sqrt{2 + 2 \cos \frac{\pi}{\beta}}$$  

(40)

The Dynamic Magnification Factor (DMF) is the ratio between the dynamic response amplitude and the static displacement which would be produced by the static force $F_0$.

Hence the dynamic magnification factor for this condition is

For $\beta > 1$, $t > T$:  

$$DMF = \frac{U_{\text{max}}}{F_0/k} = \frac{2\beta}{1 - \beta^2} \cdot \cos \left(\frac{\pi}{2\beta}\right)$$  

(41)

From this equation we can derive the desired rule of thumb for our cantilever beam.

First, let us simplify the cantilever beam into a simple one degree of freedom dynamic system:

![Figure 36: Simplified Model for the Cantilever Beam](image)

57
There is some difference between the original beam and the simplified model, mainly because the mass in the model moves as a rigid body while the beam acts differently.

The applied forcing frequency is

\[ \omega_a = \frac{\pi}{T_a} \]  

(42)

The first natural frequency of a cantilever beam (clamped in one edge and free in the other edge) is

\[ \omega = 1.875^2 \cdot \sqrt{\frac{EI}{mL^4}} \]  

(43)

where \( m \) is the weight of the beam per unit of length.

For a rectangular cross section we can write the following equation:

\[ \omega = 1.875^2 \cdot \sqrt{\frac{EI}{ab\rho \cdot L^4}} \]  

(44)

Now we can calculate the frequency ratio:

\[ \beta = \frac{\overline{\omega}}{\omega} \]  

(45)

The dynamic magnification factor can be obtained by equation (41) derived before:

\[ DMF = \frac{2\beta}{1 - \beta^2} \cdot \cos \left( \frac{\pi}{2\beta} \right) \]  

(46)

The static displacement for cantilever beam subjected to a constant tip load \( F_0 \) is given by:

\[ U_{\text{static}} = \frac{F_0 \cdot L^3}{3EI} \]  

(47)

By using the static displacement and the dynamic magnification factor, the desired value can be obtained:

\[ U_{\text{max}} = U_{\text{static}} \cdot DMF = \frac{F_0 \cdot L^3 \cdot 2\beta}{3EI(1 - \beta^2)} \cdot \cos \left( \frac{\pi}{2\beta} \right) \]  

(48)

This approximation considers only the first mode displacement and it does not consider higher modes. This approximation is valid for \( \beta > 0.25 \), for lower values of \( \beta \) we will use the static response as a good approximation for the dynamic maximum response.
Figure 37 summarizes the above process:

Given: $L$, $E$, $a$, $b$, $\rho$, $T$, $F_0$

Calculate $\bar{\omega}$, $\omega$ and $\beta$:

$$\bar{\omega} = \frac{\pi}{T}, \quad \omega = 1.875^2 \cdot \frac{EI}{ab\rho \cdot L^4}, \quad \beta = \frac{\bar{\omega}}{\omega}$$

$\beta < 0.25$?

Yes $\Rightarrow U_{\text{max}} \approx \frac{P_0 \cdot L^3}{3EI}$

No

$$U_{\text{max}} \approx \frac{F_0 \cdot L^3 \cdot 2\beta \cos\left(\frac{\pi}{2\beta}\right)}{3EI(1-\beta^2)}$$

Figure 37: Approximate Solution Scheme for a Cantilever Beam

The same approximate solution, with minor changes, can be used for other beam cross sections like I, T or U cross sections. In addition, we can modify this approximate solution for different beam boundary conditions as presented in Table 4.

Table 4: Modifying the Approximate Solution for different Beam Boundary Conditions

<table>
<thead>
<tr>
<th>Boundary Conditions</th>
<th>Illustration</th>
<th>Maximum Static Deflection</th>
<th>First Mode Frequency</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clamped-Free</td>
<td></td>
<td>$U_{\text{max}} = \frac{F_0 \cdot L^3}{3EI}$</td>
<td>$\omega = 1.875^2 \cdot \sqrt{\frac{EI}{m \cdot L^4}}$</td>
</tr>
<tr>
<td>Clamped-Hinged</td>
<td></td>
<td>$U_{\text{max}} = \frac{7 \cdot F_0 \cdot L^3}{768EI}$</td>
<td>$\omega = 3.927^2 \cdot \sqrt{\frac{EI}{m \cdot L^4}}$</td>
</tr>
<tr>
<td>Clamped-Clamped</td>
<td></td>
<td>$U_{\text{max}} = \frac{F_0 \cdot L^3}{192EI}$</td>
<td>$\omega = 4.730^2 \cdot \sqrt{\frac{EI}{m \cdot L^4}}$</td>
</tr>
<tr>
<td>Hinged-Hinged</td>
<td></td>
<td>$U_{\text{max}} = \frac{F_0 \cdot L^3}{48EI}$</td>
<td>$\omega = 3.1416^2 \cdot \sqrt{\frac{EI}{m \cdot L^4}}$</td>
</tr>
</tbody>
</table>
5.9 Comparing the Approximate Solution with the FEA

In order to verify the accuracy of the approximate solution, we will compare its results with a well established method. The table below compares the solutions obtained by the approximate method with solutions obtained with FEA for over 30 different test cases:

