Experimental and finite element studies of shock transmission through bolted joints

Masoud Feghhi
University of Nevada, Las Vegas

Follow this and additional works at: https://digitalscholarship.unlv.edu/rtds

Repository Citation
Feghhi, Masoud, "Experimental and finite element studies of shock transmission through bolted joints" (2007). UNLV Retrospective Theses & Dissertations. 2765.
https://digitalscholarship.unlv.edu/rtds/2765

This Dissertation is brought to you for free and open access by Digital Scholarship@UNLV. It has been accepted for inclusion in UNLV Retrospective Theses & Dissertations by an authorized administrator of Digital Scholarship@UNLV. For more information, please contact digitalscholarship@unlv.edu.
EXPERIMENTAL AND FINITE ELEMENT STUDIES OF SHOCK TRANSMISSION THROUGH BOLTED JOINTS

by

Masoud Feghhi

A dissertation submitted in partial fulfillment of the requirements for the

Doctor of Philosophy Degree in Mechanical Engineering
Department of Mechanical Engineering
Howard R. Hughes College of Engineering

Graduate College
University of Nevada, Las Vegas
December 2007
INFORMATION TO USERS

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleed-through, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

UMI Microform 3302352
Copyright 2008 by ProQuest LLC.
All rights reserved. This microform edition is protected against unauthorized copying under Title 17, United States Code.

ProQuest LLC
789 E. Eisenhower Parkway
PO Box 1346
Ann Arbor, MI 48106-1346

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Dissertation Approval
The Graduate College
University of Nevada, Las Vegas

18 October 2007

The Dissertation prepared by
Masoud Pehghi

Entitled
Experimental and Finite Element Studies of Shock Transmission Through
Bolted Joints

is approved in partial fulfillment of the requirements for the degree of
Ph.D. in Mechanical Engineering

Brendan Groz
Examination Committee Chair

Dean of the Graduate College

Examination Committee Member

Examination Committee Member

Graduate College Faculty Representative
ABSTRACT

Experimental and Finite Element Studies of Shock Transmission Through Bolted Joints

by

Masoud Feghhi

Dr. Brendan J. O'Toole, Examination Committee Chair
Associate Professor of Mechanical Engineering
University of Nevada, Las Vegas

The aim of this study is to analyze and assess the dynamic behavior of bolted joint connections subjected to impact loads using Finite Element Analysis (FEA) and experiment. Also, it investigates the effect of the joint on shock propagation through the structure. There is little or no literature available describing the proper method for analyzing the transient shock propagation across bolted connections. The main study will be performed on hat sections bolted to a flat plate. These simple configurations are representative of structures found in many military ground vehicles that can be subjected to transient impact and blast loads. The best way to approach this problem is first to compare and verify the experiment and modeling results on the plate and hat section individually. The next step is to verify the result of a bolted structure. The last step would be a parametric study of the bolted joints with different variables, such as contact type and area, friction, preload on bolt, Vibration characteristics of bolt and spacers and FEA results output frequency.
An impulse hammer with built in load cell along with accelerometers have been used to obtain the response of the shock for the experimental work. Finite element Method (FEM) is used for analysis. The model has been made and meshed in HyperMesh®, and then exported to LS-DYNA to solve and obtain the results from the shock applied to the structure.

The results will be presented in three categories. First the modal analysis is performed both numerically and experimentally. The results were in excellent agreement with less than 2% error. Secondly, the time history response of FEA and experimental results are compared together. Different methods such as Root Mean Square (RMS), moment method and maximum peak acceleration method was used to obtain the resemblance of experimental and Finite Element responses. The results show that solid elements with a fine mesh must be used in the modeling the structure to obtain a reliable response from FEA. Finally, the Shock Response Spectrum (SRS) is used to calculate the critical frequency for design purposes. As long as the structure is modeled with the solid elements and mesh is refined properly the FEA and experiment detects the same critical frequency.

The study of shock propagation through structure with bolted joints showed that joint is reducing the maximum acceleration amplitude by a factor of 3. Furthermore, using a washer and bolt with a lower stiffness material can attenuate shock significantly. In some cases there is up to 40% reduction in peak acceleration.
TABLE OF CONTENTS

ABSTRACT .................................................................................................................................. iii

LIST OF FIGURES .................................................................................................................... vii

LIST OF TABLES ......................................................................................................................... x

NOMENCLATURE .................................................................................................................... xi

ACKNOWLEDGEMENTS ...................................................................................................... xiii

CHAPTER 1  INTRODUCTION ............................................................................................. I
   1.1 Project Overview ............................................................................................................. 1
   1.2 Application ...................................................................................................................... 2
   1.3 Problem Configuration .................................................................................................... 3
   1.4 Review of Literature ........................................................................................................ 6
   1.5 Dissertation Objectives ................................................................................................. 10

CHAPTER 2 COMPARISON OF TWO TRANSIENT RESPONSES........................... 12
   2.1 The Need for Establishing Error Criteria ...................................................................... 12
   2.2 Applications ................................................................................................................... 14
   2.3 Error Criteria Objectives ............................................................................................... 16
   2.4 Review of the Literature in Transient Response Comparison ....................................... 17
   2.5 Error Calculation Methods ............................................................................................ 19
   2.6 Summary of Error Calculation Methods ....................................................................... 32
   2.7 The Dissimilarity Factor (DF) ....................................................................................... 34

CHAPTER 3 EXPERIMENTAL CALIBRATION OF FEA ............................................ 35
   3.1 Introduction ................................................................................................................... 35
   3.2 Geometry of the Bar ...................................................................................................... 35
   3.3 Experimental Procedure ............................................................................................... 36
   3.4 Natural Modes of Vibration .......................................................................................... 47
   3.5 Finite Element Analysis of a Simple Structure ............................................................. 53
   3.6 Comparison of Results ................................................................................................. 58

CHAPTER 4 STRUCTURES WITHOUT JOINTS ........................................................... 62
   4.1 Introduction .................................................................................................................. 62
   4.2 Quarter Inch Steel Plate ............................................................................................... 62
   4.3 Hat Section .................................................................................................................... 83
   4.4 Reflection of the Shock Wave ..................................................................................... 102
LIST OF FIGURES

Figure 1.1. Dingo armored vehicle ............................................................................................4
Figure 1.2. The simplified model of the armored vehicle..........................................................5
Figure 2.1. A typical result of a shock response of a structure from the experiment .......... 12
Figure 2.2. Shell element model and experimental results .......................................................13
Figure 2.3. Solid element model and experimental results .......................................................14
Figure 2.4. Seven days forecast and recorded temperature from Fox News [25] .............. 15
Figure 2.5. Seven days forecast and recorded temperature from Weather Channel [26] ......16
Figure 2.6. Illustration of the problem with two sets of analysis and experimental curves... 17
Figure 2.7. Accelerations obtained from experimental results and shell FE model ...............20
Figure 2.8. Error calculated with the regular method ..............................................................21
Figure 2.9. Two sets of curves with the same amplitude and same phase shift ................. 23
Figure 2.10. Example of curves with positive and negative skewness .................................26
Figure 2.11. Example of curves with kurtosis greater and less than 3 ................................. 27
Figure 2.12. Illustration of maximum peak error calculation method ................................. 30
Figure 2.13. Comparison of two accelerations with peak counting method ...................... 32
Figure 3.1. Dimensions of the Bar ...........................................................................................36
Figure 3.2. PCB 086C02 impulse hammer (Small) .................................................................37
Figure 3.3. PCB 086C20 impulse hammer (Large) .................................................................37
Figure 3.4. Comparison of Large and small impulse hammers ............................................. 38
Figure 3.5. PCB 352C22 ceramic shear ICP accelerometer ....................................................38
Figure 3.6. PCB 394C06 hand held calibrator .........................................................................40
Figure 3.7. A six FFT input channel Pulse data acquisition hardware .................................41
Figure 3.8. A-frame and pendulum setup ................................................................................43
Figure 3.9. Experimental FFT plot of the round bar ...............................................................44
Figure 3.10. Applied force measured by small impulse hammer to the round bar ...............45
Figure 3.11. Experimentally measured acceleration of the round bar in the time domain .... 46
Figure 3.12. Coherence plot ....................................................................................................46
Figure 3.13. Force curve applied to the finite element models ............................................. 55
Figure 3.14. Impact point and accelerometers locations on hat section .............................. 56
Figure 3.15. Solid element model of the round bar ............................................................... 57
Figure 3.16. Time history response of the bar determined by finite element analysis .......... 59
Figure 3.17. Time history response of the bar (Experiment vs. FEA) ................................. 60
Figure 3.18. SRS of experimental and FE analysis of the bar ............................................... 61
Figure 4.1. Dimensions of the flat plate in m ...........................................................................63
Figure 4.2. Experimental setup of the flat plate .....................................................................65
Figure 4.3. Impact point and accelerometer location on the plate ........................................65
Figure 4.4. Applied force to the flat plate measured by the instrumented hammer ............ 66
Figure 4.5. Experimental result of the flat plate ................................................................. 67
Figure 4.6. Study of mesh refinement on plate finite analysis .............................................. 68
Figure 4.7. Shell element model of the plate ......................................................................... 69
Figure 4.8. Visualization of mesh refinement on shell element models of the plate .......... 70
Figure 4.9. Solid element model of the plate ..........................................................................71
Figure 4.10. Mode shapes of the plate obtained by the finite elements modal analysis ....... 74
Figure 4.11.  Time history response of the plate: Experimental and FE analysis  
Figure 4.12.  Shock response spectrum (SRS) of the plate: (shell element model)  
Figure 4.13.  Time history response of the plate: Experimental and FE analysis  
Figure 4.14.  Difference in moment values and dissimilarity factors for plate models  
Figure 4.15.  Time history response of the plate: Experimental and FE analysis  
Figure 4.16.  Shock response spectrum (SRS) of the plate: (solid element model)  
Figure 4.17.  Hat section configuration and dimensions are in m  
Figure 4.18.  Experimental setup for hat section  
Figure 4.19.  Hat section, accelerometers and large impact hammer  
Figure 4.20.  Impact point and accelerometer location on the hat section  
Figure 4.21.  Applied force to the hat section by large instrumented hammer  
Figure 4.22.  Experimental result of the hat section  
Figure 4.23.  Study of mesh refinement on finite element modeling  
Figure 4.24.  Shell element model of a hat section configuration  
Figure 4.25.  Close up of part of shell element model of steel hat section  
Figure 4.26.  Solid element model of a hat section configuration  
Figure 4.27.  Mode shapes of the hat section obtained by the FE modal analysis  
Figure 4.28.  Time history response of the hat section: Experimental and FEA  
Figure 4.29.  Shock response spectrum (SRS) of the hat section: (shell element model)  
Figure 4.30.  Time history response of the hat section: Experimental and FEA  
Figure 4.31.  Difference in moment values and dissimilarity factors for hat section models  
Figure 4.32.  Time history response of the hat section: Experimental and FEA  
Figure 4.33.  Shock response spectrum (SRS) of the hat section: (solid element model)  
Figure 5.1  Assembly drawing of the bolted joint structure  
Figure 5.2  Hat section configuration (dimensions are in mm)  
Figure 5.3  Plain washer, narrow, steel, zinc plated (dimensions are in mm)  
Figure 5.4  Flat plate (dimensions are in mm)  
Figure 5.5  M10×1.25, class 8.8, hex bolt (dimensions are in mm)  
Figure 5.6  Bolted joint experimental setup  
Figure 5.7  Hat section and plate connected together with bolts  
Figure 5.8  The location of accelerometers  
Figure 5.9  Force curve applied to the finite element models  
Figure 5.10  Finite element modeling of the bolted joint structure  
Figure 5.11  Shell element structure with beam element bolts  
Figure 5.12  Solid element structure with solid element bolts  
Figure 5.13  FEA comparisons using hat shell-beam model with experimental results  
Figure 5.14  FEA comparisons using plate shell-beam model with experimental results  
Figure 5.15  FEA comparisons of hat section with experimental results (SRS)  
Figure 5.16  FEA comparisons of plate with experimental results (SRS)  
Figure 5.17  FEA comparisons using hat solid-solid model with experimental results  
Figure 5.18  FEA comparisons using plate solid-solid model with experimental results  
Figure 5.19  FEA comparisons of hat section with experimental results(SRS)  
Figure 5.20  FEA comparisons of plate solid-solid model with experimental results (SRS)  
Figure 5.21  Error of shell-beam finite element prediction versus filtering frequency  
Figure 5.22  Error of solid-solid finite element prediction versus filtering frequency  
Figure 5.23  Experimental time history response of the (a) hat section and (b) plate  
Figure 5.24  Experimental SRS of the (a) hat section and (b) plate  
Figure 6.1  Acceleration of the hat section (different contacts)  
Figure 6.2  Acceleration of the plate (different contacts)  
Figure 6.3  Hat section acceleration plots (different friction coefficients)  
Figure 6.4  Plate acceleration plots (different friction coefficients)
Figure 6.5. Structure showing the constant pre-stress of 460 MPa ....................................... 135
Figure 6.6. FFT of hat section for 100, 75 and 21Nm Torque ............................................... 136
Figure 6.7. Time History response on the structure .......................................................... 137
Figure 6.8. Hat accelerations – results output frequency ...................................................... 139
Figure 6.9. Plate acceleration – results output frequency ...................................................... 139
Figure 6.10. Comparison of hat section acceleration using zeroth moment (E & p) ................. 143
Figure 6.11. Comparison of hat section acceleration using first moment (E & p) .............. 143
Figure 6.12. Dissimilarity Factor for hat section acceleration (E & p) ............................. 144
Figure 6.13. Comparison of plate acceleration using zeroth moment (E & p) ....................... 146
Figure 6.14. Comparison of plate acceleration first moment (E & p) ................................... 146
Figure 6.15. Dissimilarity Factor for plate acceleration (E & p) ........................................... 147
Figure 6.16. Orientation of local axis in orthotropic bolt model ......................................... 148
Figure 6.17. Hat section acceleration versus time obtained from experiment and FEA ...... 151
Figure 6.18. Plate acceleration versus time obtained from experiment and FEA ........... 151
Figure 6.19. Hat section dissimilarity factor versus time span ........................................... 153
Figure 6.20. Plate dissimilarity factor versus time span ...................................................... 153
LIST OF TABLES

Table 3-1. PCB 086C02 Modally Tuned Impulse Hammer ...................................................39
Table 3-2. PCB 086C20 Modally Tuned Impulse Hammer ...................................................39
Table 3-3. Accelerometer information ..............................................................................40
Table 3-4. Accelerometer Calibrator information .............................................................41
Table 3-5. Pulse data acquisition hardware information ....................................................42
Table 3-6. Experimental natural frequencies of the round bar .........................................44
Table 3-7. Analytical natural frequencies of the round bar ................................................52
Table 3-8. Units used in the Modeling Analysis ..................................................................54
Table 3-9. Material properties of cold roll steel ...............................................................54
Table 3-10. Natural frequencies of the round bar obtained by finite element analysis .......58
Table 3-11. Natural frequencies of the round bar ............................................................60
Table 3-12. Experiment and FEA comparison of the solid cylinder ..................................61
Table 4-1. Units on the experiment and analysis ................................................................63
Table 4-2. Material properties of ASTM-A36 steel ........................................................64
Table 4-3. Mesh properties of shell element models of the plate .......................................70
Table 4-4. Mesh properties of solid element models of the plate .......................................72
Table 4-5. Modal analysis result of the plate ....................................................................73
Table 4-6. Comparison of the plate experiment and FE shell model ..................................76
Table 4-7. Comparison of the plate experiment and FE solid model ..................................79
Table 4-8. Mesh properties of shell element models of the hat section ...............................90
Table 4-9. Mesh properties of solid element models of the hat section ...............................91
Table 4-10. Modal analysis of hat section ..........................................................................93
Table 4-11. Error analysis of the time domain response of the hat section shell model .......95
Table 4-12. Error analysis of the time domain response of the hat section solid model ......98
Table 5-1. Mechanical properties of the bolted joint parts ................................................108
Table 5-2. Transient response comparison for bolted joint structure .................................121
Table 6-1. Comparison of experiment and FE model on bolted joint structure ..................127
Table 6-2. Coefficient of friction for steel surfaces ..........................................................131
Table 6-3. Comparison of finite element models with different contact area .....................134
Table 6-4. Natural frequency of structure .......................................................................136
Table 6-5. Simulation time step for finite element model ..................................................138
Table 6-6. Modulus and density of the bolt/spacer ...........................................................141
Table 6-7. Comparison of Hat section acceleration(bolt/spacer material properties) ..........142
Table 6-8. Comparison of plate acceleration (bolt/spacer material properties) ....................145
Table 6-9. Material properties of bolt and spacer modeled as orthotropic material ..........149
Table 6-10. Comparison of finite element models with orthotropic bolt and spacer ...........150
Table 6-11. Discritized hat section dissimilarity factor on impact time span .....................152
Table 6-12. Discritized plate dissimilarity factor on impact time span ..............................153
NOMENCLATURE

a Time location

A(t) Analysis result, data or curve (function of time)

e The value of the error

D Root mean square duration value

D² Mean square duration value

DF Dissimilarity Factor

E[ ] Expected value of [ ]

E(t) Error signal; Error curve (function of time)

f(x) Probability density function; pdf

K Kurtosis

mₙ The nth generalized moment

Mᵣ The rth temporal moment.

Max [ ] Maximum value of [ ]

N Sample size, Number of sample records

Pr[ ] Probability of [ ]

s Sample standard deviation

s² Sample variance

S Skewness

t Time variable

Var[ ] Variance of [ ]
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>( x )</td>
<td>Random variable</td>
</tr>
<tr>
<td>( \bar{x} )</td>
<td>Sample mean</td>
</tr>
<tr>
<td>( X(t) )</td>
<td>Experimental result, signal, data or curve (function of time), Time History response</td>
</tr>
<tr>
<td>(</td>
<td>[ ]</td>
</tr>
<tr>
<td>( \alpha )</td>
<td>Arbitrary point as the origin of the moment</td>
</tr>
<tr>
<td>( \lambda )</td>
<td>Time history energy</td>
</tr>
<tr>
<td>( \mu )</td>
<td>Population mean; Mean value</td>
</tr>
<tr>
<td>( \mu_n )</td>
<td>Central moments</td>
</tr>
<tr>
<td>( \mu'_n )</td>
<td>Raw moments</td>
</tr>
<tr>
<td>( \sigma )</td>
<td>Standard deviation</td>
</tr>
<tr>
<td>( \sigma^2 )</td>
<td>Variance</td>
</tr>
<tr>
<td>( \Sigma[ ] )</td>
<td>Summation of ([ ])</td>
</tr>
<tr>
<td>( \tau )</td>
<td>Centroid</td>
</tr>
<tr>
<td>( \psi )</td>
<td>Root mean square value</td>
</tr>
<tr>
<td>( \psi^2 )</td>
<td>Mean square value</td>
</tr>
</tbody>
</table>
ACKNOWLEDGEMENTS

The author is truly grateful to his advisor Dr. Brendan J. O’Toole, the Committee Chair Person for his guidance and encouragement throughout this investigation. Dr. Brendan J. O’Toole has been an inspiration to me in both academic and personal life. I would also like to thank Mr. Kumarswamy Karpanan whose suggestions and advices have been great help for me. Kumar is an individual with drive; he is a dedicated engineer and a great friend and colleague.

The author wishes to express his sincere thanks and heartiest gratitude to Dr. Samaan Ladkany, Dr. Woosoon Yim, Dr. Samir Moujaes and Dr. Douglas Reynolds for their time in reviewing the prospectus, participation of defense, and counseling of the thesis as the committee members.

I would like to thank my father, Mr. Abdollah Feghhi, and remember my beloved mother. What I owe them cannot be described by words. They have given me everything. Let it be known, this dissertation, every work I have ever done, and every work I will ever do is solely dedicated to my father and mother.

The financial support provided by the Army Research laboratory (ARL), under project BS3 is thankfully acknowledged.

The author expresses his thanks to the support and help of my friends and colleagues throughout this investigation.

xiii
CHAPTER 1

INTRODUCTION

1.1 Project Overview

It has been a while since scientists first started investigating different methods to find the response of a joint to an impact or shock. The finite element method has been very useful in the simulation of mechanical joints behavior. The finite element method is a powerful computer based mathematical analysis and design tool, which emerged with the advent of the high speed digital computer. Its development was pioneered during the 1950's and 1960's by structural engineers working in the aerospace industry. Since then it has been widely used for modeling and simulation of different linear and nonlinear problems, both static and dynamic in subjects of structural analysis, fluid flow, heat transfer, and fracture mechanics.

Mechanical joints, especially fasteners have a complex nonlinear behavior. The finite element method seems to be the only option for simulating the transient response of a joint under dynamic loading. Even this method has some limitations in simulating dynamic response. This study investigates the dynamic response of structures with and without joints to suggest a simulation method with the most accurate response. The first part of this study focuses on structures without any joints. Simple structures like a beam and a flat plate are employed for the simulation proposes.
Most of the time, simulation of a system response is the only way to understand the system behavior. There are many parameters to choose or ignore when it comes to building a model for the simulation. Picking the right parameters leads to a reliable simulation, and it is impossible to get an exact match between any simulation or analysis and experimental data. The goal of this work is to determine a satisfactory method for analyzing shock propagation across bolted joints and to provide experimental guidelines for verifying the analysis procedures.

1.2 Application

The main part of this study will be performed on a steel hat section bolted to a flat plate. These simple configurations are representative of structures found in many military ground vehicles that can be subjected to transient impact and blast loads. This is the main application of this project. In order to understand the response of a structure, we must have a good understanding of its components. Joints are the key components of structures. Almost every structure uses one or a mixture of mechanical joints such as welding, adhesive bonding and mechanical fasteners. Extensive research is in progress to analyze the dynamic response of complex structures involving assemblies, such as a light combat vehicles. This study evaluates the structural integrity of such structures when they are subjected to transient loading [1].

Joints play a very important role in maintaining the structural integrity of a combat vehicle. Non-linear shock transfer performance of joints has substantial influence on the dynamics of assembled structures as they induce a large amount of damping into the structure [2]. Study of shock transmission through the various jointed (both
mechanical and adhesive) components of the combat vehicle is of particular interest to
the army. There is a need to guarantee the survivability and minimize the damage caused
to both the primary and secondary electronic systems present inside the combat vehicle.
Another area of concern is to reduce or damp the shock transmission caused by a
projectile impact. On a armored vehicle, there is an immediate need to develop
methodologies for constructing predictive models of structures with joints and shock
based dynamic response analysis in order to ensure the safety of critical equipment and
hardware [3].

1.3 Problem Configuration

Many military systems must be capable of sustained operation in the face of
mechanical shocks due to projectile or other impacts. Many Army platforms (such as
vehicles) are made of the chassis and top part, which are usually bolted together. Figure
1.1 shows the Dingo armored vehicle [4], which is made of top part and chassis bolted
together. The vehicle consists of several parts, which some of them that can be clearly
seen are the tires, driver and commander doors (dome-shaped which open upwards),
latches, and connections. Several of the components are joined together with bolts
through flanges. It is nearly impossible to model all the bolted connections with complete
detail because of computational limitations

3

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
It is important to understand the physical mechanism of shock transfer through bolted connections, so that simplified, but accurate modeling methods can be incorporated into large vehicle design models. This dissertation focuses on developing an understanding of shock propagation through a bolted structure that is typical to a variety of military vehicle structures (Figure 1.2). The bolted hat section and plate structure (was selected for study based on numerous discussions with structural dynamic research staff at the U.S. Army Research Laboratory (ARL). Impact loads to this structure cause axial, bending and shear shock loading through bolted connections.
The finite element analysis of the vehicle model is carried out in steps which are listed briefly as follows.

Step 1: Geometry creation in the pre-processor Altair HyperMesh® directly.

Step 2: Material definition, meshing, application of boundary conditions along with appropriate contact definitions, application of the loading curve that best simulates the real life loading scenario, using Altair HyperMesh®.

Step 3: Perform the modal analysis using Altair OptiStruct®.

Step 4: Solving the problem using LS-DYNA, a nonlinear finite element solver.

Step 5: Post-processing using Altair HyperView® or LS-POST to view the analysis results.

Figure 1.2. The simplified model of the armored vehicle
1.4 Review of Literature

Little work has been published on the study of shock transmission through jointed structures; however there has been a great deal of work done on both shock propagation in structures and static analysis of jointed structures. The design of structural systems involves elements that are joined through bolts, rivets and pins. Joints and fasteners are used to transmit loads from one structural element to another. In structures, there are three types of joints commonly used, namely, welded, mechanically fastened joints and adhesive bonded joints. Fastened joints include bolts, rivets, and pins [3, 5].

Despite the adhesive joints being used for joining secondary structures, bolting and welding are the main solution for joining the crucial structure parts. Nevertheless it cannot be said that one particular type of joint is better than the other as all the joints have their own advantages. For instance adhesive bonding offers improved joint stiffness compared to mechanical fasteners. An adhesive is essentially used for dual purposes, it not only provides mechanical strength but it also seals the joint against moisture and debris ingress [5].

The joint represents a discontinuity in the structure and results in high stresses that often initiate structural failure [3]. The complex behavior of connecting elements plays an important role in the overall dynamic characteristics of structures. This complex behavior can be the effect of slip in contact area around the bolted joints [6-9].

Detailed finite element models have been developed to establish an understanding of the slip-stick mechanism in the contact areas of the bolted joints [6]. Bolted or riveted joints are the primary source of damping in the structure, because of the friction in the contact area [2]. The nonlinear transfer behavior of the frictional interfaces often provides
the dominant damping mechanism in jointed structure. They play an important role in the vibration properties of the structure [7].

Friction in bolted joints is one of the sources of energy dissipation in mechanical systems. The finite element models are constructed in a nonlinear framework to simulate the energy dissipation through joints [9]. Sandia National Laboratory also has an extensive research program for investigating energy dissipation due to microslip in bolted joints [8].

‘Preload’ and ‘mechanical clearance’ are two parameters that might effect the dynamic behavior of bolted joints. Most of the research in the modeling of preload has been done for fatigue or cyclic loading. These kinds of loads are usually in the category of the static loading, but because of the importance of these parameters it is useful to mention them in dynamic response of the joints. Duffey, Lewis and Bowers [10] present two types of pulse-loaded vessel closers to determine the influence of bolt preload on the peak response of closure and bolting system. The effect of bolt prestress on the maximum bolt displacement and stress has been investigated by Esmalizedeh et al [11]. The loading is assumed to be initially peaked, exponentially decaying internal pressure pulse acting on the bolted closure. Kerekes [12] use a simple beam model of the screw with fatigue loading to show the damage vulnerability of prestressed screws on the flange plate. In all of these studies there is no indication about how to apply the preload to the finite element model. O’Toole et al. [13] show several different preload modeling procedures for dynamic finite element analysis and make recommendations on the most suitable methods. Szwedowicz et al. [14] presented the modal analysis of a pinned-clamped beam for three different magnitudes. They have determined that even for fine mechanical fit
with the maximum bolt clearance up to 5 μm, the analytical and numerical eigenfrequencies above the 2nd mode show discrepancies with the measured results.

Zhange and Poirier [15] have developed a new analytical model of bolted joints. In this model, the member deformation is determined by the member stiffness that remains unchanged whether the external load is present. They have used finite element analysis to confirm the new model and observation. Song et al. [16, 17] have developed an Adjusted Iwan Beam Element (AIBE), which can simulate the non-linear dynamic behavior of bolted joints in beam structures. The same element was used to replicate the effects of bolted joints on a vibrating frame; the attempt was to simulate the hysteretic behavior of bolted joints in the frame. The simulated and experimental impulsive acceleration responses had good agreement validating the efficacy of the AIBE. The beam element developed is two-dimensional and consists of two adjusted Iwan models and maintains the usual complement of degrees of freedom: transverse displacement and rotation at each of the two nodes. This element includes six unknown parameters. A multi-layer feed-forward neural network is considered to obtain these parameters, from measured acceleration responses. The experimental result has been used to validate the simulated acceleration responses [17].

Different methods have been employed to determine the dynamic response of complex jointed structures. Studying the natural frequencies, modal behavior and damping of a structure, which constitute its dynamic characterization, gives us a better understanding of the dynamics of a structure and its reliability [18]. The Frequency Response Function (FRF), which is obtained from Fast Fourier transform (FFT), is the widely used method for determining the natural frequencies and mode shapes of a
structure [19]. Nevertheless it is possible to determine the natural frequencies of a structure using FFT; determining the conspicuous peaks in the FFT analysis does this, the frequencies corresponding to these peaks are the natural frequencies of the structure [20].

Responses measured from impulsive loading (like blast or impact) are typically accelerations, velocities and displacements at the crucial locations on the structure. While comparing the finite element results with the results obtained from experiments, one of these parameters is considered [21]. Little work has been published on presenting the study of shock transmission through jointed structures; however there has been a great deal of work done on both shock propagation in structures and jointed static analysis of joints.

A few finite element models for joints are being developed [22, 23] which can predict the dynamic response for a particular application. Adoption of this type of analysis early in the design phase can influence decisions that improve the structural performance. Crash modeling and simulation is one of the subjects that finite element analysis has been employed to obtain the dynamic response of the whole structure, including joints. A truck impacting a guardrail system is one of the examples of these crash analyses [22]. In this study a spring has been used to simulate component crashworthiness behavior, like the bolted connection between the rail and block-out.

For the safety of the driver of a delivery motor vehicle, a new concept of breakaway mailbox support has been developed by Reid [23]. The new breakaway concept consists of modifying the material of anchor bolt to have a higher strength and lower percent elongation. Nonlinear finite element analysis with LS-DYNA was also used to predict the potential for the new breakaway mount and attached mailbox to meet
the crash test requirements of NCHRP Report No. 350 [24]. Most of these research efforts have followed the Federal Highway Administration (FHWA) safety performance criteria. FHWA policy requires the use of devices on the National Highway System that have been successfully tested in accordance with the guidelines contained in the National Cooperative Highway Research Program (NCHRP) Report 350, “Recommended Procedures for the Safety Performance Evaluation of Highway Features”. The procedure in ‘NCHERP report 350’ requires the use of dynamic time history response to verify the finite element simulation with experimental results [24].

Semke et al. [20] has analyzed the dynamic response of a piping system with a bolted flange. Experimental and numerical results are presented and show excellent correlation. The experimental procedure utilizes an accelerometer to gather the dynamic response output of the piping system due to an impulse. The resonant frequencies are then determined using a Fast Fourier Transform (FFT) method. The dynamic effects of a bolted flange and gasket on a piping system are critical in their use and it has been demonstrated that the finite element method can simulate the response of an overhanging beam with a varying mid span.

1.5 Dissertation Objectives

The aim of this study is to analyze and assess the dynamic behavior of bolted joint connections subjected to impact loads using finite element analysis. In other words, the objective is to develop solutions that enable designers to generate improved physics-based shock models for structures focusing mainly on shock transmission across structural joints. The first step is to study the response of individual components that
make up the bolted system. Transient analysis and experiments are performed on a flat plate and a single hat section to benchmark both methodologies. Then similar analysis and experiments are performed on an assembly with multiple joints.

The goal is to perform a detailed analysis of a jointed structure that verifies a response within 15 to 20 percents of experimental data and shows quantitatively the effect of joint configuration on structural response. The following steps have been employed and presented in the following chapters to study the response of the jointed hat section:

- Choosing a proper comparison factor to quantify the difference between time histories.
- Perform FFT analysis on the structures without the joints and compare the natural frequencies obtained from the finite element analysis.
- Perform impact experiments on the structures without the joints, which will provide input data (force vs. time) and response data (acceleration and/or strain vs. time).
- Demonstrate that this experiment can be computationally simulated using a detailed 3-D LS-DYNA analysis.
- Investigate the ability to accurately simulate the structural response for the structures without joints
- Describe a simulation procedure, which obtains the most accurate dynamic response of a structure.
- Verify the simulation procedure on the geometrically nonlinear bolted joint structures.
CHAPTER 2

COMPARISON OF TWO TRANSIENT RESPONSES

2.1 The Need for Establishing Error Criteria

This chapter investigates the methods for comparing the transient response from experiments and analyses. The idea presented in this chapter can be applied to any experimental verification results. Our particular interest is to compare the dynamic acceleration predictions from different models of a structure under an impact with experimental results. A typical time response of a structure is shown in Figure 2.1.

![Figure 2.1. A typical result of a shock response of a structure from the experiment](image-url)
This graph shows the acceleration of a particular point in the structure caused by an impact force or shock. The response presented in Figure 2.1 is only an experimental result. Figure 2.2 shows the acceleration of the same point obtained from a finite element model with shell elements, and the experimental result. Figure 2.3, shows the acceleration of the same point from the experiment and a finite element model with solid elements. The experimental results in both Figure 2.2 and Figure 2.3 are the same. The only difference between these two plots is the results obtained from the two different finite element models.

![Graph showing acceleration vs. time](image)

*Figure 2.2. Shell element model and experimental results*

The purpose of these graphs is to show the experimental verification of the finite element model. Which finite element analysis procedure provides a better match to the experimental data? This is a complex question that may have different answers depending
on which criterion is used to compare the curves. Visual inspection of the curves is one method for comparing the curves. However it is often used in the technical literature, it may not be reliable. The need for a quantitative Error Criteria seems unavoidable for comparing the transient responses from finite element models and experiment.

![Figure 2.3. Solid element model and experimental results](image)

2.2 Applications

The application of this study is not limited to the vibration of structures. This problem can be applied to any subject where there is an interest in comparing two sets of random data, two signals or two curves. In the previous section, an engineering application was used as an example to explain the problem. In this section we are going to use a totally different area to show that the application of this study goes beyond the engineering problems. There are a lot of sources to obtain the seven days weather
forecast, but which one is more accurate and reliable? Obviously the only way to investigate the reliability of these sources is looking at their previous records. To answer this question we need to look at the seven days forecasted temperature and the measured temperature after seven days. To treat each source with the fair condition, we compare each source forecast with its own measured temperature after seven days. Figure 2.4 and Figure 2.5 show the forecasted and recorded temperature from the Weather Channel and Fox News in Las Vegas from Nov 15, 2005 to Dec 30, 2005 [25, 26].

![Graph showing temperature forecast and measured data from Fox News from Nov 15, 2005 to Dec 30, 2005.]

Figure 2.4. Seven days forecast and recorded temperature from Fox News [25]

It is hard or maybe even impossible to recommend any of these sources without having a comparison criteria. This example shows the application of error criteria is not limited to the engineering applications and can be useful in many other subjects.
2.3 Error Criteria Objectives

A consistent error criterion for comparing two transient curves was not found in the structural dynamic literature. Several different methods are used by most researchers. The most common methods are reviewed in this chapter. Advantages and disadvantages are discussed for each method and recommendations are made for the most suitable for structural dynamics problem.

One might be interested to determine which set of curves presented in Figure 2.6 is a better match. The word ‘set’ has been used because we are interested in comparing two pairs of curves. Generally there is one experiment and one analysis in each pair. The objective is to determine which pair is more similar.

This illustration shows that the visual judgment is very subjective. If someone says the set on the right plot in Figure 2.6 is closer to the experiment, unless he or she does not bring a logical explanation the conclusion is not valid.
2.4 Review of the Literature in Transient Response Comparison

Research in different disciplines has addressed this problem. Subjects like signal processing, statistics, stochastic analysis, time series analysis, random vibration and quantitative finance have talked about error calculation or error analysis. Also, the subject of error analysis might have been discussed under different titles like “Comparison of Two Signals” and “Difference between Two Time Series” [27-29].

In the field of statistics, error is discussed in the regression analysis. Root mean square [29] is widely used in calculation of error between the best fit and original data [30]. Comparison of two sets of random variables is another subject in statistics, which can be related to this study. In order to compare two populations, we can compare their means (t-test). If the means are in desired confidence level, we can also compare the variances (F-test). Analysis of Variance ANOVA, is a statistical test for comparison of means by analysis of group variances. Almost every statistics textbook has a chapter on this subject, so there is no need to mention any reference for these methods. Although these methods are the most general and reliable for comparison, they do not quantify the similarity of two sets. Running these tests on your data, is going to show whether the two

Figure 2.6. Illustration of the problem with two sets of analysis and experimental curves
sets of data are close or not, but they are not able to determine how much these sets of data are close together.

The moment method is another way for comparison of two sets of data. This method is also used by the scientists in the subject of signal processing [28] and statistical signal processing subjects [27]. In this method each signal or curve will be represented with its moment, like raw moments or central moment [31]. Federal Highway Administration (FHWA) [24] has a validation procedure for comparing two signals. The main part of this procedure uses method of moments for validation of models with tests or experimental results. Smallwood [32] and Cap [33] use the band limited temporal moments to calculate a normalized error between two transient time histories. With this method, they calculate the normalized error over different bandwidths.

Geers [34] defines an error measure for the comparison of calculated and measured transient time histories. His suggested error factor assigns a single numerical value to the discrepancy existing over a specified comparison period. Information regarding the nature of the discrepancy is provided by the magnitude and phase error factors, which constitute orthogonal components of error. Geer's work had been followed by Whang, Gilbert and Zilliacus [35]. They have introduced two correlation measures for comparing calculated and measured response histories. The first one is an error index, which is a simplification of root sum square error factor. The second one is an inequality index that is a simplification of Theil's Inequality.
2.5 Error Calculation Methods

The error calculation methods can be divided into two different categories: the full and the partial error calculation methods. The full methods calculate the error over the whole curve such as root mean square, while the partial calculation methods consider specific characteristic of the curve as a criterion. In partial calculation method, the error is defined as the error between the characteristics of each curve. For example, one can pick 'the maximum peak' as a criterion. With this criterion, the error would be the difference between the maximum peaks of two sets of data.