Table 5: Beam Approximate Solution Test Cases

<table>
<thead>
<tr>
<th>Test Case</th>
<th>L [m]</th>
<th>E [Pa]</th>
<th>a, b [m]</th>
<th>ρ [kg/m^3]</th>
<th>F₀</th>
<th>FEA Solution [m]</th>
<th>Approximate Solution [m]</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.1</td>
<td>1.5*10^11</td>
<td>0.01</td>
<td>6000</td>
<td>50</td>
<td>1.373*10^-4</td>
<td>1.333*10^-4</td>
<td>2.96</td>
</tr>
<tr>
<td>2</td>
<td>2*10^-1</td>
<td>2*10^-1</td>
<td>0.01</td>
<td>7000</td>
<td>100</td>
<td>1.25*10^-5</td>
<td>1.25*10^-5</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>2.5*10^-1</td>
<td>0.03</td>
<td>8000</td>
<td>150</td>
<td>2.982*10^-6</td>
<td>2.963*10^-6</td>
<td>0.64</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>0.25</td>
<td>1.5*10^11</td>
<td>0.01</td>
<td>5000</td>
<td>100</td>
<td>4.562*10^-3</td>
<td>4.167*10^-3</td>
<td>9.05</td>
</tr>
<tr>
<td>5</td>
<td>2*10^-1</td>
<td>0.02</td>
<td>7500</td>
<td>200</td>
<td>4.102*10^-4</td>
<td>3.906*10^-4</td>
<td>4.90</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>2.5*10^-1</td>
<td>0.03</td>
<td>9000</td>
<td>300</td>
<td>9.650*10^-5</td>
<td>9.259*10^-5</td>
<td>4.14</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>0.5</td>
<td>1*10^-1</td>
<td>0.01</td>
<td>6000</td>
<td>100</td>
<td>0.0783</td>
<td>0.081</td>
<td>3.39</td>
</tr>
<tr>
<td>8</td>
<td>2*10^-1</td>
<td>0.02</td>
<td>8000</td>
<td>200</td>
<td>3.288*10^-3</td>
<td>3.443*10^-3</td>
<td>4.61</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>3*10^-1</td>
<td>0.03</td>
<td>9000</td>
<td>300</td>
<td>7.23*10^-4</td>
<td>6.173*10^-4</td>
<td>15.77</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>0.75</td>
<td>1.5*10^11</td>
<td>0.01</td>
<td>6000</td>
<td>50</td>
<td>0.0584</td>
<td>0.06</td>
<td>2.70</td>
</tr>
<tr>
<td>11</td>
<td>2*10^-1</td>
<td>0.02</td>
<td>7000</td>
<td>100</td>
<td>8.702*10^-2</td>
<td>8.937*10^-2</td>
<td>2.66</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>2.5*10^-1</td>
<td>0.03</td>
<td>8000</td>
<td>150</td>
<td>2.13*10^-3</td>
<td>1.765*10^-3</td>
<td>18.74</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>1</td>
<td>1.5*10^11</td>
<td>0.01</td>
<td>5000</td>
<td>100</td>
<td>0.184</td>
<td>0.183</td>
<td>0.54</td>
</tr>
<tr>
<td>14</td>
<td>2*10^-1</td>
<td>0.02</td>
<td>7500</td>
<td>200</td>
<td>0.0294</td>
<td>0.03</td>
<td>2.02</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>2.5*10^-1</td>
<td>0.03</td>
<td>9000</td>
<td>300</td>
<td>9.130*10^-3</td>
<td>9.406*10^-3</td>
<td>2.98</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>1.75</td>
<td>1.5*10^11</td>
<td>0.01</td>
<td>6000</td>
<td>50</td>
<td>0.164</td>
<td>0.15</td>
<td>8.92</td>
</tr>
<tr>
<td>17</td>
<td>2*10^-1</td>
<td>0.02</td>
<td>7000</td>
<td>100</td>
<td>0.0316</td>
<td>0.03</td>
<td>5.19</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>2.5*10^-1</td>
<td>0.03</td>
<td>8000</td>
<td>150</td>
<td>0.01097</td>
<td>0.011</td>
<td>0.27</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>1</td>
<td>1.5*10^11</td>
<td>0.01</td>
<td>5000</td>
<td>25</td>
<td>0.156</td>
<td>0.144</td>
<td>8.00</td>
</tr>
<tr>
<td>20</td>
<td>2*10^-1</td>
<td>0.02</td>
<td>7500</td>
<td>50</td>
<td>0.0227</td>
<td>0.021</td>
<td>7.78</td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>2.5*10^-1</td>
<td>0.03</td>
<td>9000</td>
<td>75</td>
<td>7.407*10^-3</td>
<td>6.856*10^-3</td>
<td>7.73</td>
<td></td>
</tr>
<tr>
<td>22</td>
<td>5</td>
<td>1.5*10^11</td>
<td>0.01</td>
<td>6000</td>
<td>50</td>
<td>0.473</td>
<td>0.431</td>
<td>9.29</td>
</tr>
<tr>
<td>23</td>
<td>2*10^-1</td>
<td>0.02</td>
<td>7000</td>
<td>100</td>
<td>0.097</td>
<td>0.086</td>
<td>12.02</td>
<td></td>
</tr>
<tr>
<td>24</td>
<td>2.5*10^-1</td>
<td>0.03</td>
<td>8000</td>
<td>150</td>
<td>0.0357</td>
<td>0.032</td>
<td>10.93</td>
<td></td>
</tr>
<tr>
<td>25</td>
<td>7.5</td>
<td>1.5*10^11</td>
<td>0.01</td>
<td>5000</td>
<td>100</td>
<td>1.559</td>
<td>1.415</td>
<td>9.68</td>
</tr>
<tr>
<td>26</td>
<td>2*10^-1</td>
<td>0.02</td>
<td>7500</td>
<td>200</td>
<td>0.275</td>
<td>0.25</td>
<td>9.52</td>
<td></td>
</tr>
<tr>
<td>27</td>
<td>2.5*10^-1</td>
<td>0.03</td>
<td>9000</td>
<td>300</td>
<td>0.1026</td>
<td>0.091</td>
<td>11.98</td>
<td></td>
</tr>
<tr>
<td>28</td>
<td>10</td>
<td>1*10^-1</td>
<td>0.01</td>
<td>5000</td>
<td>25</td>
<td>0.6374</td>
<td>0.578</td>
<td>9.77</td>
</tr>
<tr>
<td>29</td>
<td>2*10^-1</td>
<td>0.02</td>
<td>7500</td>
<td>50</td>
<td>0.0919</td>
<td>0.083</td>
<td>10.18</td>
<td></td>
</tr>
<tr>
<td>30</td>
<td>3*10^-1</td>
<td>0.03</td>
<td>9000</td>
<td>75</td>
<td>0.0316</td>
<td>0.028</td>
<td>12.08</td>
<td></td>
</tr>
</tbody>
</table>

Average Error 6.95%
The finite elements solution was obtained using Hermitian beam elements with very fine mesh (100 elements along the beam) and very small time steps (1msec.).

Here are the results obtained for different elements and various beam lengths plotted against the exact analytical solution and the approximate solution:

![Beam Length Study - FEA vs. Analytical Solutions](image)

Figure 38: Beam Length Study – FEA vs. Analytical Solutions

As presented above, almost all the finite element results follow the exact solution very closely. For 10<L/H<100 (Beam length < 1m.), the approximate solution predicts the maximum displacement very accurately. For 100<L/H<1000 (Beam length > 1m.), the approximate solution predicts maximum displacements that are slightly lower than the actual displacements.
We can also perform some sensitivity analyses using the approximate method and compare the results with the results presented before:

![Graph 1](image1.png)

Figure 39: FEA vs. Approximate Solution – Young’s Modulus Study

![Graph 2](image2.png)

Figure 40: FEA vs. Approximate Solution – Material Density Study

As presented above, the approximate solution gives quite accurate results with only few simple calculations that can be carried out on the back of an envelope using only a calculator. Such a method can be very useful to quickly check a finite element solution, done by a third party, without redoing the whole finite element analysis.
Chapter 6 – Analysis of a Stiffened Plate

6.1 Geometry, Loading and Boundary Conditions

After analyzing the simple geometry of a cantilever beam, let us move forward to a more complicated structure, the stiffened plate:

![Stiffened Plate Geometry and Principal Dimensions](image)

The stiffened panel has an overall dimension of 90 cm X 90 cm with two equally spaced vertical stiffeners and two equally spaced horizontal stiffeners. The plate thickness $t$, the stiffeners width $w$ and the stiffeners height $h$ will vary to represent multiple geometries.

Table 6: Stiffened Plate Geometry

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Default Value</th>
<th>Minimum Value</th>
<th>Maximum Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plate Thickness ($t$)</td>
<td>4 mm.</td>
<td>3.5 mm.</td>
<td>6 mm.</td>
</tr>
<tr>
<td>Stiffener Width ($w$)</td>
<td>5 mm.</td>
<td>3 mm.</td>
<td>8 mm.</td>
</tr>
<tr>
<td>Stiffener Height ($h$)</td>
<td>5 cm.</td>
<td>3 cm.</td>
<td>8 cm</td>
</tr>
</tbody>
</table>
Two different types of boundary conditions will be employed for the stiffened panel:

- Hinged edges or simply supported plate – Only the translations are fixed at the edges (enabling rotations)
- Clamped edges – All six degrees of freedom are fixed at the edges.

![Figure 42: Stiffened Plate Boundary Conditions](Image)

The material of the panel is considered to be elastic-isotropic with the approximate properties of structural steel. The external load will be a half sine-wave impulsive load of uniform lateral pressure with peak value of 1,000 Pa. lasting from $t = 0$ to $t = 20$ msec.

![Figure 43: Applied Load on the Stiffened Panel](Image)

In all the finite element analyses, we will track the maximum displacement, occurring at the center of the plate, within 20 msec. from the start of the load (This time limit is mainly to keep the usage of computation resources at a reasonable level).
As we will observe later on, there is no need to calculate the solutions for longer periods than 20 msec. since this small structure is very stiff and the maximum response is obtained while the load is still acting on the plate. These geometry, loading and boundary conditions are chosen to roughly simulate a ship’s bottom plate subjected to slamming pressure at high sea state.

An important note should be made about the design criteria of a stiffened panel: In most of the design scenarios of a marine stiffened panel, the maximum deflection in the center of the panel is not the main consideration in the panel design. Other issues like buckling or yielding dominate the geometry of the panel rather than the maximum deflection.

(a) Overall Buckling  (b) Local Torsional Buckling of Stiffeners  (c) Local Plate Buckling

Figure 44: Buckling Modes of a Stiffened Panel, Hughes [12]

Stiffened panels in a ship suffer from various loads and not only lateral pressure. Although there are several indices that reflect the panel ability to withstand these loads, we will ignore most of them. The only external load in our study will be the lateral pressure and we will track only the maximum deflection of the panel as the panel’s ability to resist this load.
6.2 Hierarchic Modeling of a Stiffened Plate

Using the various finite elements for both the plate and the stiffeners, we can model the stiffened plate in dozens of different combinations. As this structure is slightly more complicated than the cantilever beam, we cannot model it in only one dimension, but we still have numerous ways to model the plate and the stiffeners.

Out of all the possible combinations, we will observe the behavior of 4 models, as presented in Table 7 and in Figure 45.

Table 7: Hierarchic Modeling of a Stiffened Plate

<table>
<thead>
<tr>
<th>Model</th>
<th>Plate</th>
<th>Stiffeners</th>
<th>Number of Nodes per Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4-node shell elements</td>
<td>Iso beam 2-node elements</td>
<td>1081</td>
</tr>
<tr>
<td>2</td>
<td>4-node shell elements</td>
<td>4-node shell elements</td>
<td>1329</td>
</tr>
<tr>
<td>3</td>
<td>9-node shell elements</td>
<td>Iso beam 3-node elements</td>
<td>3961</td>
</tr>
<tr>
<td>4</td>
<td>9-node shell elements</td>
<td>9-node shell elements</td>
<td>5181</td>
</tr>
</tbody>
</table>

The 2-node and 3-node iso beam elements suit the 4-node and 9-node shell elements, respectively. This match between these elements is to construct a compatible element assemblage, i.e., to make sure that every node on the stiffener will have a compatible node on the plate.

The number of nodes listed in Table 7 is obtained when the plate is represented by 900 shell elements (30X30) and each of the stiffeners is represented by 90 shell elements (30X3) or 30 iso-beam elements, as seen in Figure 45. These numbers are listed above just to demonstrate the computational resources needed for each model relative to the others.

We are using the same half-sine impulsive load from the previous chapter and we will skip the time discretization study. We will use time steps of $T_a/40$. 
Note that all the models above represent the exact same geometry but the stiffeners in models 1 and 3 look smaller than the stiffeners in models 2 and 4. This is because the iso beam elements are located at a distance of h/2 from the plate where h is the stiffener height. Between the iso beam nodes of the stiffeners and the shell nodes of the plate there are rigid links.
6.3 Plate Thickness Study

Here are the results obtained in the plate thickness sensitivity analysis:

![Plate Thickness Study - Clamped B.C.](image)

Figure 46: Plate Thickness Study – Clamped Boundary Conditions

![Plate Thickness Study - Hinged B.C.](image)

Figure 47: Plate Thickness Study – Hinged Boundary Conditions
6.4 Stiffeners Width Study

Here are the results obtained in the stiffeners width sensitivity analysis:

![Stiffeners Width Study - Clamped B.C.](image1)

**Figure 48: Stiffener Width Study – Clamped Boundary Conditions**

![Stiffeners Width Study - Hinged B.C.](image2)

**Figure 49: Stiffener Width Study – Hinged Boundary Conditions**
6.5 Stiffeners Height Study

Here are the results obtained in the stiffeners height sensitivity analysis:

Figure 50: Stiffener Height Study – Clamped Boundary Conditions

Figure 51: Stiffener Height Study – Hinged Boundary Conditions
### 6.6 Comparing with Static Results and Discussion

The average maximum displacement for clamped boundary conditions in all the three studies is $20.3 \times 10^{-6}$ meter and the average maximum displacement for hinged boundary conditions is almost 3 times larger, $60 \times 10^{-6}$ meter. Although this difference is quite significant, when we divide each result by the displacement obtained in an equivalent static FEA (constant load of $P_0$), the values of the dynamic magnification factors (DMF) are very close to each other.