The full error calculation methods are regular (common) method, root mean square (RMS), band limited temporal moment method and the method of moments. The partial error calculation methods are: 'error in maximum value' and 'peak counting methods'. Of course there are more error criteria than the methods discussed here, but they have either no application in our problem or they do not quantify the error as a value for comparison purposes.

2.5.1 Regular Method

This is the most common method used for error calculation. One value is used as the reference when calculating the error with this method. For example the analytical answer can be the reference if it is available. Error can be calculated as the difference between two curves at each particular point (or time), divided by the reference value. The following formula shows how to calculate the error between finite element model and experiment.

\[ E(t) = \frac{X(t) - A(t)}{X(t)} \]
where $X(t)$ is the acceleration measured by an accelerometer mounted on a vibrating structure and $A(t)$ is the acceleration of the same point obtained from finite element model. Since $X(t)$ and $A(t)$ are functions of time, the error is also time dependent. For the sake of comparison, we need a single value over a comparison time period. The regular error method fails to do that. A suggestion to solve this problem is to take the average values of the error to get a single number.

The accelerations plotted in Figure 2.7 are from experimental and finite element analysis of a rectangular steel flat plate. Shell elements have been used to model the plate for finite element analysis. The accelerometer has been placed 0.52 m from impact point. An instrumented hammer has been used to excite the plate.

![Figure 2.7. Accelerations obtained from experimental results and shell finite element model](image)

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 2.8 shows the error signal calculated with the regular method. The error at each instant is the difference between two accelerations divided by the experimental acceleration at that instant.

This method has two disadvantages that make it a less suitable method for the application of this study. In fact because of these reasons, it is not useful for many applications. The first problem is that every time the reference signal becomes zero, the error is not defined. In the case of the flat plate presented in Figure 2.7, the experimental signal is considered as the reference signal. Every time this signal has a value of zero, the value of error is not defined. Most computer software including MS Excel and MATLAB, substitute a very large number when a number is divided by zero (∞). That is why there are big spikes in the error plotted in Figure 2.8. The actual number shown on
the plot is random because the plotting software substitutes a large number for $\infty$. This does not mean there is a large error on those instances. It simply means that the value of experimental signal is zero on those instances. The regular method does not quantify the similarity of the two curves (signals). This is the second disadvantage of this method. It means we do not obtain a single value for the error. This problem makes it impossible to use this method for the comparison proposes.

2.5.2 Root Mean Square (RMS)

Root mean square is the most common used error criteria in statistics. This method is generally used in regression analysis. The following formula shows how to calculate the mean square value of the experimental and finite element results.

$$\psi^2 = \frac{\sum_{i=0}^{N} (X_i - A_i)^2}{\sum_{i=0}^{N} (X_i)^2}$$

where $X(t)$ is the experimental result, and $A(t)$ is the analysis result.

2.5.3 Characteristics of root mean square

Error generated in a time history response can be the effect of two reasons; the difference between amplitudes and the phase shift. The RMS has a very interesting characteristic in that it can detect both phase shift and amplitude difference. The following example proves that the root mean square obtains the same error for two sets of curves with the same phase shift. Consider the following sets of curves presented in Figure 2.9 (a) and (b). Here are the mathematical expressions of these sets:
The mathematical representations of first set of curves show that there is no difference in amplitudes, but they have a $90^\circ$ phase shift. The curves presented in the second set have the same phase shift with no change in amplitudes. This means that these two sets of curves are exactly similar to each other. In other words one cannot say that $y_1$ and $y_2$ are more similar compare to $y_3$ and $y_4$ or vice versa. Generally speaking the error between $y_1$ and $y_2$ is equal to the error between $y_3$ and $y_4$. See if we can confirm this by calculating the root mean square ($\psi$). Here is calculated root mean square for each set of curves.

$$\psi (y_1, y_2)= 1.41$$  \hspace{1cm} (Corresponding to Fig. 9(a))

$$\psi (y_3, y_4)= 1.41$$  \hspace{1cm} (Corresponding to Fig. 9(b))
With a similar example, it can be shown that one can obtain an almost equal RMS for signals with the same difference in amplitudes. The only disadvantage of this method is that there is only one value for the error. This means if there is both phase shift and amplitude difference in two signals, the RMS will show the difference, but it is impossible to determine which one of these sources contributes more in generating the error.

2.5.4 Moment Method

In order to compare two curves, we can compare the relative absolute difference of characteristic values. Each set of data can be characterized by a few numbers that are related to its moments. The moments of a random variable are the expected values of its powers [31]. It is assumed that if the difference between two signals in terms of a particular order is less than 20 percent, the signals are considered to be sufficiently close to one another [24]. It is useful to review the definitions and formulas of different moments. The nth generalized moment of x about a point (α), can be written [36] as

\[ m_n = E[(x - \alpha)^n] = \int_{-\infty}^{\infty} (x - \alpha)^n f(x) \, dx \]

where \( f(x) \) is the probability density function. The moment about the mean of a random variable \( x \), denoted by \( \mu_n \) is called central moment [36].

\[ \mu_n = E[(x - \mu)^n] = \int_{-\infty}^{\infty} (x - \mu)^n f(x) \, dx \]

The raw moments are the moments about zero (origin).

\[ \mu'_n = E[x^n] = \int_{-\infty}^{\infty} x^n f(x) \, dx \]

The central moments and raw moment are related to each other. The relation between them can be found in [31, 36]. This means either one can represent a curve.
Instead of presenting a curve with its consequence moments, we can present it with the meaningful quantities (like mean, variance, ...) derived from its moments.

**The mean**

The first raw moment is called mean ($\mu$).

\[
\mu = \mu' = E[x] = \int_{-\infty}^{\infty} xf(x) \, dx
\]

For discrete uniform distribution, $f(x) = \text{Pr}[x=k] = \frac{1}{N}$, where $N$ is the number of collected data. In this case, the *mean* is simply the average of these data values. Therefore, it is the sum of the data values divided by $N$ [37].

\[
\bar{x} = \frac{\text{sum of data values}}{N} = \frac{\sum x}{N}
\]

The symbol $\Sigma$ represents the sum of data values.

**The Variance**

The second central moment is called variance ($\sigma^2$).

\[
\mu_2 = E[(x - \mu)^2] = \int_{-\infty}^{\infty} (x - \mu)^2 f(x) \, dx
\]

For a discrete uniform distribution with $N$ data values, the sample variance can be defined as [37]:

\[
s^2 = \frac{\sum (x - \bar{x})^2}{N} = n \frac{\sum x^2 - (\sum x)^2}{N(N-1)}
\]

The variance sometimes is denoted by var(X) or $V(X)$. The square root of variance is called standard deviation and it is denoted by ($\sigma$).

**Skewness**

The third central moment is called skewness.

\[
\mu_3 = E[(x - \mu)^3] = \int_{-\infty}^{\infty} (x - \mu)^3 f(x) \, dx
\]
Skewness is the symmetry of a distribution about its mean. Figure 2.10 shows two distributions with positive and negative skewness [38].

![Figure 2.10. Example of curves with positive and negative skewness](image)

If the curve, at the left side of the mean line, is more stretched compared to the right side, then it has positive skewness. If the reverse is true, it has negative skewness (Figure 2.10). If the curve is equally stretched on both sides on the mean line, it has zero skewness. For example, the normal distribution has zero skewness. Some literature use normalized skewness instead of the skewness. The normalized skewness of a distribution is defined to be

$$\mu'_3 = \frac{\mu_3}{(\mu_2)^{3/2}}$$

**Kurtosis**

The fourth central moment is called kurtosis.

$$\mu_4 = E[(x - \mu)^4] = \int_{\infty}^{\infty} (x - \mu)^4 f(x) \, dx$$

Kurtosis is the peakedness of a distribution. The normalized kurtosis is defined as
For a normal distribution, kurtosis is equal to 3. Figure 2.11 shows a curve and plotted with its normal distribution in the same graph. If the value of kurtosis is more than 3, there would be presence of peaks of high value. In this case the peak of the probability density function is higher than its normal distribution. The curves with kurtosis less then 3 have a flat probability density function, and they have a smaller peak compare to their normal distribution [38].

![Figure 2.11. Example of curves with kurtosis greater and less than 3](image)

The moment method defines a curve with four quantities. In order to compare two curves, we can compare the moments of each curve, with the other one, i.e. 1st moment with 1st moment, 2nd moment with 2nd moment and so on. Having more than one quantity for comparison makes it easier to understand the source of error. The moment method shows whether the error is coming from amplitude difference or phase shift. On the other hand, having a couple of quantities as error instead of one value, some times would be confusing. For example assume that signals Y_1 and Y_2 show small error in 1st and 2nd
moments but large error in 3rd and 4th moments. Vice versa, signals Y3 and Y4 show large error in 1st and 2nd moments but high error in large error in 3rd and 4th moments. In this case or cases similar to this example, there would not be a solid result of the similarity between two sets of signals, and it is dependent to the user’s interpretation. In order to compare two signals with the moment method, they must be stationary. It means that all of their statistical properties should not vary with time. Because of this property the application of moment method is very limited.

2.5.5 Method of Temporal Moments

The temporal moments [32] are like the moments of probability density functions, but with different functionality. The rth temporal moment is defined as:

\[ M_r(\alpha) = \int_{-\infty}^{\infty} (t - \alpha)^r X^2(t) \, dx \]

where \( m_r(\alpha) \) is the rth temporal moment. X(t) is the time history and \( \alpha \) is the time location. The centroid is defined as the point \( \tau \) where the first moment is equal to zero.

\[ M_1(\tau) = 0 \Rightarrow \tau = \frac{M_1}{M_0} \]

The zero order moment is independent of the shift (\( \alpha \)) and the centroid (\( \tau \)). The zero order moment is called time history energy.

\[ \lambda = M_0 \]

The second moment normalized by the energy is defined as the mean square duration (\( D^2 \)) of the time history.

\[ D^2 = \frac{M_2(\tau)}{\lambda} \]

The third temporal moment normalized by the root mean square duration is skewness.
The skewness presents the shape of the function, as it was described on the moment method. A positive skewness indicates high amplitudes on the left of the centroid, and a long low amplitude tail on the right of the centroid.

The forth normalized central moment is called kurtosis. The kurtosis is useful for time histories that have more than one maximum.

\[ K^4_i = \frac{M_4(t)}{\lambda} \]

\[ K = \frac{K^4_i}{D} \]

The objective is to characterize each time history with as few as parameters as possible. The first few parameters are the centroid \( (\tau) \), the time history energy \( (E) \), the root mean square duration \( (D) \), the normalized skewness \( (S) \), the normalized Kurtosis \( (K) \). Each time history can be represented with these parameters \( (\tau, \lambda, D, S, K) \). In order to compare two time histories, we can compare the characteristics of them. The method of temporal moments only characterizes the transient time histories, so it is not applicable for the cases that part of transient response is in the interest of the researchers. Also, this method is not applicable for the time histories that cannot be divided to transient and steady state response like the weather forecast example.

### 2.5.6 Maximum Peak

In this method we consider a particular characteristic of the curve as the error criterion. The error calculated with this method does not represent the error between whole curves. It only presents the error of that particular criterion. In the maximum peak method the error is defined as the difference between maximum values. In the case of
existing both positive and negative values, the absolute values must be considered for error calculation.

\[ e_{\text{peak}} = \frac{\text{Max}(|X(t)|) - \text{Max}(|A(t)|)}{\text{Max}(|X(t)|)} \times 100 \]

To illustrate this method let's present the experimental and analysis results of the quarter inch steel plate under the effects of an impact for 0.004 s (Figure 2.12).

![Illustration of maximum peak error calculation method](image)

Figure 2.12. Illustration of maximum peak error calculation method

The absolute maximum experimental and analysis accelerations are 1361.6 and 1136.1 m/s² respectively.

Peak Error: \[ e = \frac{1361.6 - 1136.1}{1361.6} = 16.5\% \]

30
This method will be applicable in particular cases where only the maximum acceleration is important. This method is time independent. In other words, to use this method, the time at which maximum acceleration happens, should not be important. Having a small error between the maximum value of two curves or signal does not mean that those two curves or signals are similar, unless it is justified properly. Even if it is justified by the author, it is not expandable to other cases and it is valid only for that particular case.

2.5.7 Peak Counting Methods

This method is another way to approach the comparison problem with considering partial characteristics of a curve or signal. The criterion used in this method is the number of times a signal exceeds a certain value. This method was originally developed for the study of fatigue damage in structures [38], but with a little modification it can be used for the comparison of two signals.

The objective is to count the peaks above or below a certain value. This value can be any number, like the mean of amplitudes or any other predefined value. If both signals have the same number of peaks above that value, they are said to be similar to each other. This is the simplest way to apply the peak counting method. Depending on the accuracy of the similarity of two signals, one can use other peak counting methods such as Range-restricted peak count or Level-restricted peak count [38] for the purpose of comparison.

Let's re-demonstrate the experimental and analysis results of the quarter inch steel plate under the effects of an impact for 0.004 seconds (Figure 2.13). With this method, we would like to compare the two accelerations with number of times they exceed 1000 m/s^2.
The experimental result shows that the acceleration of the structure twice exceeds 1000 m/s^2. The finite element analysis shows only one peak higher than 1000 m/s^2. In this case there would be a 50% error between experimental and analysis result.

2.6 Summary of Error Calculation Methods

This study searches for a quantified comparison factor to compare two sets of data, curves or signals. The error generated between two curves can be the effect of phase shift or amplitude difference or both. The error calculation methods can be divided to either the full or partial methods. The full methods calculate the error over the whole curve such as root mean square, while the partial calculation methods consider specific characteristic of the curve as a criterion. Regular Method, Root Mean Square (RMS),
Moment Method, Method of Temporal Moments, Maximum Peak and Peak Counting Methods are the methods presented in this study.

Regular method is easy to use, but the error calculated with this method is time dependent. For the sake of comparison of two signals, we need a single value over a comparison time period. This makes the regular method not applicable for the objective of this study. The moment method determines whether the error is coming from amplitude difference or phase shift, but in order to compare two signals with the moment method they must be stationary. This property of moment method makes it limited for most applications, and not useful for comparison purposes. The method of temporal moment characterizes the transient time histories, so it is not applicable for the cases that part of transient time history is in the interest of the researchers. This method is not applicable for the time histories that cannot be divided to transient and steady state response.

Both maximum peak method and peak counting method consider a particular characteristic of a curve as error criterion. In many cases, the peak amplitude is the result that is of most interest; therefore it makes sense to use this criterion to compare two signals.

Root mean square can detect both phase shift and amplitude difference. Since only one value for error can be obtain from this method, it is impossible to determine the source of the error. Having said that, having a single value as the error quantity makes it easy to compare two sets of curves.
All of these methods have advantages and disadvantages. Unfortunately, there is not a single criterion that seems to be the best for comparing transient acceleration curves due to impact loading.

2.7 The Dissimilarity Factor (DF)

The Dissimilarity Factor (DF) is defined to have a single value for comparison of two sets of curves. The dissimilarity factor is a linear combination of some of the error’s defined in previous sections. The general form of the dissimilarity factor as follows:

$$DF = w_1 E_1 + w_2 E_2 + w_3 E_3 + w_4 E_4$$

where $E_1$ ... $E_4$ are the values of error and $w_1$ ... $w_4$ are the weights assigned to each error. The summation of all weight factors must be one, i.e. $w_1+w_2+w_3+w_4=1$. The maximum peak value, zeroth, first and second moment method are the four error method contributing to the calculation of dissimilarity factor. The maximum peak value, zeroth and first moment method detect the amplitude difference. In the application of this study, the difference between amplitudes is more important, so the weight factor assigned to these errors is twice as the weight factor assigned to the second moment method. So, the final form of the dissimilarity factor can be written as follows:

$$DF = \frac{2}{7} E_1 + \frac{2}{7} E_2 + \frac{2}{7} E_3 + \frac{1}{7} E_4$$

where $E_1$ is the maximum peak value, $E_2$ is the zeroth moment, $E_3$ is the first moment and $E_4$ is second moment.
CHAPTER 3

EXPERIMENTAL CALIBRATION OF FINITE ELEMENT ANALYSIS

3.1 Introduction

This chapter investigates the dynamic response of a simple structure used for impact testing. Performing the experiment on a simple structure is the most important part of any type of experimental verification projects. The first step of every experimental project is to determine whether the method of approach is able to solve a simple problem or not. This chapter demonstrates the procedure and setup for experimental verification of the Finite Element Analysis (FEA) of a simple structure. A solid bar or beam is one of the simplest structures that can be employed for the testing of dynamic behavior of the structures. Modal analysis is the first step in investigation of dynamic response of structures. Once the modal analysis of the system is verified experimentally, we can look at the time history response of a shock or impact to the system. The impulse hammers are used to excite a system with a known input force so the dynamic response of the system can be analyzed.

3.2 Geometry of the Bar

Figure 3.1 shows the dimensions of the solid cylinder with the base diameter of 0.0381 m and height of 0.1968 m. The mass of the cylinder is 1.711 kg.
The cylinder has been made of steel. The material properties of the steel will be mentioned in the modeling analysis part.

3.3 Experimental Procedure

The experimental procedure is the same throughout the rest of the dissertation. The only difference between experiments throughout this project is the structure, which is the subject of the experiment. First the equipment used in this project has been described, and then test setup and experimental procedure is explained. The last part of this section talks about experimental results obtained from the solid uniform bar.