**Table 8: Plate Thickness Study - Comparison with Static Results**

<table>
<thead>
<tr>
<th>t [mm]</th>
<th>3.5</th>
<th>3.75</th>
<th>4.0</th>
<th>4.25</th>
<th>4.5</th>
<th>4.75</th>
<th>5.0</th>
<th>5.25</th>
<th>5.5</th>
<th>5.75</th>
<th>6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hinged</td>
<td>1.18</td>
<td>1.20</td>
<td>1.20</td>
<td>1.19</td>
<td>1.16</td>
<td>1.14</td>
<td>1.15</td>
<td>1.16</td>
<td>1.16</td>
<td>1.14</td>
<td>1.12</td>
</tr>
<tr>
<td>Clamped</td>
<td>1.15</td>
<td>1.15</td>
<td>1.14</td>
<td>1.14</td>
<td>1.14</td>
<td>1.14</td>
<td>1.13</td>
<td>1.12</td>
<td>1.11</td>
<td>1.11</td>
<td>1.11</td>
</tr>
<tr>
<td>Difference</td>
<td>2.6%</td>
<td>4.4%</td>
<td>5.5%</td>
<td>4.4%</td>
<td>2.0%</td>
<td>0.1%</td>
<td>0.9%</td>
<td>2.3%</td>
<td>3.0%</td>
<td>2.6%</td>
<td>1.3%</td>
</tr>
</tbody>
</table>

**Table 9: Stiffener Width Study - Comparison with Static Results**

<table>
<thead>
<tr>
<th>w [mm]</th>
<th>3.0</th>
<th>3.5</th>
<th>4.0</th>
<th>4.5</th>
<th>5.0</th>
<th>5.5</th>
<th>6.0</th>
<th>6.5</th>
<th>7.0</th>
<th>7.5</th>
<th>8.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hinged</td>
<td>1.10</td>
<td>1.14</td>
<td>1.18</td>
<td>1.20</td>
<td>1.20</td>
<td>1.21</td>
<td>1.21</td>
<td>1.22</td>
<td>1.21</td>
<td>1.21</td>
<td>1.22</td>
</tr>
<tr>
<td>Clamped</td>
<td>1.12</td>
<td>1.12</td>
<td>1.12</td>
<td>1.14</td>
<td>1.14</td>
<td>1.15</td>
<td>1.15</td>
<td>1.16</td>
<td>1.16</td>
<td>1.16</td>
<td>1.16</td>
</tr>
<tr>
<td>Difference</td>
<td>2.2%</td>
<td>2.3%</td>
<td>4.7%</td>
<td>5.3%</td>
<td>5.5%</td>
<td>5.3%</td>
<td>4.7%</td>
<td>5.3%</td>
<td>4.6%</td>
<td>4.2%</td>
<td>4.4%</td>
</tr>
</tbody>
</table>

**Table 10: Stiffener Height Study - Comparison with Static Results**

<table>
<thead>
<tr>
<th>h [mm]</th>
<th>30</th>
<th>35</th>
<th>40</th>
<th>45</th>
<th>50</th>
<th>55</th>
<th>60</th>
<th>65</th>
<th>70</th>
<th>75</th>
<th>80</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hinged</td>
<td>1.48</td>
<td>1.34</td>
<td>1.20</td>
<td>1.20</td>
<td>1.20</td>
<td>1.15</td>
<td>1.14</td>
<td>1.15</td>
<td>1.16</td>
<td>1.16</td>
<td>1.17</td>
</tr>
<tr>
<td>Clamped</td>
<td>1.20</td>
<td>1.19</td>
<td>1.13</td>
<td>1.13</td>
<td>1.14</td>
<td>1.13</td>
<td>1.11</td>
<td>1.09</td>
<td>1.06</td>
<td>1.05</td>
<td>1.05</td>
</tr>
<tr>
<td>Difference</td>
<td>20%</td>
<td>12%</td>
<td>5.5%</td>
<td>0.7%</td>
<td>5.5%</td>
<td>5.2%</td>
<td>2.0%</td>
<td>2.2%</td>
<td>5.9%</td>
<td>8.9%</td>
<td>11%</td>
</tr>
</tbody>
</table>
Another observation that can be drawn from this study is the effect of weight increase on the maximum displacement (i.e., the stiffness of the structure). Like any structure, we want the stiffened panel to be as light as possible and as stiff as required by its usage. The following figure presents the same results as before, but now the horizontal axis is the gross panel weight.

Figure 52: Effect of Weight Increase on Maximum Displacement

Figure 52 demonstrates that increasing the stiffener’s height has the most remarkable effect while increasing the plate’s thickness has the most insignificant effect. It is obvious that adding height to the stiffener is the most efficient way to gain more stiffness to the structure while increasing the plate thickness is the most inefficient way to do so.
6.7 Approximate Solution for the Problem Considered

Introduction:

As presented before, the problem of transient dynamic response of a stiffened panel is quite complicated and the number of independent parameters is large because of the special geometry of the structure. This problem becomes even more complicated when we consider all the boundary conditions available for this geometry. In the following section we will present a solution that would give us an approximate result without the usage of a finite element code or even a computer.

Problem Statement:

A rectangular stiffened panel with overall dimensions of $a \times b$ has a cross stiffening in both the long and the short dimensions. The repeating stiffeners in the two orthogonal directions do not have to be similar. In addition, the central stiffener in each direction can be similar to repeating stiffeners, but it might be also larger to gain some more stiffness in the critical zone.

![Stiffened Plate Geometry for the Approximate Solution](image)

Figure 53: Stiffened Plate Geometry for the Approximate Solution
The stiffened panel has one of the following four boundary conditions:

Case 1 – All four edges simply supported (allow rotations at the edges).
Case 2 – Both short edges fixed and both long edges simply supported.
Case 3 – Both long edges fixed and both long edges simply supported.
Case 4 – All four edges are clamped (All six degrees of freedom are fixed at the edges).

The plate thickness as well as all the geometries of the stiffeners and the material properties are given. A half sine wave impulsive uniform pressure (20msec. long) is applied to the stiffened panel and we need to evaluate the maximum response of the panel (The maximum deflection at the center).

Solution:

In order to obtain the approximate solution, we will multiply the static response with the DMF (dynamic magnification factor). The static response is a function of the panel geometry, material properties and the constant pressure while the DMF is also a function of T, the time duration of the load.

According to O’Leary and Harari [17], exact solutions are not known for rib-reinforced plates, so an exact analytical solution for the stiffened plate problem is out of the question. Instead, we will use a close approximation developed by H.A. Schade [22]. This close approximation was accepted by the US Navy as a method to determine the static response of stiffened panels.

It appears that the analytical solution of cross-stiffened rectangles of plating to withstand uniform bending loads is usually based either on beam theory, or on plate theory. If beam theory is used, the stiffness of the weaker set of stiffeners may be ignored altogether and the stiffer set is represented as taking the entire load or the method of equating deflections may be used. If plate theory is used, the stiffened rectangle is supposed to behave as a solid homogeneous plate as thick as the depth of the stiffeners. Neither method is very accurate except in special cases.
There is another method which includes the beam theory and the plate theory as special cases, and which represents a more accurate approach. The theoretical basis for this method is found elsewhere, it is called the "orthotropic plate" theory and this theory was adopted by Schade [22] to obtain the approximate static solution for the cross-stiffened panels.

According to Schade’s original paper from 1941, the effective breadth should be calculated using the following rule of thumb:

"If the stiffeners are so close together that the effective width of plating is the same as the stiffener spacing; i.e., if the effective width is 100 per cent, then in calculating the moment of inertia I the thickness of the plating should be increased 10 percent over its actual dimension."

The method we will use in our calculations is much more accurate, based on the paper published ten years later by the same author [23].

The general function that correlates between the DMF and the duration of the load will be developed by comparing finite element analysis results for static and dynamic loads for similar panels. The final dynamic results will suffer from errors of two sources:

- Difference between the FEA static response and the approximate response.
- Difference between the DMF found using FEA for this specific panel and the approximate DMF as found from our correlation.

Later on, by comparing the results with FEA, we will be able to quantify each of the errors.
Step 1 – Evaluation of the Effective Breadth of Plating:

The strength of small sections of deck plating between frames can be analyzed by flat plate formulas. Large areas of a hull structure that include stiffening frames or longitudinals, or cross stiffening, require a different approach.

Simple beam theory can be applied to stiffened plating if two conditions are met: adjacent units exert no influence on the edges of the isolated unit. The deflection of the supporting structure is negligible compared with the deflection of the isolated beam.

These assumptions are likely to be valid if the end supports are bulkheads or shell plating, and less likely if they are orthogonal beams. Simple beam theory is not completely applicable to stiffened plating because of the way shear diffuses from the webs into and across the flanges. The direct stress in flanges and plating differs from that predicted by simple beam theory because sections do not remain plane.

\[ \lambda = \text{EFFECTIVE BREADTH OF PLATING} \]

Figure 54: Stress Distribution in Stiffened Plate

The wavy line in Figure 54 shows the distribution of normal stress across stiffened plating under bending load. This effect is known as shear lag. The maximum stress is found by assuming that a certain part of the plating is wholly effective and applying simple beam theory to the effective part.

The effective breadth of plating (\( \lambda \)) is used to calculate the effective moment of inertia of the cross section. Table 11 gives the breadths for various conditions.
Table 11: Effective Breadth of Stiffened Panel

<table>
<thead>
<tr>
<th>L/S</th>
<th>0.5</th>
<th>1.0</th>
<th>2.0</th>
<th>4.0</th>
<th>6.0</th>
<th>8.0</th>
<th>10.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\lambda$/S for Uniform Load</td>
<td>0.196</td>
<td>0.369</td>
<td>0.737</td>
<td>0.989</td>
<td>1.045</td>
<td>1.069</td>
<td>1.080</td>
</tr>
</tbody>
</table>

These points can be connected by the following 4th order polynomial term, as presented in Figure 55:

$$\frac{\lambda}{S} = -0.0004 \cdot \left( \frac{L}{S} \right)^4 + 0.0112 \cdot \left( \frac{L}{S} \right)^3 - 0.1247 \cdot \left( \frac{L}{S} \right)^2 + 0.6158 \cdot \left( \frac{L}{S} \right) - 0.0995$$

(49)

Figure 55: Effective Breadth of Stiffened Panels

where L is the span of the supported panel for hinged boundary conditions or the distance between points of zero bending moment for clamped boundary conditions (0.58 X span for uniformly distributed load).
Step 2 – Calculate the Unit Stiffness in Both Directions:

Calculate $I_{na}$, $I_{nb}$, $I_a$ and $I_b$ using the following definitions:

$I_{na} =$ Moment of inertia of repeating long stiffeners, including effective breadth of plating.

$I_{nb} =$ Moment of inertia of repeating short stiffeners, including effective breadth of plating.

$I_a =$ Moment of inertia of central long stiffeners, including effective breadth of plating.

$I_b =$ Moment of inertia of central short stiffeners, including effective breadth of plating.

The parameters $i_a$ and $i_b$, the unit stiffness in the long and short directions, are the moment of inertia of the stiffeners per unit width and these parameters are given in the following equations:

$$i_a = \frac{I_{na}}{S_a} + 2 \cdot \left( \frac{I_a - I_{na}}{b} \right) \quad i_b = \frac{I_{nb}}{S_b} + 2 \cdot \left( \frac{I_b - I_{nb}}{a} \right)$$

Note that if the central stiffener is equal to the repeating stiffeners, then $I_a = I_{na}$ and these equations reduce to:

$$i_a = \frac{I_{na}}{S_a} \quad i_b = \frac{I_{nb}}{S_b}$$

For an unstiffened plate the following equations are valid:

$$i_a = i_b = \frac{t^3}{12 \cdot (1 - v^2)}$$

For a plate with single stiffening (stiffeners in short direction only), the following equations are valid:

$$i_a = 0 \quad i_b = \frac{I_{nb}}{S_b}$$
Step 3 – Calculation of the Virtual Side Ratio and the Torsion Coefficient:

The virtual side ratio ($\rho$) is the actual side ratio, $a/b$, modified by the ratio of the unit stiffness in the two directions. The virtual side ratio is always equal or greater than 1 and it is given in the following definition:

$$\rho = \frac{a}{b} \sqrt{\frac{i_b}{i_a}}$$  \hspace{1cm} (54)

The torsion coefficient ($\eta$) accounts for horizontal shear stress in the plating, and is defined roughly as the ratio of the inertia of the material subject to horizontal shear stress to the inertia of the material subject to bending:

$$\eta = \frac{a}{b} \sqrt{\frac{I_{pa}}{I_{na}} \cdot \frac{I_{pb}}{I_{nb}}}$$  \hspace{1cm} (55)

In a grid without plating, no material is subjected to horizontal shear and $\eta = 0$. In an unstiffened plate, all the material is subjected to both horizontal shear and bending and $\eta = 1$. In stiffened plate structures, only the plating is subjected to horizontal shear, but both plating and stiffeners are subjected to bending, so $0 < \eta < 1$.