3.3.1 Equipment

An accelerometer and impact hammers with hard tips are used to make the necessary measurements for the test. The impact hammers used in the experiments can be seen in Figure 3.2 – 3.4. Figure 3.5 shows the accelerometer used in the experiment. The pertinent information for the impact hammers and the accelerometer can be found in Tables 3.1 -3.3.
Figure 3.2. PCB 086C02 impulse hammer (Small)

Figure 3.3. PCB 086C20 impulse hammer (Large)
Figure 3.4. Comparison of Large and small impulse hammers

Figure 3.5. PCB 352C22 ceramic shear ICP accelerometer
Table 3-1. PCB 086C02 Modally Tuned Impulse Hammer [39]

<table>
<thead>
<tr>
<th>Model No.</th>
<th>086C02</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Performance</strong></td>
<td>Units (SI)</td>
</tr>
<tr>
<td>Sensitivity (± 15%)</td>
<td>11.2 mV/N</td>
</tr>
<tr>
<td>Measurement</td>
<td>± 440 N pk</td>
</tr>
<tr>
<td>Frequency Range (Hard Tip)</td>
<td>8 kHz</td>
</tr>
<tr>
<td>(Medium Tip)</td>
<td>2.5 kHz</td>
</tr>
<tr>
<td>Resonant Frequency</td>
<td>≥ 22 kHz</td>
</tr>
</tbody>
</table>

**Physical**

| Sensing Element | Quartz                  |
| Hammer Mass     | 0.16 kg                 |
| Extender Mass Weight | 75 gm                  |
| Head Diameter   | 1.57 cm                 |
| Tip Diameter    | 0.63 cm                 |
| Hammer Length   | 21.6 cm                 |

Table 3-2. PCB 086C20 Modally Tuned Impulse Hammer [39]

<table>
<thead>
<tr>
<th>Model No.</th>
<th>086C20</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Performance</strong></td>
<td>Units (SI)</td>
</tr>
<tr>
<td>Sensitivity (± 15%)</td>
<td>0.23 mV/N</td>
</tr>
<tr>
<td>Measurement</td>
<td>± 22,000 N pk</td>
</tr>
<tr>
<td>Frequency Range (Hard Tip)</td>
<td>1 kHz</td>
</tr>
<tr>
<td>Resonant Frequency</td>
<td>≥ 12 kHz</td>
</tr>
</tbody>
</table>

**Physical**

| Sensing Element | Quartz                  |
| Hammer Mass     | 1.1 kg                  |
| Head Diameter   | 5.1 cm                  |
| Tip Diameter    | 5.1 cm                  |
| Hammer Length   | 37 cm                   |
Table 3-3. Accelerometer information [39]

<table>
<thead>
<tr>
<th>Model No.</th>
<th>352C22</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Performance</strong></td>
<td>Units (SI)</td>
</tr>
<tr>
<td>Sensitivity (± 15%)</td>
<td>1.0 mV/(m/s²)</td>
</tr>
<tr>
<td>Measurement</td>
<td>± 4900 m/s² pk</td>
</tr>
<tr>
<td>Frequency Range (± 5%)</td>
<td>1.0 to 10,000 Hz</td>
</tr>
<tr>
<td>(± 10%)</td>
<td>0.7 to 13,000 Hz</td>
</tr>
<tr>
<td>Resonant Frequency</td>
<td>≥ 50 kHz</td>
</tr>
<tr>
<td><strong>Physical</strong></td>
<td></td>
</tr>
<tr>
<td>Sensing Element</td>
<td>Ceramic</td>
</tr>
<tr>
<td>Sensing Geometry</td>
<td>Shear</td>
</tr>
<tr>
<td>Size</td>
<td>3.6 mm × 11.4 mm × 6.4 mm</td>
</tr>
<tr>
<td>Weight</td>
<td>0.5 gm</td>
</tr>
<tr>
<td>Mounting</td>
<td>Adhesive</td>
</tr>
</tbody>
</table>

The accelerometer calibrator listed in Table 3.4 is used to calibrate the accelerometers. The accelerometer is attached to the calibrator using a threaded adapter. This calibration is performed using PCB 394C06 hand held calibrator (Figure 3.6) connected to Pulse hardware. In order to have more precise experiments, the accelerometers have been calibrated before each set of experiments.

Figure 3.6. PCB 394C06 hand held calibrator
Table 3-4. Accelerometer Calibrator information [39]

<table>
<thead>
<tr>
<th>Model No.</th>
<th>394C06</th>
</tr>
</thead>
<tbody>
<tr>
<td>Performance</td>
<td>Units (SI)</td>
</tr>
<tr>
<td>Operating Frequency (± 1%)</td>
<td>159.2 Hz</td>
</tr>
<tr>
<td>Acceleration Output (± 3%)</td>
<td>9.81m/s² rms</td>
</tr>
<tr>
<td>Maximum Load</td>
<td>210 gm</td>
</tr>
<tr>
<td>Physical</td>
<td></td>
</tr>
<tr>
<td>Size (Diameter)</td>
<td>56 mm</td>
</tr>
<tr>
<td>Weight (with batteries)</td>
<td>900 gm</td>
</tr>
</tbody>
</table>

In order to process the signals generated by the hammer and the accelerometer we need to connect them to a computer. This is accomplished by using a Pulse Data Acquisition (DAQ) Hardware made by Brüel & Kjær [40] (Figure 3.7). The Pulse Hardware accompanied by the Pulse software installed on a Laptop is a portable data processing system suitable for vibration testing.

![Figure 3.7. A six FFT input channel Pulse data acquisition hardware](image)

The Pulse software is used for analyzing the data. Fast Fourier Transform Analyzer (FFT) in Pulse software is used to perform modal analysis on structures. Time
Capture Analyzer is another package in Pulse software, which enables us to obtain the time history response of structures to shock and impacts.

Table 3-5. Pulse data acquisition hardware information [40]

<table>
<thead>
<tr>
<th>Model No.</th>
<th>Portable PULSE - 3560C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dynamic Range (FFT)</td>
<td>160 dB</td>
</tr>
<tr>
<td>Real-time Rate</td>
<td>25.6 kHz</td>
</tr>
<tr>
<td>I/O (FFT &amp; Time Capture)</td>
<td>6 channels</td>
</tr>
</tbody>
</table>

3.3.2 Test Setup and Procedure

The whole test setup includes an A-frame structure to support a hanging mass, a known cylindrical mass, accelerometer, impulse hammer, Pulse Hardware and a laptop computer with Pulse software (Figure 3.8). The A-frame supports one of the masses by a cable and an accelerometer is then mounted to one end of the mass. The accelerometer is attached to the data acquisition hardware, which provides power for internal circuitry and links the accelerometer to the computer. The impulse hammer is connected to the computer in the same manner as the accelerometer. Performing the experiment requires connecting the accelerometer and hammer to the Pulse hardware, and then connect the hardware units to the computer. There is a routine setup procedure for Pulse Lab Software. This includes setting up FFT and Time analyzer. Next, we impact the hanging mass with the impact hammer. A drop of super glue has been applied to stick accelerometer on the cylinder body. After mounting each accelerometer on the cylinder, the cylinder has been hit by the hammer three times, and three different data has been collected to insure the repeatability of the experiment for each accelerometer. Mounting
accelerometer on structures with super glue is a reliable technique for impact testing up to 10,000 Hz frequency range [39].

3.3.3 Experimental Results

The experimental results include two parts: Modal analysis and Time history response. Fast Fourier Transform (FFT) analyzer is used for the modal analysis of the structure. Time capture analyzer is used for time response of the structure. The measurement units for the accelerometer and hammers are m/s² and N respectively.

3.3.3.1 Modal Analysis

The peaks on FFT analyzer show the natural frequency of the structure. Figure 3.9 shows the FFT plot of the round bar. The peak corresponding to the 8th mode is not very
obvious. This peak belongs to the torsional natural frequency. It is very hard to excite this mode with impact to the solid bar. Table 3-6 presents the natural frequencies of the solid bar.

![Experimental FFT plot of the round bar](image)

**Figure 3.9. Experimental FFT plot of the round bar**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Experimental Natural Frequency (KHz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>4.3</td>
</tr>
<tr>
<td>8</td>
<td>8.4</td>
</tr>
<tr>
<td>9</td>
<td>10.4</td>
</tr>
<tr>
<td>10</td>
<td>13.4</td>
</tr>
</tbody>
</table>

**Table 3-6. Experimental natural frequencies of the round bar**

3.3.3.2 Time History

The acceleration and force in the time domain should have a sharp peak at the maximum impact force and quickly damp out to zero Figure 3.10 and Figure 3.11. If the signal has multiple peaks the test can be disregarded. The frequency range of the load cell
within the impulse hammer is 2500 (Hz). This force will be used for finite element
analysis later on in this chapter, so the force data has been filtered at 2500 Hz (Figure
3.10 and Figure 3.11). The time history response has been filtered at 10,000 Hz because
the accelerometer frequency range is 1.0 to 10,000 Hz. The sampling rate for data
presented in Figure 3.10 and Figure 3.11 rate is 65536 Hz.

![Graph showing force vs. time](image)

Figure 3.10. Applied force measured by small impulse hammer to the round bar

3.3.3.3 Coherence

Coherence measures the similarity of vibration in two locations. The two
locations for the hammer calibration are the force transducer on the hammer and the
accelerometer on the opposite side of the impact surface on the mass. If the signals are
correlated the coherence function will be one (Figure 3.12). However, if the signals do
not have a coherence of one the test is discarded and retested. If the coherence function is not one, possible problems could be power supplies turned on, a glancing blow with the hammer, bad wire connections, or low batteries in the power supplies. These should all be checked before continuing the tests.

Figure 3.11. Experimentally measured acceleration of the round bar in the time domain

Figure 3.12. Coherence plot
3.4 Natural Modes of Vibration

It is possible to treat certain systems more rigorously, without discretization of the analytical model. In this section we will analyze beams in which mass and deformation properties are continuously distributed. The best examples of these structures are bars, shafts and beams.

3.4.1 Longitudinal Vibration of a Bar or Rod

The differential equation for longitudinal motion of a thin rod can be written as follows [41]:

\[
\frac{\partial}{\partial x} \left[ EA(x) \frac{\partial u(x,t)}{\partial x} \right] = m(x) \frac{\partial^2 u(x,t)}{\partial t^2}
\]

which must be satisfied over the domain 0<x<L. In addition, \( u \) must be such that at the end point for a free-free beam we have

\[
EA(x) \frac{\partial u(x,t)}{\partial x} \bigg|_{x=0,L} = 0
\]

In this chapter \( u(x,t) \) denotes the longitudinal displacement and \( y(x,t) \) is a transverse displacement. Let us pursue further the case of free-free rod, for which the eigenvalue problem reduces to differential equation.

\[
- \frac{d}{dx} \left[ EA(x) \frac{dU(x)}{dx} \right] = \omega^2 m(x) U(x)
\]

The homogenous boundary condition

\[
EA(x) \frac{dU(x)}{dx} \bigg|_{x=0,L} = 0
\]

must be satisfied at the end point. For a uniform rod the eigenvalue problem reduces to the solution of the following differential equation.
\[
\frac{d^2U(x)}{dx^2} + \beta^2 U(x) = 0, \quad \beta^2 = \omega^2 \frac{m}{EA} \tag{5}
\]

The frequency equation can be obtained by applying the boundary conditions (Eq. 4) to the longitudinal vibration differential equation (Eq. 5).

\[
\sin \beta L = 0 \tag{6}
\]

Calculation of natural frequencies

\[
\sin \beta L = 0 \Rightarrow \beta L = r\pi \quad \text{Where } r=0,1,2,\ldots
\]

\[
\omega_r \sqrt{\frac{m}{EA}} = \frac{r\pi}{L} \Rightarrow (2\pi)f_r = \frac{r\pi}{L} \sqrt{\frac{EA}{m}}
\]

\[
f_r = \frac{r}{2L} \sqrt{\frac{EA}{m}} \tag{7}
\]

considering \( m \) is mass per unit length, \( m = \frac{M}{L} = \frac{\rho V}{L} = \frac{\rho AL}{L} = \rho A \)

\[
f_r = \frac{r}{2L} \sqrt{\frac{E}{\rho}} \tag{8}
\]

3.4.2 Torsional Vibration of a Shaft or Rod

If \( \theta(x,t) \) represent the angle of twist of a cross section at the point \( x \) and at time \( t \), the equation of motion in torsion is [41]

\[
\frac{\partial}{\partial x} \left[ GJ(x) \frac{\partial \theta(x,t)}{\partial x} \right] + m_r(x,t) = I(x) \frac{\partial^2 \theta(x,t)}{\partial t^2} \tag{9}
\]

\( G \) is the shear modulus and \( J(x) \) is a geometric property of the cross section, which in the case of a circular cross section is polar moment of inertia.

\[
J = \frac{\pi d^4}{32} \tag{10}
\]
The product $GJ(x)$ is called Torsional stiffness. $I(x)$ is the mass polar moment of inertia per unit length of bar and $m_T(x,t)$ is the external twisting moment per unit length of bar. In the case of free vibration, $m_T(x,t)=0$, and eq. 9 reduces to

$$\frac{\partial}{\partial x} \left[ GJ(x) \frac{\partial \theta(x,t)}{\partial x} \right] = I(x) \frac{\partial^2 \theta(x,t)}{\partial x^2}$$  \hfill (11)

The boundary conditions for a beam free at both ends are

$$GJ(x) \left. \frac{\partial \theta(x,t)}{\partial x} \right|_{x=0,L} = 0$$  \hfill (12)

Letting $\theta(x,t)=\Theta(x)f(t)$ and recalling $f(t)$ is harmonic, the eigenvalue problem reduces to the differential equation

$$- \frac{d}{dx} \left[ GJ(x) \frac{d \Theta(x,t)}{dx} \right] = \omega^2 I(x) \Theta(x)$$  \hfill (13)

and the boundary conditions are

$$GJ(x) \left. \frac{d \Theta(x)}{dx} \right|_{x=0,L} = 0$$  \hfill (14)

at the ends.

Let the bar be uniform and denote

$$\frac{\omega^2 I}{GJ} = \beta^2$$  \hfill (15)

so that eq. (13) reduces to

$$\frac{d^2 \Theta(x)}{dx^2} + \beta^2 \Theta(x) = 0$$  \hfill (16)

Applying boundary conditions to eq. 16 give us the frequency equation.

$$\sin \beta L = 0$$  \hfill (17)

So that the natural frequencies are
\sin \beta, L = 0 \Rightarrow \beta, L = r\pi \quad \text{Where } r=0,1,2,\ldots

\omega_r \sqrt{ \frac{I}{GJ} } = \frac{r\pi}{L} \Rightarrow (2\pi)f_r = \frac{r\pi}{L} \sqrt{ \frac{GJ}{I} }

f_r = \frac{r}{2L} \sqrt{ \frac{GJ}{I} } \quad (18)

for a cylindrical bar (rod), \(dJ = \frac{\pi d^4}{32}\)

\[ f_r = \frac{r}{2L} \sqrt{ \frac{G}{\rho} } \quad (19) \]

### 3.4.3 Flexural Vibration of Bars or Beams

In this section the transverse vibration of beams is studied. Timoshenko [42] has derived the general equation for transverse free vibrations of beam as follows:

\[ \frac{\partial^2}{\partial x^2} \left( EI(x) \frac{\partial^2 y}{\partial x^2} \right) dx = -\rho Adx \frac{\partial^2 y}{\partial t^2} \quad (20) \]

Equation 20 will reduce to the following equation for the solid bar [41], [42]:

\[ \frac{d^2}{dx^2} \left[ EI(x) \frac{\partial^2 y}{\partial x^2} \right] = \omega^2 m(x)y(x) \quad (21) \]

In the above equation, \(Y(x)\) satisfies all four boundary conditions, two at each end. At a free end the bending moment and the shearing force both vanish, and we obtain

\[ EI(x) \frac{d^2 y(x)}{dx^2} \bigg|_{x=0, L} = 0 \quad (22) \]

\[ \frac{d}{dx} \left[ EI(x) \frac{d^2 y(x)}{dx^2} \right] \bigg|_{x=0, L} = 0 \quad (23) \]

In the particular case of a uniform beam, where the flexural rigidity \(EI\) does not vary with \(x\), equation 21 will become
\[ EI \frac{d^4 y}{dx^4} = \omega^2 m y(x) \]  
(24)

or

\[ \frac{d^4 y(x)}{dx^4} - \beta^4 y(x) = 0 \]  
(25)

where

\[ \beta^4 = \frac{\omega^2 m}{EI} \quad \text{or} \quad \beta^2 = \omega \sqrt{\frac{m}{EI}} \]  
(26)

At the free end, in the case of uniform bar, the boundary conditions are

\[ EI(x) \frac{d^2 y(x)}{dx^2} \bigg|_{x=0,L} = 0 \]  
(27)

\[ \frac{d}{dx} \left[ EI(x) \frac{d^2 y(x)}{dx^2} \right] \bigg|_{x=0,L} = 0 \]  
(28)

All boundary conditions are natural. Consequently, by applying the boundary conditions into eq. 25, we obtain the frequency equation.

\[ \cos(\beta L) \cosh(\beta L) = 1 \]  
(29)

This equation can be solved numerically for the eigenvalues \( \beta_r \). The first few roots of eq. 29 are

\[ \beta_r L = 0, 4.7, 7.9, 11.0, 14.1 \]  
(30)

The \( \beta_r L = 0 \) is for the rigid body modes. The natural frequencies are

\[ \beta_r L = a_r \Rightarrow (\beta_r L)^2 = a_r^2 \Rightarrow \beta_r^2 = \frac{a_r^2}{L^2} \Rightarrow (2\pi f_r)^2 = \frac{a_r^2}{L^2} \sqrt{\frac{m}{EI}} = \frac{a_r^2}{L^2} \sqrt{\frac{m}{EI}} \]

\[ f_r = \frac{a_r^2}{(2\pi)^2 L^2} \sqrt{\frac{EI}{m}} \]  
(31)
In the case of solid rod, 

\[ I = \frac{\pi d^4}{64} = \frac{\pi d^2}{4} \frac{d^2}{16} = A \frac{d^2}{16} \]

and 

\[ m = \frac{M}{L} = \rho A \]

\[ f_r = \frac{\alpha_r^2 d}{(8\pi)^L^2} \sqrt{\frac{E}{\rho}} \]  

(32)

where \( \alpha_r^2 = 0, 22.4, 61.7, 120.9, 199.9. \)

3.4.4 Results

We obtain the following equations for longitudinal, torsional and flexural vibration of a rod, in the last three previous sections.

\[ f_r = \frac{r}{2L} \sqrt{\frac{E}{\rho}} \]

\[ f_r = \frac{r}{2L} \sqrt{\frac{G}{\rho}} \]

\[ f_r = \frac{\alpha_r^2 d}{(8\pi)^L^2} \sqrt{\frac{E}{\rho}} \]

The diameter of the bar is 0.0508 m and the length of it is 0.1087 m. The bar is made of steel with modulus of elasticity 207 GPa, shear modulus of 81 GPa and its density is 7850 kg/m\(^3\). The following table shows the natural frequencies of the round bar obtained analytically.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Nat. Freq (KHz)</th>
<th>Modal Characteristic</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>4.5</td>
<td>1(^{\text{st}}) Transverse Mode</td>
</tr>
<tr>
<td>8</td>
<td>8.2</td>
<td>Torsional</td>
</tr>
<tr>
<td>9</td>
<td>12.4</td>
<td>2(^{\text{nd}}) Transverse Mode</td>
</tr>
<tr>
<td>10</td>
<td>13.0</td>
<td>Longitudinal</td>
</tr>
</tbody>
</table>

Table 3-7. Analytical natural frequencies of the round bar

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Note that the first six modes are rigid body modes and their corresponding natural frequency is equal to zero.

3.5 Finite Element Analysis of a Simple Structure

The last part of this chapter is finite element analysis of the simple structure. A rigid round bar has been chosen as a sample for this purpose. The geometry of the solid cylinder is described in the beginning of this chapter. The finite element analysis consists of two parts: Modal analysis and time history response. The modeling has been done in the HyperMesh® and after applying the force, the model has been exported to LS-DYNA for solution. The result has been filtered in the LS-DYNA post processor at 10000 Hz corresponding to the experimental data.

3.5.1 Units

Units in LS-DYNA must be consistent. Table 3-8 shows a set of units employed in the modeling of the structure. These units are base on the SI unit system. There are two reasons for using SI units for the finite element modeling. The first reason is that implicit method in LS-DYNA Version 9.60 for modal analysis does not give the right result if geometry has been drawn in any other units except meters. We faced the same problem if we used millimeters or inches as the length units in modal analysis in Altair OptiStruct® version 7.0. The second reason is that Pulse, the experimental data analysis software, works with SI units. The units of results obtained and saved from this software are as shown in Table 3-8, and that is the set of units has been used in finite element analysis.

53
Table 3-8. Units used in the Modeling Analysis

<table>
<thead>
<tr>
<th>Units</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>meter</td>
</tr>
<tr>
<td>Time</td>
<td>second</td>
</tr>
<tr>
<td>Mass</td>
<td>kilogram</td>
</tr>
<tr>
<td>Force</td>
<td>newton</td>
</tr>
</tbody>
</table>

3.5.2 Material Properties

The round bar is made from cold rolled steel. Table 3-9 shows the material properties of the round bar [43].

Table 3-9. Material properties of cold roll steel

<table>
<thead>
<tr>
<th>Properties</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>$\rho$ 7850 (Kg/m$^3$)</td>
</tr>
<tr>
<td>Modulus of elasticity</td>
<td>E 207 (GPa)</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>$\nu$ 0.3</td>
</tr>
<tr>
<td>Yield stress</td>
<td>$\sigma_Y$ 200 (MPa)</td>
</tr>
</tbody>
</table>

A plastic kinematic constitutive model, type 3 material card in LS-DYNA [44], has been used to define the material for the structure. The following lines show the material properties from LS-DYNA input file (.k file).

```
*MAT_PLASTIC_KINEMATIC
$++t++1++t++2++t++3++t++4++t++5++t++6++t++7++t++
MID  RO   E  PR  SIGY
  1  7850.0  207.0 E09  0.3  200 E06
```

3.5.3 Boundary Conditions

No geometric boundary conditions were applied to the model. In the experiment, the cylinder hanged such a way that it can move freely in any direction. In the finite element model it has been assumed that there is no constraint applied to the model, i.e. free-free boundary conditions.
3.5.4 Applied Force

An impact force with a peak of 440 N in less than 0.0006 s has been applied to the model. The data for the force curve, shown in Figure 3.13, has been obtained from the experiment captured by Pulse Software. The experiment procedure is explained in the experiment section. For sake of simplicity, this force considered as nodal force and has been applied to the center node on the top plane of hat section (Figure 3.14).

![Figure 3.13. Force curve applied to the finite element models](image)

The following lines show the nodal force implementation from LS-DYNA input file (.k file).

*LOAD_NODE_SET
$+++$1++$2++$3++$4++$5++$6++$7
nsid dof lcid sf cid ml m2
1 2 3 -1.0

*SET_NODE_LIST
1
7756
*DEFINE_CURVE
3
0.0000000000, 0.00000
...
...
0.0019531250, 0.00000

Figure 3.14 shows the impact point and the accelerometer location on the FEA model of the round bar. The hit point is the center of one base of the cylinder and the accelerometer has been mounted on the center of other base. Figure 3.14 shows the location of the accelerometer and the impact point on the cylinder.

3.5.5 Solid Element Modeling

The geometry has been created in HyperMesh® and exported to LS-DYNA for solving. Figure 3.15 shows the meshed cylinder in the HyperMesh®. It is shown in the figure that all elements are hexahedral elements.
3.5.6 Finite Element Results

The error between the mass obtained by the finite element analysis and real mass of the structure should be small. This is one of the key parameters to verify the finite element model. The next step after checking the structure mass is investigating modal analysis and dynamic response of the structure obtained by finite element analysis.

3.5.6.1 Mass Verification

The small cylinder weighed 1.711 Kg. It has been measured by the scale with the accuracy of 0.1 gr or 0.00001 Kg. The mass of the bar calculated by finite element solver is 1.710 (Kg).

\[
Error = \frac{1.711 - 1.710}{1.711} \times 100 = 0.06\%
\]

The mass error is almost zero and that means the bar has been modeled properly for finite element analysis. Having small error in mass calculation also means that the material properties are correct. This mass verification means that the density and discretization are correct and the geometry is measured and modeled properly, but it does mean that the elastic properties are correct.
3.5.6.2 Modal analysis

The following table shows the modal analysis result of the solid cylinder. The cylinder is free and has not been constrained in any point, so the first six modes are rigid body modes and they are equal to zero.

<table>
<thead>
<tr>
<th>Mode</th>
<th>OptiStruct (KHz)</th>
<th>Modal Shape</th>
</tr>
</thead>
<tbody>
<tr>
<td>7,8</td>
<td>4.2, 4.2</td>
<td>Bending modes (y &amp; z-directions)</td>
</tr>
<tr>
<td>9</td>
<td>8.1</td>
<td>Torsional mode</td>
</tr>
<tr>
<td>10, 11</td>
<td>10.2, 10.2</td>
<td>Bending modes (y &amp; z-directions)</td>
</tr>
<tr>
<td>12</td>
<td>13.2</td>
<td>Longitudinal mode (x-direction)</td>
</tr>
</tbody>
</table>

3.5.6.3 Time History Response

Figure 3.16 shows the dynamic behavior of the structure. This response has been obtained by solving the finite element model in duration of 0.003 sec. This result has been filtered at 10,000 Hz because the accelerometer range is 1 to 10000 Hz. The sampling rate is 65536 Hz.

3.6 Comparison of Results

This chapter investigates a dynamic response of a solid cylinder hit by an impact hammer. The report tries to prove the validity of the finite element model by the experiment. This is the full dynamic validation process. Most literature considers modal analysis as the dynamic response verification of the simulation. The verification of dynamic models involves both modal analysis and time history response verification. In
this section we first compare the modal analysis obtained from experiment and analysis and then move the time history verification.

![Graph](image)

Figure 3.16. Time history response of the bar determined by finite element analysis

3.6.1 Modal Analysis Verification

Natural frequencies of a free-free bar can be obtained by experimental, analytical or finite element method. Table 3-11 compare the natural frequencies obtained from three different methods. The modal analysis shows that the there is a good agreement between three different methods of approaching to the problem except one case. The exceptional case is the first flexural natural frequency obtained by analytical method using Timoshenko Beam theory. The effects of cross-sectional dimensions on natural frequencies cannot be neglected for the cylinder in this study.
Table 3-11. Natural frequencies of the round bar

<table>
<thead>
<tr>
<th>Mode</th>
<th>Experimental (KHz)</th>
<th>Finite Element (KHz)</th>
<th>Analytical (KHz)</th>
<th>Modal Characteristic</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>4.3</td>
<td>4.2</td>
<td>4.5</td>
<td>Transverse</td>
</tr>
<tr>
<td>8</td>
<td>8.4</td>
<td>8.1</td>
<td>8.2</td>
<td>Torsional</td>
</tr>
<tr>
<td>9</td>
<td>10.4</td>
<td>10.2</td>
<td>12.4</td>
<td>Transverse</td>
</tr>
<tr>
<td>10</td>
<td>13.4</td>
<td>13.2</td>
<td>13.0</td>
<td>Longitudinal</td>
</tr>
</tbody>
</table>

3.6.2 Time History Verification

Dynamic response of the structure obtained both from experimental and finite element model have been shown previously in figures Figure 3.11 and Figure 3.16. Figure 3.17 shows these responses plotted together. This shows that there is a good agreement between experiment and finite element analysis. Based on the acceleration shown in Figure 3.17 the finite element method is an appropriate for simulation and prediction of dynamic response of the structures.

![Graph of Time History Response](image)

Figure 3.17. Time history response of the bar (Experiment vs. FEA)
Table 3-12 shows the quantitative comparison of experimental data and FEA of the solid cylinder. Figure 3.18 shows the Shock Response Spectrum (SRS) of the solid cylinder.

<table>
<thead>
<tr>
<th>Regular Method</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acceleration</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>0th</td>
<td>1st</td>
<td>2nd</td>
</tr>
<tr>
<td>4.1</td>
<td>42</td>
<td>3.3</td>
<td>1.4</td>
<td>0.2</td>
</tr>
</tbody>
</table>

Figure 3.18. SRS of experimental and FE analysis of the bar
CHAPTER 4

STRUCTURES WITHOUT JOINTS

4.1 Introduction

This chapter investigates the finite element analysis of structures without any joints. The experimental data is employed to verify the structural response, which is predicted by the finite element analysis. Mesh refinement can lead to better results from finite element analysis. For very simple structure there is almost no other parameter to play with for having a better result. Mesh sensitivity is often ignored in finite element analysis. Results of the finite element model usually change with the mesh refinement, so it is important to study the effects of this parameter. A rectangular flat plate and a hat section are the two simple structures chosen for the study. In the first part of this chapter, the sensitivity of the response of a flat plate to the mesh refinement has been investigated. The second part of this chapter looks at the shock response of a hat section and finite element simulation of this event. Experimental procedure has been employed to benchmark and verify these studies.

4.2 Quarter Inch Steel Plate

The first part of this chapter investigates the shock response of a plate. The shape of the plate is rectangular and it is flat. This is most simple two-dimensional structure that can be chosen for the experiment and analysis.
4.2.1 Geometric Configuration

The shape of the plate is rectangular with sharp corners. The length of the plate is 0.965 m, and the width of it is 0.0337 m and the average thickness of the plate is 0.00635 m (quarter inch). The flat plate with its dimensions is shown in Figure 4.1. The plate mass is 8.2 Kg.

![Dimensions of the flat plate in m](image)

Figure 4.1. Dimensions of the flat plate in m

4.2.2 Units and Material Properties

The units are the same in experiment and FE analysis. We have not changed the units in modeling and analysis in order to be consistent with the experiment and avoid any confusion in result comparison. Since the experimental data acquisition board works on SI system, meter, kilogram and Newton is used for the analysis as well. Table 4-1 shows the units employed in the plate analysis and experimental study.

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimensions</td>
<td>meter (m)</td>
</tr>
<tr>
<td>Mass</td>
<td>kilogram (Kg)</td>
</tr>
<tr>
<td>Time</td>
<td>second (Sec)</td>
</tr>
<tr>
<td>Force</td>
<td>Newton (N)</td>
</tr>
</tbody>
</table>

Table 4-1. Units on the experiment and analysis

63

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The plate is made from hot rolled steel, ASTM-A36. Table 4-2 shows the mechanical properties of the steel used for the structure [43].

<table>
<thead>
<tr>
<th>Properties</th>
<th>Values</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>ρ</td>
<td>7850</td>
</tr>
<tr>
<td>Modulus of elasticity</td>
<td>E</td>
<td>207</td>
</tr>
<tr>
<td>Modulus of rigidity</td>
<td>G</td>
<td>79.6</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>ν</td>
<td>0.3</td>
</tr>
<tr>
<td>Yield stress</td>
<td>σ_Y</td>
<td>250</td>
</tr>
</tbody>
</table>

4.2.3 Experiment on the flat plate

The test setup includes an A-frame structure to support a hanging mass, flat plate configuration, accelerometers, impulse hammer, and a laptop computer (Figure 4.2). The A-frame supports the plate by two steel wires. The accelerometer is mounted on the center of the plate. The accelerometers and impact hammer are attached to the front box, which includes the hardware of data acquisition system. Pulse is the software, which talks to the front box for obtaining the data. Pulse uses SI units, so the units for the accelerometer and hammers are m/s² and N respectively. Figure 4.3 shows the point of impact as well as the location of accelerometer on the plate. The impact point is where hammer hits the plate. It is on the vertical centerline of the plate and 0.07 m above the bottom edge of the plate. The accelerometer is located on the center point of the plate on the impact side. It is 0.152 m and 0.271 m away from the vertical and horizontal sides of plate, respectively.
Figure 4.2. Experimental setup of the flat plate

Figure 4.3. Impact point and accelerometer location on the plate
There is a load cell embedded inside the impact hammer. This sensor measures the force applied by the hammer to the structure. Figure 4.4 shows the force applied by the large hammer (PCB 086C20) to the flat plate. The force has peak amplitude of 2000 N and duration of $1.36 \times 10^{-3}$ sec. This force, which is measured experimentally, is an input to the finite element model of the flat plate.

![Applied Force to the Plate](image)

Figure 4.4. Applied force to the flat plate measured by the instrumented hammer (PCB 086C20)

The vibration of the plate is measured by the accelerometer mounted on the center of the plate. This accelerometer is able to pick up any vibrations below 10000 Hz, as it written in accelerometer data sheet [39]. The vibration of the plate is shown in Figure 4.5. The steady state behavior of the plate is not in the interests of this dissertation. The plot shown in Figure 4.5 is the transient vibration of the plate during a short period of time after the shock applied to the plate. It can be seen from the acceleration plot that there is no damping in the first few milliseconds. The peak acceleration is about 2022 m/s, which occurs at 1.93 ms after the excitation starts.
4.2.4 Finite Element Modeling

Finding the closest finite element model that represents the structure is one of the objectives of this chapter. The finite element models can be divided into two major categories based on their element configuration. Shell element model and solid element model are these two major categories. The mesh refinement investigation is performed on shell FE models by splitting each side of the element to two. This split gives us four times more element compared to the previous model. The mesh refinement has been continued till it has a little effect on the structure's response. The number of elements along the thickness was increased to achieve a finer mesh on solid FE models. Of course, the number of elements along the length and width of the plate must be increased to avoid high aspect ratio on the mesh refinement procedure. The chart presented on Figure 4.6 shows the all the plate finite element models made from shell and solid elements.
There are four shell finite element models of plate. The models start with coarse mesh (160 elements) and goes up to the finest model with 10240 elements. There are six different solid element models. They start with having 1 element along thickness and go up to having 6 elements along thickness. The numbers 1T, 2T, ..., 6T under solid element models in stands for number of elements through thickness. Total of ten finite element models were made to investigation the effect of mesh refinement on the response of the structure.
4.2.4.1 Shell Element Modeling

Figure 4.7 and Figure 4.8 show the meshed model of the plate generated with Altair HyperMesh®. A 0.1 m by 0.86 m section of each shell FE model is presented on Figure 4.8 to compare the mesh size in all shell FE models. All elements in these models are 2-D elements. The first shell model has a very coarse mesh with only 160 elements. Each side of the shell element has been divided to two to obtain the new model with finer mesh. The next shell model has four times more elements than the pervious one. The second shell FE model of the plate has 640 elements. The mesh refinement procedure is continued till the effect of it is negligible on the structure response. Later in this chapter we will see that there is no significant change on the structure response from 2560 elements to 10240 elements.

Figure 4.7. Shell element model of the plate
Table 4-3 contains the mesh properties of each shell element model. It includes the total number of elements and element aspect ratio of each model. The model with the finest mesh has 64 times more elements than the one with coarsest mesh.

Table 4-3. Mesh properties of shell element models of the plate

<table>
<thead>
<tr>
<th>Number of elements along the width</th>
<th>Number of elements along the length</th>
<th>Total number of elements</th>
<th>Maximum aspect ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>16</td>
<td>160</td>
<td>1.11</td>
</tr>
<tr>
<td>20</td>
<td>32</td>
<td>640</td>
<td>1.11</td>
</tr>
<tr>
<td>40</td>
<td>64</td>
<td>2560</td>
<td>1.11</td>
</tr>
<tr>
<td>80</td>
<td>128</td>
<td>10240</td>
<td>1.11</td>
</tr>
</tbody>
</table>

4.2.4.2 Solid Element Modeling

Figure 4.9 shows the meshed model of the plate generated with HyperMesh® using solid elements. Six different models have been developed using the same geometry. All elements in these models are 3-D elements. The differences between models are the number of elements through the thickness. The model with the coarsest mesh has only 1 element through thickness. To refine the mesh in this model we have put two elements...
along the thickness and increase the number of elements along the length and width of the plate, so the element aspect ratio stays almost the same as the previous model. This procedure was continued till 6 elements through thickness. The detail view of the plate’s corner on each model is shown in Figure 4.9. The detail view is a cubic with the dimension of 0.04×0.04×0.00635 m, which is taken apart from the plate model as a sample. Having the exact same size on each detail view helps to visualize and compare the models together and see the changes in element size. Table 4-4 contains the mesh properties of each model. It includes the total number of elements and element aspect ratio of each model. The model with the finest mesh has 250 times more elements than the one with coarsest mesh.

Figure 4.9. Solid element model of the plate
Table 4-4. Mesh properties of solid element models of the plate

<table>
<thead>
<tr>
<th>Number of elements along the thickness</th>
<th>Number of elements along the width</th>
<th>Number of elements along the length</th>
<th>Total number of elements</th>
<th>Maximum aspect ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>30</td>
<td>600</td>
<td>2.85</td>
</tr>
<tr>
<td>2</td>
<td>40</td>
<td>60</td>
<td>4800</td>
<td>2.85</td>
</tr>
<tr>
<td>3</td>
<td>50</td>
<td>100</td>
<td>15000</td>
<td>2.87</td>
</tr>
<tr>
<td>4</td>
<td>80</td>
<td>120</td>
<td>38400</td>
<td>2.85</td>
</tr>
<tr>
<td>5</td>
<td>84</td>
<td>160</td>
<td>67200</td>
<td>2.85</td>
</tr>
<tr>
<td>6</td>
<td>100</td>
<td>200</td>
<td>120000</td>
<td>2.87</td>
</tr>
</tbody>
</table>

4.2.5 Results of the Plate Experiment and FEM Analysis

The result of transient analysis verification can be divided into two parts: modal analysis and time history comparison. When it comes to the dynamic analysis, most literature stops at the modal analysis, and never get involved in the time history comparison. This dissertation considers a full-scale dynamic analysis, which is the evaluation of both modal analysis and time history.

4.2.5.1 Modal Analysis of the Plate

The first step to verification of experiment and modeling result is to compare the natural frequencies from model and experiment. The experimental natural frequencies obtained by observing the FFT plots. Frequencies corresponding to peaks, or spikes, on these plots are natural frequencies. Modal analysis has been performed in the shell and solid element model of the plate using OptiStruct software. The results are shown in Table 4-5 with the calculated error between experiment and analysis. The errors between the numerical and experimental natural frequencies are below 2 percent. The mode
number in Table 4-5 starts from 7 because the first six modes are rigid body modes and their natural frequencies are zero.