Table 12: Types of Stiffening with Applicable Formulas for Parameters

<table>
<thead>
<tr>
<th>Cross-Stiffening</th>
<th>Single-Stiffening</th>
<th>Unstiffened</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Cross-Stiffening Diagram" /></td>
<td><img src="image" alt="Single-Stiffening Diagram" /></td>
<td><img src="image" alt="Unstiffened Diagram" /></td>
</tr>
<tr>
<td>$i_a = \frac{I_{na}}{S_a} + 2 \cdot \left( \frac{I_a - I_{na}}{b} \right)$</td>
<td>$i_a = 0$</td>
<td>$i_a = i_b = \frac{t^3}{12 \cdot (1 - \nu^2)}$</td>
</tr>
<tr>
<td>$i_b = \frac{I_{nb}}{S_b} + 2 \cdot \left( \frac{I_b - I_{nb}}{a} \right)$</td>
<td>$i_b = \frac{I_{nb}}{S_b}$</td>
<td>$\rho = \frac{a}{b}$</td>
</tr>
<tr>
<td>$\rho = \frac{a}{b} \sqrt{\frac{i_b}{i_a}}$</td>
<td>$\rho = \infty$</td>
<td>$\eta = 1.0$</td>
</tr>
<tr>
<td>$\eta = \frac{a}{b} \sqrt{\frac{I_{pa}}{I_{na}} \cdot \frac{I_{pb}}{I_{nb}}}$</td>
<td>No need to calculate $\eta$ because all the values of $K_7$ are similar at $\rho = \infty.$</td>
<td></td>
</tr>
</tbody>
</table>

79
Step 4 – Evaluation of the Dimensionless Coefficient K and the Maximum Static Deflection at the Center:

The dimensionless coefficient K is used to calculate the maximum static deflection using the following equations:

\[
W_{\text{max} \,(\text{static})} = K \cdot \frac{P \cdot b^4}{E \cdot i_b}
\]  

(56)

where \( P \) is the uniform pressure acting on the stiffened panel.

For unstiffened plates, the following formula applies:

\[
W_{\text{max} \,(\text{static})} = 10.91 \cdot K \cdot \frac{P \cdot b^4}{E \cdot t^2}
\]  

(57)

Given the panel boundary conditions, \( \eta \) and \( \rho \), we can find the coefficient K by using Figure 54 or Tables 12-15, which represent the same data in different forms.

![Diagram showing deflection at the center of a stiffened plate](image)

Figure 56: Deflection at the Center of a Stiffened Plate, Schade [22]
Table 13: K Values for Case 1 – All Four Edges Simply Supported

<table>
<thead>
<tr>
<th>ρ</th>
<th>K Values</th>
<th></th>
<th>K Values</th>
<th></th>
<th>K Values</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>η = 0.0</td>
<td>η = 0.5</td>
<td>η = 1.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.00</td>
<td>0.0082</td>
<td>0.0054</td>
<td>0.0041</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.20</td>
<td>0.0111</td>
<td>0.0075</td>
<td>0.0056</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.40</td>
<td>0.0129</td>
<td>0.0092</td>
<td>0.0071</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.60</td>
<td>0.0140</td>
<td>0.0104</td>
<td>0.0083</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.80</td>
<td>0.0146</td>
<td>0.0114</td>
<td>0.0093</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.00</td>
<td>0.0148</td>
<td>0.0120</td>
<td>0.0101</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.00</td>
<td>0.0139</td>
<td>0.0132</td>
<td>0.0122</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.00</td>
<td>0.0131</td>
<td>0.0131</td>
<td>0.0128</td>
<td></td>
<td></td>
</tr>
<tr>
<td>∞</td>
<td>0.0130</td>
<td>0.0130</td>
<td>0.0130</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 14: K Values for Case 2 – Long Edges Supported, Short Edges Fixed

<table>
<thead>
<tr>
<th>ρ</th>
<th>K Values</th>
<th></th>
<th>K Values</th>
<th></th>
<th>K Values</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>η = 0.0</td>
<td>η = 0.5</td>
<td>η = 1.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.00</td>
<td>0.0027</td>
<td>0.0023</td>
<td>0.0019</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.25</td>
<td>0.0054</td>
<td>0.0043</td>
<td>0.0035</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.50</td>
<td>0.0083</td>
<td>0.0065</td>
<td>0.0053</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.75</td>
<td>0.0107</td>
<td>0.0085</td>
<td>0.0070</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.00</td>
<td>0.0124</td>
<td>0.0100</td>
<td>0.0084</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.50</td>
<td>0.0140</td>
<td>0.0120</td>
<td>0.0105</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.00</td>
<td>0.0141</td>
<td>0.0128</td>
<td>0.0117</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.50</td>
<td>0.0138</td>
<td>0.0131</td>
<td>0.0123</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.00</td>
<td>0.0134</td>
<td>0.0131</td>
<td>0.0127</td>
<td></td>
<td></td>
</tr>
<tr>
<td>∞</td>
<td>0.0130</td>
<td>0.0130</td>
<td>0.0130</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 15: K Values for Case 3 – Long edges fixed, Short Edges Supported