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Experiment (Hz)</th>
<th>Shell Model (Hz)</th>
<th>Solid Model (Hz)</th>
<th>% Error in shell model</th>
<th>% Error in solid model</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>115.09</td>
<td>114.5</td>
<td>114.4</td>
<td>0.5</td>
<td>0.6</td>
</tr>
<tr>
<td>8</td>
<td>128.19</td>
<td>126.2</td>
<td>126.4</td>
<td>1.6</td>
<td>1.4</td>
</tr>
<tr>
<td>9</td>
<td>286.12</td>
<td>282.1</td>
<td>282.4</td>
<td>1.4</td>
<td>1.3</td>
</tr>
<tr>
<td>10</td>
<td>320.24</td>
<td>316.5</td>
<td>316.4</td>
<td>1.2</td>
<td>1.2</td>
</tr>
<tr>
<td>11</td>
<td>377.02</td>
<td>374.4</td>
<td>374.0</td>
<td>0.7</td>
<td>0.8</td>
</tr>
<tr>
<td>12</td>
<td>462.65</td>
<td>457.2</td>
<td>457.1</td>
<td>1.2</td>
<td>1.2</td>
</tr>
<tr>
<td>13</td>
<td>506.46</td>
<td>501.0</td>
<td>501.4</td>
<td>1.1</td>
<td>1.0</td>
</tr>
<tr>
<td>14</td>
<td>631.53</td>
<td>619.0</td>
<td>618.9</td>
<td>2.0</td>
<td>2.0</td>
</tr>
</tbody>
</table>

The extracted mode shapes are shown in Figure 4.10. One can see that there are certain regions where the hat section deforms periodically when the mode shape is animated, particularly the corners of the horizontal flat plates and the edges of the vertical sides.

4.2.5.2 Time History Response of the Plate

The next step to verification of experiment and modeling result is to compare the accelerations in time domain. Four shell and six solid finite element models were made to study the impact analysis of the plate. Figure 4.11 shows the finite element analysis comparisons using shell elements with experimental results (Acceleration Vs Time). Most of these time history plots looks the same if they are compared by eye. The error analysis presented in chapter two is employed to capture the difference between time domain plots.
Figure 4.10. Mode shapes of the plate obtained by the finite elements modal analysis
Table 4-6 shows the comparison of the shell element models with experimental data. The dissimilarity factor was obtained by comparing time domain response of the plate with the experimental result. There is a major error reduction with increasing the number of elements from 160 to 640. Models with 640, 2560 and 10240 show a very small error values.

Considering the long simulation run for models with high number of elements, the model with 640 elements should be accurate enough for most of engineering applications. Having said that none of these models gives us an identical match with experimental
acceleration. According to the error analysis the shell model with 2560 elements is the closest match we can get from the flat plate shell models. The sampling rate for both experimental and FEA data is $1.5259 \times 10^5$ s. Figure 4.12 shows the shock response spectrum (SRS) response of the model with experimental data in the same plot.

Table 4-6. Comparison of the plate experiment and FE shell model

<table>
<thead>
<tr>
<th>Total Number of Elements</th>
<th>Regular Method</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acceleration</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>0th</td>
<td>1st</td>
<td>2nd</td>
</tr>
<tr>
<td>160</td>
<td>12.1</td>
<td>53.8</td>
<td>12.2</td>
<td>13.3</td>
<td>14.8</td>
</tr>
<tr>
<td>640</td>
<td>4.81</td>
<td>46.9</td>
<td>4.9</td>
<td>6.3</td>
<td>7.0</td>
</tr>
<tr>
<td>2560</td>
<td>2.81</td>
<td>38.9</td>
<td>2.7</td>
<td>2.1</td>
<td>1.5</td>
</tr>
<tr>
<td>10240</td>
<td>3.64</td>
<td>45.8</td>
<td>3.6</td>
<td>3.0</td>
<td>2.4</td>
</tr>
</tbody>
</table>

Figure 4.13 shows the acceleration response of the solid element models with experimental data in the same plot. The first interesting point is the high error value for the model with 1 element along thickness. Looking at the graph leads to the basic point the results from the model using only one solid element along the thickness are not acceptable. The visual inspection of the time histories plotted in Figure 4.13 shows that the responses from finite element models with 3, 4, 5 and 6 elements along the thickness are similar to the experimental data.
Figure 4.12. Shock response spectrum (SRS) of the plate: (shell element model)
Figure 4.13. Time history response of the plate: Experimental and FE analysis (solid element model)
Table 4-7 shows the errors of response of the plate solid element models. The errors are small for the models with 38400, 67200 and 120000 elements.

Table 4-7. Comparison of the plate experiment and FE solid model

<table>
<thead>
<tr>
<th>No of Elements Along Thickness</th>
<th>Total Number of Elements</th>
<th>Regular Method</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acceleration</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0th 1st 2nd</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>600</td>
<td>98.4</td>
<td>97.9</td>
<td>98.5 98.6 98.6</td>
<td>98.8</td>
<td>98.6</td>
</tr>
<tr>
<td>2</td>
<td>4800</td>
<td>56.7</td>
<td>60.1</td>
<td>57.7 61.6 63.9</td>
<td>61.6</td>
<td>60.8</td>
</tr>
<tr>
<td>3</td>
<td>15000</td>
<td>8.4</td>
<td>46.6</td>
<td>9.6 12.8 13.9</td>
<td>14.9</td>
<td>12.6</td>
</tr>
<tr>
<td>4</td>
<td>38400</td>
<td>3.7</td>
<td>34.2</td>
<td>4.5 7.9 9.7</td>
<td>9.8</td>
<td>7.7</td>
</tr>
<tr>
<td>5</td>
<td>67200</td>
<td>1.4</td>
<td>30.9</td>
<td>1.9 4.4 6.1</td>
<td>4.9</td>
<td>4.1</td>
</tr>
<tr>
<td>6</td>
<td>120000</td>
<td>0.5</td>
<td>30.5</td>
<td>0.1 1.5 2.8</td>
<td>2.1</td>
<td>1.5</td>
</tr>
</tbody>
</table>

Figure 4.14 (a) shows the difference between moment method values versus number of elements in the plate shell element models and experimental data. Similarly, Figure 4.14 (b) shows the difference between moment method values versus number of elements in the plate solid element models and experimental data. The decreasing trend of dissimilarity factor versus number of elements is shown in Figure 4.14 (c) and (d). The values presented in Figure 4.14 (a) and (c) obtained by applying moment method on the experiment and analysis data and finding the difference between the experimental and FEA zeroth and first moment values. The curve in Figure 4.14 (c) were generated by calculating the dissimilarity factor between plate shell finite element models and experimental data. The dissimilarity factor between plate solid finite element models and experimental data plotted in Figure 4.14 (d). The complete definition and characteristics of these moment method and dissimilarity factor were explained in chapter two.
Values of dissimilarity factor in Table 4-7 and graphs plotted in Figure 4.14 (d) shows that the accuracy of finite element model made with solid elements keep increasing with mesh refinement. Figure 4.15 shows experimental acceleration and the finite element acceleration obtained from a model with six solid elements along the thickness. According to the error analysis and dissimilarity factor this is the closest match we can get from the flat plate shell and solid models. Figure 4.16 shows the shock response spectrum (SRS) response of the model with experimental data in the same plot.
Figure 4.15. Time history response of the plate: Experimental and FE analysis (6 elements along the thickness)
Figure 4.16. Shock response spectrum (SRS) of the plate: (solid element model)
4.3 Hat Section

This section of chapter 4 investigates the shock response of a hat section. The hat section has three sides and two flanges. Due to its nonlinear geometry, hat section is not a simple structure, like the flat plate.

4.3.1 Geometric Configuration

Hat section is made from 6.35 mm steel plate as shown in Figure 4.17. The three sides of the hat section are square with the dimensions of 0.3×0.3 m. The flanges are rectangular with the dimensions of 0.125×0.3 m. The hat section is made of a flat plate, which is bent on different locations to form a hat section. The hat section structure is one piece of steel plate, and no welding or any other kind of mechanical joint connection is applied to hold the sides together.

Figure 4.17. Hat section configuration and dimensions are in m
4.3.2 Units and Material Properties

The SI units is used both in experimental and finite element analysis of hat section (Table 4-1). The hat section is made of hot roll steel, ASTM-A36. The mechanical properties of this material can be found in Table 4-2.

4.3.3 Experiment

The test setup includes an A-frame structure to support a hanging mass, hat section configuration, accelerometers, impulse hammer, and a laptop computer (Figure 4.18).

![Figure 4.18. Experimental setup for hat section](image)

The A-frame supports the hat section by two steel wires. The accelerometer is mounted on center of the side of the hat section (Figure 4.18). The detailed experimental procedure were explained in chapter 3. Figure 4.19 show a close up photo of the hat section with impact hammer and accelerometer mounted on it.
Figure 4.19. Hat section, accelerometers and large impact hammer

Figure 4.20 shows the point of impact as well as the location of accelerometer on the hat section. The impact point is on the center of the side of the hat section. The accelerometer is located on the center point of the other side of the hat section. As it can be seen in Figure 4.19 and Figure 4.20, the impact point is on the right side and the accelerometer is located on the left side of the hat section.

Figure 4.20. Impact point and accelerometer location on the hat section
Figure 4.21 shows the force applied by the hammer to the hat section. This force, which is measured experimentally, is the input to the finite element model of the hat section. The force has peak amplitude of 4000 N and duration of $1.6 \times 10^{-3}$ sec.

![Applied Force to the Hat Section](image)

Figure 4.21. Applied force to the hat section by large instrumented hammer

Since the duration of force duration is short, this impact can be categorized as a shock. The structure starts vibrating due to this shock exerted by the hammer. The acceleration of the hat section is shown in Figure 4.22. The accelerometer is located about 0.6 m from the point of impact. This distance is not a straight line. This is a hat shape path from the impact point to accelerometer. This means the shock will travel for 0.6 m before it reaches the accelerometer.
Looking at the acceleration plot shows that there is no damping in the transient response of the hat section during the first 10 msec. The peak of absolute acceleration is about 3200 m/s², which occurs at 8.41 ms after the excitation starts.

4.3.4 Finite Element Modeling

The chart presented in Figure 4.23 shows the different finite element models of the hat section. Ten finite element models of hat section were made, four using the shell elements and six using solid element. The modeling starts with coarse mesh and continues to very fine mesh.
4.3.4.1 Shell Element Modeling

Figure 4.24 shows the meshed model of the quarter inch hat section generated with Altair HyperMesh®. Figure 4.25 shows a 0.091 m by 0.084 m magnified section of the shell element model in detail for better visualization. All elements in these models are 2-D elements. Each side of the shell element has been divided to two to obtain the new model with finer mesh. This procedure was continued till the effect of mesh refinement is negligible on the structure response. Table 4-8 includes the mesh properties of all the shell finite element models made for the hat section. The geometry of the hat section is
not as simple as the plate, so it is not possible to keep the aspect ratio same on all the shell element models. The shell model with the coarsest mesh has 420 elements, which is reasonable, considering overall geometry of the hat section. The finest meshed model has 26800 elements, which is very high for this structure.

Figure 4.24. Shell element model of a hat section configuration

Figure 4.25. Close up of part of shell element model of steel hat section
Table 4-8. Mesh properties of shell element models of the hat section

<table>
<thead>
<tr>
<th>Number of elements along the width</th>
<th>Total number of elements</th>
<th>Maximum aspect ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>420</td>
<td>2.38</td>
</tr>
<tr>
<td>20</td>
<td>1680</td>
<td>2.23</td>
</tr>
<tr>
<td>40</td>
<td>6720</td>
<td>2.14</td>
</tr>
<tr>
<td>80</td>
<td>26880</td>
<td>2.14</td>
</tr>
</tbody>
</table>

Note: The maximum aspect ratio in Hat Section shell model cannot be constant, because of the accuracy on bended lines for shell element model with fewer elements.

4.3.4.2 Solid Element Modeling

Figure 4.26 shows the meshed model of the quarter inch hat section generated with HyperMesh®. A small part of each hat section solid element model is magnified for comparison. These views are a cubic piece cut from the corner of each hat section model. The dimension of the sample cubic is 0.0262×0.00635×0.02432 m. The detailed views belong to solid element model with 1 through 6 elements along the thickness. Six different models have been developed using the same geometry. The differences between models are the number of elements through the thickness. All elements in these models are 3-D elements. Table 4-9 includes the mesh properties of all the solid finite element models made for the hat section.
Figure 4.26. Solid element model of a hat section configuration

Table 4-9. Mesh properties of solid element models of the hat section

<table>
<thead>
<tr>
<th>Number of elements along the thickness</th>
<th>Number of elements along the width</th>
<th>Total number of elements</th>
<th>Maximum aspect ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>1176</td>
<td>3.69</td>
</tr>
<tr>
<td>2</td>
<td>35</td>
<td>10290</td>
<td>3.11</td>
</tr>
<tr>
<td>3</td>
<td>50</td>
<td>30900</td>
<td>3.18</td>
</tr>
<tr>
<td>4</td>
<td>60</td>
<td>62400</td>
<td>3.63</td>
</tr>
<tr>
<td>5</td>
<td>80</td>
<td>104000</td>
<td>3.69</td>
</tr>
<tr>
<td>6</td>
<td>100</td>
<td>206400</td>
<td>3.32</td>
</tr>
</tbody>
</table>
4.3.5 Results of the Hat Section

This section compares the results of the finite element analysis with the experimental data. The modal analysis is the first part of the result investigation. The time history comparison is the second part of result interpretation.

4.3.5.1 Hat Section Modal Analysis

The first step to verification of experiment and modeling results is to compare the natural frequencies from model and experiment. The experimental natural frequencies are obtained by observing the Fast Fourier Transform (FFT) plots. Frequencies corresponding to peaks, or spikes, on FFT plots are natural frequencies. The numerical natural frequencies obtained by performing a modal analysis on a hat section finite element model using OptiStruct® software. Exact shell and solid elements models of the hat section are created in HyperMesh®, which are shown in Figure 4.24 and Figure 4.26. Modal analysis is performed on these models.

Table 4-10 includes natural frequency of the hat section obtained numerically and experimentally. The first six modes are the rigid body modes; hence their corresponding natural frequencies are zero. The results are shown in with the calculated error between experiment and finite element model. In solid FE model error for all modes are very small. They are below 3%. The errors in Shell FE model are higher than solid FE model. The error in predicting mode number 9 jumps up to 10 percent. The result presented in Table 4-10 recommends using solid FE model for modal analysis purposes.

92

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Table 4-10. Modal analysis of hat section

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Experiment (Hz)</th>
<th>Shell Model (Hz)</th>
<th>Solid Model (Hz)</th>
<th>% Error in shell model</th>
<th>% Error in solid model</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>22.25</td>
<td>22.8</td>
<td>22.7</td>
<td>2.5</td>
<td>2.0</td>
</tr>
<tr>
<td>8</td>
<td>47.84</td>
<td>45.75</td>
<td>48.3</td>
<td>4.4</td>
<td>1.0</td>
</tr>
<tr>
<td>9</td>
<td>60.75</td>
<td>54.63</td>
<td>62.2</td>
<td>10.1</td>
<td>2.4</td>
</tr>
<tr>
<td>10</td>
<td>83.52</td>
<td>85.38</td>
<td>84.2</td>
<td>2.2</td>
<td>0.8</td>
</tr>
<tr>
<td>11</td>
<td>109.14</td>
<td>113.10</td>
<td>110.9</td>
<td>3.6</td>
<td>1.6</td>
</tr>
<tr>
<td>12</td>
<td>158.75</td>
<td>159.50</td>
<td>160.6</td>
<td>0.5</td>
<td>1.2</td>
</tr>
<tr>
<td>13</td>
<td>184.05</td>
<td>185.06</td>
<td>186.9</td>
<td>0.5</td>
<td>1.5</td>
</tr>
<tr>
<td>14</td>
<td>259.84</td>
<td>263.04</td>
<td>263.7</td>
<td>1.2</td>
<td>1.5</td>
</tr>
</tbody>
</table>

The extracted mode shapes are shown in Figure 4.27. One can see that there are certain regions where the hat section deforms periodically when the mode shape is animated, particularly the corners of the horizontal flat plates and the edges of the vertical sides.

4.3.5.2 Time History Response of the Hat Section

The next step to verification of experiment and modeling result is to compare the accelerations in time domain. Four shell and six solid finite element models made to study the impact analysis of the hat section. Figure 4.28 shows the finite element analysis comparisons using shell elements with experimental results. Most of these time history plots looks the same if they are compared with eyes. The error analysis presented in chapter two is employed to capture the difference between time domain plots. Table 4-11 shows the errors of response of the shell element models. These errors were obtained by comparing time domain response of the hat section with the experimental result. There is a major error reduction with increasing the number of elements from 420 to 1680, but refining mesh more than 1680 elements does not change the error. Figure 4.29 shows the finite element analysis comparisons using shell elements with experimental results (SRS).
Figure 4.27. Mode shapes of the hat section obtained by the finite elements modal analysis
Figure 4.28. Time history response of the hat section: Experimental and FE analysis (Shell element)

Table 4-11. Error analysis of the time domain response of the hat section shell model

<table>
<thead>
<tr>
<th>Total Number of Elements</th>
<th>Regular Method</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acceleration</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>0th</td>
<td>1st</td>
<td>2nd</td>
</tr>
<tr>
<td>420</td>
<td>12.9</td>
<td>67</td>
<td>13.8</td>
<td>11.3</td>
<td>7.4</td>
</tr>
<tr>
<td>1680</td>
<td>6.6</td>
<td>62</td>
<td>3.8</td>
<td>4.9</td>
<td>6.5</td>
</tr>
<tr>
<td>6720</td>
<td>11.4</td>
<td>59</td>
<td>8.2</td>
<td>8.7</td>
<td>10.9</td>
</tr>
<tr>
<td>26880</td>
<td>9.9</td>
<td>61</td>
<td>8.9</td>
<td>10.4</td>
<td>13.1</td>
</tr>
</tbody>
</table>
Figure 4.29. Shock response spectrum (SRS) of the hat section: (shell element model)

Figure 4.30 shows the acceleration response of the hat section solid model with experimental data in the same plot. The first interesting point is the high error value for the model with 1 element along thickness. Looking at the graph leads to the basic point the results from the model using only one solid element along the thickness are not acceptable. More conclusions can be withdrawn by performing quantitative comparison of time histories of hat section solid models and experimental data.
Figure 4.30. Time history response of the hat section: Experimental and FE analysis (Solid element)
Table 4-12 shows the quantitative comparison of the solid element models and experimental data. There is a major error reduction with increasing the number of elements from 10290 to 30900. The 10290 and 30900 elements are corresponding to models with two and three elements along the thickness respectively. This drastic reduction in dissimilarity factor is more obvious in Figure 4.31 (d), which show the dissimilarity factor versus number of elements in the solid models.

Table 4-12. Error analysis of the time domain response of the hat section solid model

<table>
<thead>
<tr>
<th>No of Elements Along Thickness</th>
<th>Total Number of Elements</th>
<th>Regular Method</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acceleration</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>0th</td>
<td>1st</td>
<td>2nd</td>
<td></td>
</tr>
<tr>
<td>1 1176</td>
<td>100</td>
<td></td>
<td>100</td>
<td>99.9</td>
<td>99.9</td>
<td>100</td>
</tr>
<tr>
<td>2 10290</td>
<td>35.6</td>
<td>54</td>
<td>36.1</td>
<td>33.3</td>
<td>30.2</td>
<td>43.34</td>
</tr>
<tr>
<td>3 30900</td>
<td>17.1</td>
<td>62</td>
<td>17.8</td>
<td>14.6</td>
<td>10.5</td>
<td>17.57</td>
</tr>
<tr>
<td>4 62400</td>
<td>12.7</td>
<td>52</td>
<td>13.3</td>
<td>12.6</td>
<td>11.0</td>
<td>7.8</td>
</tr>
<tr>
<td>5 104000</td>
<td>7.5</td>
<td>49</td>
<td>7.6</td>
<td>9.0</td>
<td>9.4</td>
<td>0.51</td>
</tr>
<tr>
<td>6 206400</td>
<td>0.4</td>
<td>49</td>
<td>0.5</td>
<td>2.9</td>
<td>4.1</td>
<td>15.7</td>
</tr>
</tbody>
</table>

Figure 4.31 (a) shows the difference between moment method values versus number of elements in the hat section shell element models and experimental data. Similarly, Figure 4.31 (b) shows the difference between moment method values versus number of elements in the hat section solid element models and experimental data. The decreasing trend of dissimilarity factor versus number of elements is shown in Figure 4.31 (d). The values presented in Figure 4.31 (a) and (c) obtained by applying moment method on the experiment and analysis data and finding the difference between the experimental and FEA zeroth and first moment values. The curve in Figure 4.31 were generated by calculating the dissimilarity factor between plate shell finite element models.

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
and experimental data. The dissimilarity factor between plate solid finite element models and experimental data plotted in Figure 4.31 (d). The complete definition and characteristics of these moment method and dissimilarity factor were explained in chapter two.

Figure 4.31. Difference in moment values and dissimilarity factors versus number of elements for shell and solid hat section models

Comparison of time histories presented in Table 4-12 and the graphs plotted in Figure 4.31 (b) and (d) shows that the accuracy of finite element model made with solid
elements keep increasing with mesh refinement. The higher the number of elements along the thickness, the better accuracy of the model’s response compare to the experimental result. However, the mesh refinement procedure cannot be continued due to the limitation of the computers. In fact, the double CPU with 4 GB RAM computer used for modeling, hanged many times during the modeling of the hat section with 6 elements through the thickness. Transient response of finite element model with six solid elements along thickness and experimental results are shown in Figure 4.32. According to the error analysis this is the closet match we can get from the hat section shell and solid models. Figure 4.33 shows the shock response spectrum (SRS) of the model with experimental data in the same plot.

Figure 4.32. Time history response of the hat section: Experimental and FE analysis (6 elements along thickness)
Figure 4.33. Shock response spectrum (SRS) of the hat section: (solid element model)
The study of plate and hat section finite element analysis proves that the models with high number of elements show a good accuracy. However, one might say that it is impossible to model the complicated mechanical parts with this high number of elements. The answer to this point is one of the purposes of this study. In the study of the transient response of the vibrating systems, the results might be far away from the reality, if a very fine mesh were not used in the finite element modeling of the system.

4.4 Reflection of the Shock Wave

The sum of two counter-propagating waves (of equal amplitude and frequency) creates a standing wave. Standing waves commonly arise when a boundary blocks further propagation of the wave, thus causing wave reflection, and therefore introducing a counter-propagating wave. For example when the flat plate is excited by the impulse hammer, longitudinal waves propagate out to the end of the plate, there upon the waves are reflected back. The two traveling waves can either cancel or amplify the wave intensity of the other. This effect is known as interference. The wave speed is a constant given by [45]

\[ c = \sqrt{\frac{E(1-\nu)}{\rho(1+\nu)(1-2\nu)}} = 5960 \frac{m}{s} \]

where \( E \) is Young’s modulus, \( \nu \) is the Poisson ratio, and \( \rho \) is the mass density.

Time for the reflection of shock wave to pass and return to accelerometer can be obtained by dividing the plate length to the wave speed.

\[ t = \frac{L}{c} = 0.09 \text{ ms} \]
where \( L \) is the length of the plate 0.54 m and \( c \) is the wave speed. Figure 4.5 shows the experimental time history of the plate. Since the reflection time is very small the high amplitude can be seen in different places in the time history response.

The hat section has three sides and two flanges with the length of 0.304 m and 0.125 m respectively. The total length of the hat section is 1.16 m. The wave reflection time can be calculated by dividing the hat section length to the wave speed.

\[
t = \frac{L}{c} = 0.2 \text{ ms}
\]

Figure 4.22 shows the experimental time history of the hat section. The peaks are appearing in time spots multiple of 0.2 ms. At these times, the shock and its reflection either mitigate or amplify each other.
CHAPTER 5

SHOCK TRANSMISSION THROUGH THE BOLTED JOINTS

5.1 Introduction

Mechanical fastening remains the primary means of joining components in any mechanical structure. Stress concentrations that develop around the holes severely reduce the strength of the structure, it is important that the best available tools are used for analysis and design of mechanical joints. Inefficiently designed joints can have a severely detrimental effect on the weight-saving advantage of composites over metals, while incomplete understanding of stresses and failure in joints could lead to catastrophic failure of the structure. Current industry design methods are largely based on design charts and stress handbooks. Finite element modeling plays a limited role, analyses generally being two-dimensional. Even in the finite element model of large mechanical structures, the details of small components usually are ignored. Considering the small components details in large scale models results in a model which either can not be solved by today's systems and solvers or wait days to get the result from the computer. It seems the predefined procedures in finite element solvers would be necessary in future, to save computer solving time, designer or analyzer time. It will help to make the model less complicated and more understandable.
5.2 Geometry and Dimensions of the Structure

Figure 5.1 shows the bolted joint structural configuration chosen for impact response analysis. The structure consists of five major parts: Hat section, spacers (washers), flat plate, bolts and nuts. The structure is assembled by putting the spacers between flat plate and hat section. Hex bolts and nuts are used to put them together. The dimension of each component is presented in this section.

![Assembly drawing of the bolted joint structure](image)

Figure 5.1 Assembly drawing of the bolted joint structure

**Hat section**

The hat section is made from 6.35 mm (¼ in) steel plate (Figure 5.2). These dimensions have been suggested by an Army Research Laboratory (ARL) team as a good start for joint configuration.
Spacer

The metric plain washer has been used as the spacer between hat section and flat plate. Figure 5.3 shows the dimensions of the washer. The plain washer is 10 mm, narrow, steel, zinc plated according the ANSI B18.22M-1981, R1990 [46].

Figure 5.3. Plain washer, narrow, steel, zinc plated (dimensions are in mm).
Flat plate

The rectangular flat plate made from 6.35 mm (¼ in) steel (Figure 5.4). It is made from the same material as the hat section.

Figure 5.4. Flat plate (dimensions are in mm)

Bolts and nuts

Class 8.8, M10×1.25 hex bolts and nuts are used to connect the flat plate to the hat section. The bolts and nuts dimensions follow the ANSI B18.2.3.5M-1979, R1989 standard [46]. Figure 5.5 and show the dimensions of the hex bolts and nuts.

Figure 5.5. M10×1.25, class 8.8, hex bolt (dimensions are in mm)
5.3 Material Properties

Bolts, nuts and washers are made from class 8.8 steel. Hat section and flat plate are made from hot roll ASTM-A36 steel. Table 5-1 shows the material properties of each part of the structure [43].

Table 5-1. Mechanical properties of the bolted joint parts

<table>
<thead>
<tr>
<th>Part</th>
<th>Material</th>
<th>Density (Kg/m3)</th>
<th>Modulus of elasticity (Pa)</th>
<th>Yield stress (Pa)</th>
<th>Poisson ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hat section</td>
<td>ASTM-A36 steel (hot roll)</td>
<td>7.85x10^6</td>
<td>200x10^9</td>
<td>250x10^6</td>
<td>0.3</td>
</tr>
<tr>
<td>Flat plate</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Spacers (washers)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bolts</td>
<td>Class 8.8 steel</td>
<td>7.85x10^6</td>
<td>200x10^9</td>
<td>660x10^6</td>
<td>0.3</td>
</tr>
<tr>
<td>Nuts</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

5.4 Appropriate Bolt Size

In the experiment the large impulse hammer (PCB 086C20) is used to apply a shock to the structure. The maximum peak force can be applied by hammer is about 20000 N [39]. A M10x1.25 class 8.8 would be the required bolt size for this application.

5.5 Experiment

5.5.1 Test Setup and Procedure

The tests setup includes an A-frame structure to support a hanging mass, hat section configuration, accelerometers, impulse hammer, and a laptop computer. Detailed experimental procedure was explained on chapter two. Figure 5.6 shows the hammer hitting the side of the hat section. Figure 5.7 is a close up photo of hat section and the plate connected together with four hex bolts and nuts.
Figure 5.6. Bolted joint experimental setup

Figure 5.7. Hat section and plate connected together with bolts
5.5.2 Accelerometer Locations

Figure 5.8 shows the places that accelerations have been measured. One of the accelerometers is on the hat section, and the other one is on the plate. The accelerometer on the hat section is located at the center of the side of the hat section. It is mounted on the outside surface of the hat section. The center of the other side of the hat section is the impact point. The accelerometer on the plate is located exactly on the center of the plate. It is mounted on the inside surface of the plate.

![Diagram of accelerometer locations](image)

Figure 5.8. The location of accelerometers

5.5.3 Applied Force

An impact force with the peak about 2000 N with duration of 0.0016 s has been applied to the model. The data for the force curve, shown in Figure 5.9, has been obtained.
from the experiment captured by Pulse Software. The detailed experiment procedure was explained on chapter two. For sake of simplicity this force considered as nodal force and has been applied to the center node on the side plane of hat section (Figure 1.8).

![Applied Force to Bolted Joint Structure](image)

Figure 5.9. Force curve applied to the finite element models

5.6 Finite Element Analysis

Two finite element models have been generated using beam, shell and solid elements. The first model contains all solid elements. The second finite element model uses shell element for hat section and flat plate and beam element for the bolt. Figure 5.10 shows compare the two the finite element models side by side.
5.6.1 Shell-Beam Finite Element Model

The shell-beam finite element model uses shell element for hat section and flat plate and beam element for the bolt. There are 41868 elements on this model. The hat section and plate have 7452 and 3772 shell elements, respectively. The spacers modeled with 640 solid elements. They size are small compare to the whole structure and there is no point modeling them as shell elements. There are 4 beam models, which represent the four bolts on the structure. The nut was not modeled because the beam elements were used for modeling the bolt. Figure 5.11 is a screen shot of this model. Altair HyperMesh® is used for modeling the structure. The model then is exported to LS-Dyna for solving and simulating the shock transmission through the joint. There are two set of contact surface in this model. One set is between the spacer and the hat section, and the
other set is between the spacer and the plate. AUTOMATIC_SURFACE_TO_SURFACE control card in LS-Dyna were used to define the contact behavior AUTOMATIC_SURFACE_TO_SURFACE allows the two surfaces to slide on each other.

Figure 5.11. Shell element structure with beam element bolts
5.6.2 Solid-Solid Finite Element Model

Figure 5.12 shows the solid element model of the bolted structure. All parts are modeled with 97424 solid elements. The hat section, plate and washer are made of 58592, 31536 and 512 solid elements respectively. The bolts are made of 5760 elements (1440 elements per bolt) and the nuts are made of 1024 elements (256 elements per nut). There are contact surfaces between hat section and bolt, hat section and spacer, plate and bolt, plate and spacer, plate and nut and also spacer and bolt. The AUTOMATIC_SURFACE_TO_SURFACE control card is used for all these contact surfaces.

5.7 Results

Experiments were conducted on the bolted joint structure to determine the transient response in a similar fashion as the structure without joints. Accelerometers were placed on the side of the hat section and the middle of flat plate. The hat section is impacted at the opposite side as was done in the case of single hat sections. Load curve obtained from the experiment, which was applied on the finite element model. The finite element was solved and the acceleration of nodes corresponding the accelerometer locations were extracted from the result. There are two time history plots for each finite element model. One of them shows the experimental and FE acceleration of the hat, and the other one shows the experimental and FE acceleration of the plate.
5.7.1 Time history comparison

5.7.1.1 Shell-Beam Finite Element Model

The comparisons between the experimental and finite element analysis for the shell beam model is shown in Figure 5.13 and Figure 5.14. Figure 5.13 is generated by the accelerometer mounted on the hat section and the acceleration of the corresponding
node on the finite element model. Figure 5.14 is generated by the accelerometer mounted on the plate and the acceleration of the corresponding node on the finite element model.

![Graph showing FEA comparisons using shell-beam model with experimental results (Acceleration Vs Time) obtained from accelerometer mounted on hat]

Figure 5.13. FEA comparisons using shell-beam model with experimental results (Acceleration Vs Time) obtained from accelerometer mounted on hat

![Graph showing FEA comparisons using shell-beam model with experimental results (Acceleration Vs Time) obtained from accelerometer mounted on plate]

Figure 5.14. FEA comparisons using shell-beam model with experimental results (Acceleration Vs Time) obtained from accelerometer mounted on plate
As it can be seen from Figure 5.13, the finite element analysis predicts almost the same response at the hat acceleration up to 0.005 seconds after which discrepancies creep in and a phase shift is observed. Nevertheless the amplitudes remain almost the same, which are most important since damages to components in vehicles due to shock are a function of the magnitudes of the accelerations that the components are subjected to. The acceleration plot from the plate shows that the finite element prediction does not match with the experimental response, there is not only a difference seen in the magnitudes but it is also observed that they do not follow a similar pattern. Relative error between the experimental and finite element analysis data was calculated using the methods shown in Chapter 2, error in the finite element analysis models is also calculated with respect to the experiment for the peak amplitudes which are of utmost importance in shock analysis. The error analysis results are presented in Table 5-2. Figure 5.15 and Figure 5.16 shows the shock response spectrum for the shell beam model.

![Graph showing FEA comparisons using shell-beam model with experimental results (SRS) obtained from accelerometer mounted on hat](image)

Figure 5.15. FEA comparisons using shell-beam model with experimental results (SRS) obtained from accelerometer mounted on hat.
5.1.2 Solid-Solid Finite Element Model

The comparisons between the experimental and solid-solid finite element analysis for this model is shown in Figure 5.17 and Figure 5.18. Figure 5.17 is generated by the accelerometer mounted on the hat section and the acceleration of the corresponding node on the finite element model. Figure 5.18 is generated by the accelerometer mounted on the plate and the acceleration of the corresponding node on the finite element model.

The hat acceleration plots in Figure 5.17 shows that finite element model follows the general trend of the vibration, however it shows a lower amplitude vibration after 0.005 seconds. It can be seen from the plate acceleration graph in Figure 5.18, that the finite element prediction does not match with the experimental response. The finite element response becomes worse after 0.005 sec. Some peaks were matched in the simulation before this time, but there is no partial match after 0.005 sec. Figure 5.19 and Figure 5.20 shows the shock response spectrum of solid-solid finite model.

Figure 5.16. FEA comparisons using shell-beam model with experimental results (SRS) obtained from accelerometer mounted on plate
Figure 5.17. FEA comparisons using solid-solid model with experimental results (Acceleration Vs Time) obtained from accelerometer mounted on hat.

Figure 5.18. FEA comparisons using solid-solid model with experimental results (Acceleration Vs Time) obtained from accelerometer mounted on plate.
Figure 5.19. FEA comparisons using solid-solid model with experimental results (SRS) obtained from accelerometer mounted on hat

Figure 5.20. FEA comparisons using solid-solid model with experimental results (SRS) obtained from accelerometer mounted on plate
The conclusion drawn from the acceleration plots can be verified by running error analysis over the experimental and FE analysis results. Table 5-2 includes the error of finite element models on both hat section and plate.

<table>
<thead>
<tr>
<th>Finite Element Model</th>
<th>Structure</th>
<th>Regular Method (Average Acc.)</th>
<th>Mean Square Error</th>
<th>Moment Method</th>
<th>Max Peak Acc.</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shell-Beam Model</td>
<td>Hat</td>
<td>35.6</td>
<td>54.0</td>
<td>36.0</td>
<td>41</td>
<td>42.2</td>
</tr>
<tr>
<td></td>
<td>Plate</td>
<td>26.9</td>
<td>61.9</td>
<td>27.3</td>
<td>32.5</td>
<td>34.5</td>
</tr>
<tr>
<td>Solid-Solid Model</td>
<td>Hat</td>
<td>24.2</td>
<td>26.9</td>
<td>24.7</td>
<td>26.1</td>
<td>26.3</td>
</tr>
<tr>
<td></td>
<td>Plate</td>
<td>13.9</td>
<td>59.1</td>
<td>14.4</td>
<td>19.0</td>
<td>21.3</td>
</tr>
</tbody>
</table>

Results of the bolted joint structure can be divided to four cases. These cases are

- Hat section acceleration generated by shell-beam model
- Plate acceleration generated by shell-beam model
- Hat section acceleration generated by solid-solid model
- Plate acceleration generated by solid-solid model

Two of these cases belong to shell-beam model, and the other two cases belong to solid-solid model. According to error analysis presented in Table 5-2, the solid-solid model gives a better match in both hat section and plate. This is not very surprising since from the analysis of the structures without joints we find out that solid element models always gave better match with the experimental result. One more conclusion can be drawn from the error analysis of the bolted joint. Finite element simulation of the hat section acceleration is better than the plate acceleration. This is true on both models. This fact leads us to the point that some changes such as changing the CONTACT card or bolt
material properties might improve our result. This investigation is the subject of the next chapter.

5.8 Filtering the High Frequency

"Classical numerical techniques for the prediction of dynamic behavior like FEM/BEM become less suitable at high frequencies." Similar to this argument can be found in many literatures. It was claimed, "FEM is unable to predict the spatial variation of energy throughout the structure. This energy is transported by waves of different types in components like beams, plates and acoustic cavities". We would like to see if this argument is true for the transient response. If the finite element method is unable to predict the high frequency responses, then we should be able to get a better response by filtering high frequency components from the acceleration. To prove this argument we filtered the responses at 10000, 8000, 6000, 4000, 2000 and 1000 (Hz). Then we calculate the error between experiment and finite element method. Based on the argument we should get a better finite element and smaller error with decreasing the filtering frequency. The filtered time history responses of the hat section and plate are presented in the appendix. Figure 5.21 shows the error of filtered acceleration predicted by shell-beam element. The error changes only few percent. Even when the result is filtered at 1000 Hz, we still see a large error in both hat section and plate acceleration. This means the shell-beam model cannot be reliable in simulating the transient responses, either in high frequency or low frequency.
Figure 5.21. Error of shell-beam finite element prediction versus filtering frequency

Figure 5.22 shows the error of filtered acceleration predicted by solid-solid element. The error decreases with decreasing filtering frequency. This means, from solid-solid model, we can obtain a better match in low frequencies transient responses compare to high frequency. This graph shows the finite element method can be slightly reliable in predicting the transient responses with low frequencies.