<table>
<thead>
<tr>
<th>ρ</th>
<th>K Values</th>
<th></th>
<th>K Values</th>
<th></th>
<th>K Values</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>η = 0.0</td>
<td>η = 0.5</td>
<td>η = 1.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.00</td>
<td>0.0027</td>
<td>0.0023</td>
<td>0.0019</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.33</td>
<td>0.0030</td>
<td>0.0026</td>
<td>0.0024</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.00</td>
<td>0.0028</td>
<td>0.0027</td>
<td>0.0026</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.66</td>
<td>0.0026</td>
<td>0.0026</td>
<td>0.0026</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.00</td>
<td>0.0026</td>
<td>0.0026</td>
<td>0.0026</td>
<td></td>
<td></td>
</tr>
<tr>
<td>∞</td>
<td>0.0026</td>
<td>0.0026</td>
<td>0.0026</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Table 16: K Values for Case 4 – All Edges Fixed

<table>
<thead>
<tr>
<th>$\rho$</th>
<th>K Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.00</td>
<td>0.0013</td>
</tr>
<tr>
<td>1.10</td>
<td>0.0015</td>
</tr>
<tr>
<td>1.20</td>
<td>0.0017</td>
</tr>
<tr>
<td>1.30</td>
<td>0.0019</td>
</tr>
<tr>
<td>1.40</td>
<td>0.0021</td>
</tr>
<tr>
<td>1.50</td>
<td>0.0022</td>
</tr>
<tr>
<td>1.60</td>
<td>0.0023</td>
</tr>
<tr>
<td>1.70</td>
<td>0.0024</td>
</tr>
<tr>
<td>1.80</td>
<td>0.0024</td>
</tr>
<tr>
<td>1.90</td>
<td>0.0025</td>
</tr>
<tr>
<td>2.00</td>
<td>0.0025</td>
</tr>
<tr>
<td>$\infty$</td>
<td>0.0026</td>
</tr>
</tbody>
</table>

Step 5 – Evaluation the Maximum Dynamic Deflection at the Center:

The DMF is a function of all the parameters in the panel geometry, the boundary conditions, the material properties and the load function. Finding the exact value of the DMF analytically as a function of all these parameters is impossible and we need a faster and easier way to estimate the DMF for our approximate solution.

As can be seen in equation (56), the maximum response of the plate is strongly dependent upon $b$, the short edge of the panel. Previous studies also showed that for the same $b$ and the same boundary conditions but with different stiffeners and different plate thickness, the DMF was almost unchanged (most of the results within 20% range). These observations might point out that the dynamic magnification factor can be roughly estimated just by the length $b$ and the boundary conditions.
Using ADINA® finite element code, we modeled and examined 28 different rectangular stiffened panels (4m < b < 10m) with various plate thicknesses, various stiffeners dimensions and two different boundary conditions (fixed and hinged). Each panel was analyzed twice for both static and dynamic response (total of 56 analyses). As the panel got larger, we added more stiffeners in order to keep the distance between stiffeners almost constant (0.8m-1m.) The results from these analyses are presented in Figure 57:

![Dynamic Magnification Ratio](image)

**Figure 57:** Dynamic Magnification Ratio as a Function of the Panel’s Length

Based on these DMF values, a 3rd order polynomial term was fitted to each of the boundary conditions:

\[
DMF_{(fixed)} = -0.0015 \cdot b^3 + 0.0532 \cdot b^2 - 0.675 \cdot b + 3.2436 \\
DMF_{(hinged)} = 0.0003 \cdot b^3 + 0.0077 \cdot b^2 - 0.2252 \cdot b + 1.3561
\] (58) (59)

The dynamic response for the panel is given in the following equation:

\[
W_{\text{max(dynamic)}} = DMF \cdot W_{\text{max( static)}}
\] (60)
Figure 58 summarizes the approximate solution for the stiffened panel:

Given: Panel dimensions, Plate thickness, Stiffener geometry, number of stiffeners, material properties, boundary conditions and peak load.

Calculate the effective breadth of the plate according to table 11 or the following equation (for clamped B.C $L = 0.58 \times $Length):

$$\frac{\lambda}{S} = -0.0004 \left( \frac{L}{B} \right)^4 + 0.0112 \left( \frac{L}{B} \right)^3 - 0.1247 \left( \frac{L}{B} \right)^2 + 0.6158 \left( \frac{L}{B} \right) - 0.0995$$

Calculate the moment of inertia of the cross section including effective breath effect relative to the cross section neutral axis.

Calculate the unit stiffness, virtual side ratio and torsion coefficient

Find the value of the coefficient $K$ from figure 54 or tables 13-16

$$W_{\text{max (static)}} = K \cdot \frac{P \cdot b^4}{E \cdot i_b}$$

Clamped edges

$DMF_{(fixed)} = -0.0015 \cdot b^3 + 0.0532 \cdot b^2 - 0.675 \cdot b + 3.2436$

Simply supported edges

$DMF_{(hinged)} = 0.0003 \cdot b^3 + 0.0077 \cdot b^2 - 0.2252 \cdot b + 1.3561$

$$W_{\text{max (dynamic)}} = DMF \cdot W_{\text{max (static)}}$$

Figure 58: Approximate solution for the stiffened panel dynamic response

A detailed example for this process, including comparison to FEA results, is given in appendix I.
6.8 Validating the Approximate Solution

The procedure described in Figure 58 is a combination of two methods:

The static response is estimated by using the effective breadth curves and the design curves published by Schade [22+23]. The DMF is estimated by using two equations, one for hinged boundary condition and one clamped boundary conditions, that where formulated after running 56 finite element analyses.

Although we used a finite element code and a computer to obtain these equations, from this point and on we will used these equation "as is", The purpose of this section is to validate the procedure described in Figure 58 and to compare the results obtained in this method with the results obtain using ADINA® finite element code.

For this purpose we will model and test 20 different test cases for both static and dynamic response of a stiffened panels. The comparison with the FEA will enable us to evaluate the error associated with the static response evaluation, the error associated with the DMF evaluation and the error associated with the dynamic response evaluation.

Table 17 presents the results of this process.
As presented above, the average error in estimating the maximum dynamic response in those test-cases is 21.7%. This error is a combination of estimating both the maximum static response and the DMF in approximate methods (14.1% error and 15.9% error for square plate, respectively).
Chapter 7 – Summary and Concluding Remarks

In the analysis of cantilever beams, we noticed that some of the elements used for modeling the physical problem are clearly inappropriate for this application. These elements, the 2D 4-node plane stress and the 3D 8-nodes, unless incompatible modes are used, are too stiff and they lock even with very fine meshes. Among the elements that yield good results, the best element for this problem was the Hermitian beam element which proved to be both accurate and very economic.

In the analysis of the stiffened panels, we noticed that all four modeling techniques resulted in the same results, regardless of geometry, boundary conditions and type of load (static/dynamic). All four models are appropriate for this purpose but the most economic, and therefore the preferable one, is modeling the plate with 4-node shell elements and modeling the stiffeners with 2-node iso-beam elements. For the same mesh density, this model has 80% less nodes than the 9-node shell-shell model and still produces the same results.

These observations are critical for conducting successful dynamic analyses, when we analyze the problem using many time steps and tend to consume a lot of computational resources.

This research concentrated on the linear dynamic response of the stiffened panel. Further research is suggested to examine and compare the various finite elements models of a stiffened panel subjected to non-linear behavior, buckling and post-buckling.

The analytical solutions, both the exact one and the approximate ones, can be employed to check and validate the results obtained in the numerical FEA. Furthermore, these solutions are very helpful in understanding the physical characteristics of the two problems and the various parameters that influence and govern the dynamic response of these structures.
References


Appendix I – Detailed Example for Stiffened Panel

Approximate Calculation

To illustrate the usage of the approximate solution for a stiffened panel, a numerical example is given. In this example the various properties of the two orthogonal directions are similar to demonstrate a clear and short solution without calculating all the parameters for each axis separately.

Problem Statement:
A rectangular stiffened panel has the following properties:
- $a$ (long edge) = $b$ (short edge) = 4m.
- $t$ (plate thickness) = 6mm.
- $N$ (number of stiffeners) = 3 in each direction (cross-stiffened panel), equally spaced.
- $h_a = h_b$ (stiffeners height) = 80mm.
- $w_a = w_b$ (stiffeners width) = 8mm.
- Boundary conditions = All edges are simply supported.
- $P$ (uniform pressure) = 1,000 Pa.
- Material properties: $E = 207$ GPa, $v = 0.3$

Solution:
Calculating the distance between the stiffeners:

$$S_a = S_b = \frac{a}{N_a + 1} = \frac{b}{N_b + 1} = \frac{4}{3 + 1} = 1 m.$$ 

Calculating the ratio between the edge of the plate to the distance between stiffeners parallel to this edge:

$$\frac{b}{S_a} = \frac{a}{S_b} = \frac{4}{1} = 4$$

Looking for the values of $\lambda/S$ in Table 11:

$$\frac{\lambda_a}{S_a} = \frac{\lambda_b}{S_b} = 0.989$$
Calculating the effective breadth in each direction:

$$\lambda_a = \lambda_b = 0.989 \cdot S_a = 0.989 \cdot S_b = 0.989 \cdot 1 = 0.989m.$$

Calculating the areas and the center of areas for the plates and the stiffeners:

$$A_{plate(a)} = A_{plate(b)} = t \cdot \lambda = 0.006 \cdot 0.989 = 5.934 \cdot 10^{-3} m^2$$

Center of area located for both plates located at

$$\frac{t}{2} = \frac{0.006}{2} = 0.003m$$

$$A_{stiffener(a)} = A_{stiffener(b)} = w \cdot h = 0.008 \cdot 0.08 = 6.4 \cdot 10^{-4} m^2$$

Center of area located at

$$t + \frac{h}{2} = 0.006 + \frac{0.08}{2} = 0.046m$$

Calculating the location of the centroid of the cross section:

$$Y_a = Y_b = \frac{A_{stiffener(a)} \left[ \left( t + \frac{h_a}{2} \right) + A_{plate(a)} \frac{t}{2} \right] + A_{plate(a)} \frac{Y_a - t}{2} + w \cdot h^3 12 + A_{stiffener(a)} \left[ \left( t + \frac{h_a}{2} \right) - Y_a \right]^2}{A_{stiffener(a)} + A_{plate(a)}} = 7.186 \cdot 10^{-3} m$$

Calculating the moment of inertia of the plate at the neutral axis (using the parallel axes theorem):

$$I_a = I_b = I_{plate(a)} + I_{stiffener(a)} = \frac{\lambda_a \cdot t^3}{12} + A_{plate(a)} \left[ Y_a - \frac{t}{2} \right]^2 + \frac{w \cdot h^3}{12} + A_{stiffener(a)} \left[ \left( t + \frac{h_a}{2} \right) - Y_a \right]^2$$

$$I_a = I_b = 1.427 \cdot 10^{-6} m^4$$

Calculating the unit stiffness:

$$i_a = i_b = \frac{I_a}{S_a} = 1.427 \cdot 10^{-6} m^3$$

Calculating the virtual side ratio (must be 1 for panel with two orthogonal axes of symmetry):

$$\rho = \frac{a}{b} \sqrt[4]{ \frac{i_b}{i_a} } = 1$$

Calculating the torsion coefficient:

$$\eta = \frac{a}{b} \sqrt[4]{ \frac{I_{pa}}{I_{na}} \frac{I_{pb}}{I_{nb}} } = \frac{4}{4} \sqrt{ \frac{1.44 \cdot 10^{-7}}{1.388 \cdot 10^{-6}} \frac{1.44 \cdot 10^{-7}}{1.388 \cdot 10^{-6}} } = 0.011$$

Obtaining the value of the coefficient $K$ from Figure 56 or from Table 13:

$$K = 0.0082$$
Plug-in all the values in the final formula:

\[ W_{\text{max}\,(\text{static})} = K \cdot \frac{P \cdot b^4}{E \cdot I_y} = 0.0082 \cdot \frac{1000 \cdot 4^4}{2.07 \cdot 10^{11} \cdot 1.427 \cdot 10^{-6}} \]

\[ W_{\text{max}\,(\text{static})} = 0.007105m. \]

The same problem was solved with ADINA® using shell-shell model (9-nodes elements):

![Figure 59: FEA Displacement Solution for the Detailed Example](image)

As presented above, the maximum displacement obtained with the FEA is 0.007366m., less than 5% away from the static analytical solution.

Estimating the DMF for L=4m and hinged boundary conditions:

\[ DMF_{\text{hinged}} = 0.0003 \cdot 4^3 + 0.0077 \cdot 4^2 - 0.2252 \cdot 4 + 1.3561 = 0.598 \]

Estimating the dynamic response:

\[ W_{\text{max}\,(\text{dynamic})} = DMF \cdot W_{\text{max}\,(\text{static})} = 0.598 \cdot 0.007105 = 0.004249m. \]

The dynamic response according to FEA is 0.005724m. The approximate dynamic solution is 25% away from the finite elements solution (average accuracy according to our study).

The dynamic magnification factor according to the FEA is 0.777.