Figure 5.22. Error of solid-solid finite element prediction versus filtering frequency
5.9 The Effect of Bolted Joints on Shock Mitigation

This section studies the effect of the joint on mitigating the shock waves, which pass through the joints. The experiment uses two accelerometers with one mounted on the hat section and the other mounted on the plate (Figure 5.8). Both accelerometers are oriented to respond to the flexural bending vibrational modes. The length of the side of the hat section from the bend point to the bolt is 0.37 m and the length of the plate from bolt to bolt is 0.41 m. The hat section and plate have the same thickness, so the flexural stiffness of the side and bottom plate are similar. Figure 5.23 shows the experimental result of the hat section and plate.

![Figure 5.23. Experimental time history response of the (a) hat section and (b) plate](image)

The maximum acceleration in the hat section is about 2400 m/s² and in the bottom plate is about 800 m/s², as it can be seen from Figure 5.23 (a) and (b). Thus, the bolted joint is reducing the maximum acceleration amplitude by a factor of 3.
Figure 5.24 shows the SRS graphs of the hat section and plate. The peaks of the two SRS graphs seem to appear at the same frequency for the hat section and the plate, which confirm the accuracy of the modal analysis results. However, the intensity of the shock in the plate is lower by 3 to 6 times than corresponding shock intensity in the hat section. This reduction in the shock intensity is due to the fact the shock waves are passing from the hat section to the plate via two sets of bolted joints. These bolted joints provide the connection between the hat section and the plate. It is also important to note that the two sections are physically separated by steel washer and there is no direct contact between them.

One possible explanation of the shock wave mitigation phenomena is the fact that flexural waves resulting from the impact have to transfer their nature into axial waves and additional shear waves to travel through the short bolts with the 0.04 m length, which have a very high flexural stiffness. Also energy is being dissipated through friction between the bolt and washer surfaces as the bolt assembly vibrates.
CHAPTER 6

FINITE ELEMENT ANALYSIS OF JOINT PERFORMANCE

6.1 Introduction

In the previous chapter, the finite element method was used for the simulation of transient shock response through bolted joints. The dissimilarity factor obtained from finite element of bolted joints were much higher than structure without joints. The first part of this chapter focuses on modeling issues to improve the simulation and reduce the difference between experiment and finite element response. One of the goals of the first part of this chapter is to try to identify and possibly fix the source of the higher errors in the bolted joint structure. The second part of this chapter investigates the effect of the bolted joint in shock transmission through the structure. The last part of this chapter discretization of finite element response over the simulation period. The discretization helps to see the change in similarity of FEA and experiment at different time intervals during the simulation.

6.2 Parameters Effecting the Simulation

One of the goals of this chapter is to try to identify and fix the source of the higher errors in the bolted joint structure. There are some parameters in finite element modeling of the bolted structure that might the simulation results. This chapter investigates the
The effect of these parameters on the transient response of the structure. The most important parameters in modeling bolted joint structures are:

- Contact types and parameters
  - Contact types
  - Friction
- Structural Damping
- Unanticipated contact surfaces
- Torque level on bolt (preload)
- Mesh refinement of bolt and spacer
- Material Damping
- FEA results output frequency

Table 6-1 includes the quantitative comparison of solid-solid finite element model and experimental data, which presented in chapter 5. In this chapter, we are going to change each parameter one by one and determine if this change affects the transient response.

<table>
<thead>
<tr>
<th>Structure</th>
<th>Regular Method (Average Acc.)</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acceleration</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hat</td>
<td>24.2</td>
<td>26.9</td>
<td>24.7 26.1 26.3</td>
<td>30.1</td>
<td>26.9</td>
</tr>
<tr>
<td>Plate</td>
<td>13.9</td>
<td>59.1</td>
<td>14.4 19.0 21.3</td>
<td>19.8</td>
<td>18.2</td>
</tr>
</tbody>
</table>

Table 6-1. Comparison of experiment and FE model on bolted joint structure
6.2.1 Contact Types and Parameters

6.2.1.1 Contact Types

Contact provides a way for treating interaction between disjoint parts. Different types of contacts are available in LS-DYNA solver. Among all existing contact keywords, the following list of contact keywords is suitable and applicable for the structural impact study.

AUTOMATIC_NODES_TO_SURFACE

AUTOMATIC_SURFACE_TO_SURFACE

It is necessary to mention that during after the impact there is sliding between surfaces, however these sliding might be so small that the eyes can not catch them. Despite this fact, we have used the following control cards to answer any doubts about the contact issues.

TIED_NODES_TO_SURFACE

TIED_SURFACE_TO_SURFACE

The contact type is not the only concern in modeling of the interfaces. The contact parameters also can be changed to define the interfaces more properly. This is the list of all contact parameters that can be customized for the study of shock transmission through bolted joints.

- Static coefficient of friction (FS)
- Dynamic coefficient of friction (FD)
- Exponential decay coefficient

The effect of contact on the transient response was determined using the following procedure. First the contact type has changed in the finite element model. The
model has been solved using LS-DYNA and in the post processing stage the acceleration was extracted from finite element solutions. This procedure was repeated four times for four different contact types. Figure 6.1 shows the hat section acceleration from two models with different contact type. One of the model uses AUTOMATIC_NODES_TO_SURFACE and the other one uses AUTOMATIC_SURFACE_TO_SURFACE. Figure 6.2 shows the plate acceleration from the same two models with different contact type. As it can be seen the responses are exactly identical. The response from the models with TIEDNODES_TO_SURFACE and TIED_SURFACE_TO_SURFACE were also identical to AUTOMATIC_SURFACE_TO_SURFACE. The acceleration of these models were identical to Figure 6.1 and Figure 6.2, so there was no point on presenting the same graph again and again. The conclusion of contact study is that changing the contact type does not affect the transient response.

6.2.1.2 Contact Parameters

Friction is the other parameter in the contact issue that might change the transient response of the bolted joint structures. In LS-DYNA, friction can be changed by modifying any of the following parameters [44]

- Static coefficient of friction
- Dynamic coefficient of friction
- Exponential decay coefficient
Figure 6.1. Acceleration of the hat section obtained by two finite element models with different contacts

Figure 6.2. Acceleration of the plate obtained by two finite element models with different contacts

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
The frictional coefficient is assumed to be dependent on the relative velocity \( v_{\text{rel}} \) of the surfaces in contact \( \mu_c = FD + (FS - FD)e^{-DC|v_{\text{rel}}|} \). In this equation FS is static coefficient of friction and must be greater than zero. FD is dynamic coefficient of friction. DC is the exponential decay coefficient. Table 6-2 shows the value for coefficient of friction for two steel surfaces sliding on each other [47]. The static coefficient of friction range is between 0.05 to 0.78 depends on the conduction of two surfaces. The range for dynamic coefficient of friction is 0.029 to 0.57.

<table>
<thead>
<tr>
<th>Material 1</th>
<th>Material 2</th>
<th>Coefficient of Friction</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Dry</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Static</td>
</tr>
<tr>
<td>Steel (Mild)</td>
<td>Steel (Mild)</td>
<td>0.74</td>
</tr>
<tr>
<td>Steel (Hard)</td>
<td>Steel (Hard)</td>
<td>0.78</td>
</tr>
<tr>
<td>Steel</td>
<td>Zinc (plated on steel)</td>
<td>0.5</td>
</tr>
</tbody>
</table>

To be more conservative, a wider range for FS and FD used to investigate the effect of Friction on the transient response of bolted joint. The FS numbers chosen for simulation was in between 0 to 4 and the FD and DS numbers were between 0 and 1. If these numbers are not defined, the LS-DYNA uses zero as default for these values. Figure 6.3 and Figure 6.4 shows the acceleration plots of two models with different friction coefficients. The acceleration plots of the models with different values of friction coefficient were exactly identical to the graphs, so there is no point of inserting all the graphs captured from the FE models with different friction coefficient values. Looking at the acceleration plot shows that friction has no effect on the transient response. This leads us to the fact that there might not be any motion between hat section and spacer or spacer.
and plate. Experimentally, it is possible to check whether there is any motion between the parts or not, and this can be one of the tasks for the continuation of this study.

Figure 6.3. Hat section acceleration plots of two models with different friction coefficients

Figure 6.4. Plate acceleration plots of two models with different friction coefficients
6.2.2 Structural Damping

Damping coefficient is one of the parameters the analyst must consider in simulating the nonlinear behavior of shock through bolted joints. The Rayleigh damping is considered the most common approach due to its simplicity and frequency-dependent characteristic. The Rayleigh damping is based on a linear viscous representation in which the damping is frequency dependent and proportional to structure velocity [48]. The damping matrix in Rayleigh damping is defined as [44]:

\[ C = \alpha M + \beta K \]

Where \( C \), \( M \) and \( K \) are the damping, mass and stiffness matrices, respectively. The constants \( \alpha \) and \( \beta \) are the mass and stiffness proportional damping constants. For large systems, identification of valid damping coefficients \( \alpha \) and \( \beta \) for all significant modes is a very complicated task. That depends on the visualization response of the structure under various natural frequencies.

Adding damping to the finite element model of bolted joint structure changes the response of the system. Comparing the finite element response with experimental data showed that adding a damping coefficient to the FE model filters the high frequency vibrations. Applying higher values of damping coefficients leads to increase in filtering high frequency vibrations. The closest match between experiment and FEA is for the case that there is zero damping (the default in LS-DYNA) applied to the finite element model.

6.2.3 Unanticipated Contact Surfaces

There could be incomplete contact (or uneven contact) between the spacer and the hat section or plate. This would be difficult to quantify in the experiment but we can check to see if this is a possible source of error computationally. We can you change the
size of the spacer slightly in the model (make the diameter a little smaller and larger) to simulate a smaller or larger contact area in the actual spacer. The objective here would be to see if this would make a significant difference in the FEA acceleration response. The diameter of the spacer is 20 mm. Two more finite element models are made where washer’s diameter has changed to 15 and 35 mm. The result of solving these two models are presented in Table 6-3. Modifying the contact area will change the transient response, but these changes are insignificant. The first two rows in Table 6-3 belong to the original model which spacer is modeled with its actual diameter. The third and forth rows belong to the model with smaller contact area. The last two rows are the error for the model with larger contact area. The regular and moment method show that using smaller contact area improves the simulation of hat section’s acceleration, but worsen the plate acceleration. Having the larger contact area in the finite element model does not change the results significantly.

Table 6-3. Comparison of finite element models with different contact area

<table>
<thead>
<tr>
<th>Contact Area</th>
<th>Structure</th>
<th>Regular Method (Average Acc.)</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acc.</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td>Equal to Spacer (Φ=20 mm)</td>
<td>Hat</td>
<td>24.2</td>
<td>26.9</td>
<td>24.7</td>
<td>26.1</td>
<td>26.3</td>
</tr>
<tr>
<td></td>
<td>Plate</td>
<td>13.9</td>
<td>59.1</td>
<td>14.4</td>
<td>19.0</td>
<td>21.3</td>
</tr>
<tr>
<td>Smaller than Spacer (Φ=15 mm)</td>
<td>Hat</td>
<td>20.8</td>
<td>26.7</td>
<td>21.3</td>
<td>21.2</td>
<td>20.3</td>
</tr>
<tr>
<td></td>
<td>Plate</td>
<td>17.0</td>
<td>51.7</td>
<td>17.5</td>
<td>21.6</td>
<td>23.6</td>
</tr>
<tr>
<td>Larger than Spacer (Φ=35 mm)</td>
<td>Hat</td>
<td>24.0</td>
<td>27.3</td>
<td>24.5</td>
<td>26.0</td>
<td>26.1</td>
</tr>
<tr>
<td></td>
<td>Plate</td>
<td>15.2</td>
<td>56.4</td>
<td>15.7</td>
<td>19.8</td>
<td>21.8</td>
</tr>
</tbody>
</table>

134
6.2.4 Torque Level on Bolt (Preload)

One of the primary parameters in analyzing bolted joints is preload in the bolt [13, 49, 50]. Three preload conditions are studied in this project. The preload of 10.5KN, 37.5 KN and 50 KN corresponding to torque of 21 N-m, 75 N-m and 100 N-m are used. The effect of preload on the structure is studied. Figure 6.5 shows the pre-stress of 456 MPa in the bolted joint for the preload of 37.5 KN. The pre-stress is constant throughout the transient analysis.

![Figure 6.5. Structure showing the constant pre-stress of 460 MPa](image)

The FFT analysis of the structure for different preload is shown in Figure 6.6. The three FFT curves corresponding to bolt torque of 100, 75, 21 N-m are identical. This shows that the preload of the bolt have no effect on the response of the structure. The Table 6-4 show the mode number and natural frequency of the structure.
Figure 6.6. FFT of hat section for 100, 75 and 21Nm Torque.

![Natural Frequency of Hat Section](image)

**Table 6-4. Natural frequency of structure**

<table>
<thead>
<tr>
<th>Mode</th>
<th>Natural Frequency (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>21 N-m pretorque</td>
</tr>
<tr>
<td>7</td>
<td>68</td>
</tr>
<tr>
<td>8</td>
<td>124</td>
</tr>
<tr>
<td>9</td>
<td>196</td>
</tr>
<tr>
<td>10</td>
<td>244</td>
</tr>
<tr>
<td>11</td>
<td>368</td>
</tr>
<tr>
<td>12</td>
<td>416</td>
</tr>
</tbody>
</table>

Figure 6.7 shows the acceleration vs. time plots for the structure measured at two points – one on the hat section and one on the plate. These results correspond to preload of 50 KN (pretorque 100 N-m) compare to experiment. The blue and red curves represent experiment and simulation results respectively.
Natural frequency of the structure is same for 100, 75 and 21 Nm torque on bolt. This concludes that the response of the structure will be same for any kind of preload. As it can be seen in Figure 6.7, there is a fairly good match between the experiment and analysis on the hat section acceleration. However, the analysis gives lower amplitude acceleration than the experiment. There is more than 50% reduction in the amplitude of the acceleration after the joint. As long as the bolts were not very loose, the change in preload is not going to affect the transient response.

6.2.5 Mesh Refinement of Bolt and Spacer

Having a finer mesh is one of the first suggestions that come after obtaining not so perfect result in finite element study. The model chosen for the study of bolted joints has 4 elements along the thickness (97424 total elements in the model). The result of mesh refinement on chapter 4 showed that there is not much improvement in the models with 5 or 6 elements along thickness instead of 4. In fact, the reduction of error was less that 1 or 2 percent, based on most of the error criteria methods. It is not expected to have much better result with having finer mesh in bolted joint structure.
In addition this reason, at this time, it is not possible to have a finer mesh for the bolted joint structure presented in chapter 5. We tried to model the bolted joint with 5 or 6 elements through the thickness, but the computer crashed during the modeling. After couple of try on different computers, we realize this task it is not possible with the available computers. Having said that the computer technology changes everyday and this issue can be one of the tasks on continuation of this study.

6.2.6 FEA Results Output Frequency

There seems to be a high frequency oscillation in the experiments that is not captured in the FEA. The question is if this is arbitrarily filtered by the output frequency of the FEA results. The finite element model of the bolted joint structure model was solved three times, with different time step each time. Table 6-5 includes the time step and output frequency of the finite element analysis.

<table>
<thead>
<tr>
<th>Case</th>
<th>Time Step (Δt) (sec)</th>
<th>Output Frequency (KHz)</th>
<th>Number of points in 0.010 sec</th>
</tr>
</thead>
<tbody>
<tr>
<td>201</td>
<td>1.5259e-5</td>
<td>65.5</td>
<td>657</td>
</tr>
<tr>
<td>202</td>
<td>1.5259e-6</td>
<td>655</td>
<td>6555</td>
</tr>
<tr>
<td>203</td>
<td>1.5259e-7</td>
<td>6553</td>
<td>65532</td>
</tr>
</tbody>
</table>

The hat accelerations for all three cases are exactly identical. Figure 6.8 shows the hat section accelerations on the same plot. Since they are identical they sit on top of each other. Figure 6.9 shows the plate accelerations for the three cases. The plate accelerations are also identical and they sit on top of each other as shown in Figure 6.9. These plots
prove that applying higher output frequency will not change the simulation of the finite element models to shock transmission through bolted joints.

Figure 6.8. Hat accelerations – results output frequency

Figure 6.9. Plate acceleration – results output frequency
6.3 Effect of the Joint in Shock Transmission Through the Structure

6.3.1 Modulus and Density of Bolt and Spacer

Modifying material properties of bolt and spacer might improve our result. The bolt and spacer are made of steel. However, they modeled exactly on the original model (chapter 5), it is interesting to see the effect of material properties on the transient response. The bolt and spacer modulus of elasticity and density are 200 GPa and 7810 Kg/m$^3$. The speed of shock through the material is depended to $\sqrt{E/\rho}$. Table 6-6 shows the bolt and spacer modulus and density of the finite element models made to study the effect of bolt/spacer material properties. The simulation results of these cases are presented in Table 6-7 and Table 6-8. Table 6-7 shows the difference of hat section acceleration obtained from different finite element models. Based on regular and moment methods, cases 12, 22, 32 and 42 show improvement in the simulation of hat section acceleration. In all of these cases the modulus of elasticity is 13.1 GPa. It is difficult to compare all the dissimilarity factors in the table format.

For better illustration of the material property investigation, the difference between zeroth moment from finite element models with different values of E and $\rho$ are plotted in Figure 6.10. Similarly, Figure 6.11 shows the difference between first moment from finite element models with different values of E and $\rho$. Figure 6.12 shows the dissimilarity factor for finite element models with different values of E and $\rho$. There are four curves in each plot, corresponding to four different values of density. The trend of change is almost similar in all these plots. The difference is minimum, when E=13.1 GPa. The best scenario are for cases 22 and 32 when E=13.1 GPa and $\rho$=7810 or 124960.
It is necessary to mention that these are the results of hat section acceleration. No definite conclusion can be withdrawn without looking at the plate acceleration as well.

Table 6-6. Modulus and density of the bolt/spacer

| Case | $E$ (GPa) | $\rho$ (kg/m³) | $\sqrt{E/\rho}$ | Ratio of $\sqrt{E/\rho}$
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>11</td>
<td>3.28</td>
<td>2710</td>
<td>1100.15</td>
<td>0.212</td>
</tr>
<tr>
<td>12</td>
<td>13.1</td>
<td>2710</td>
<td>2198.62</td>
<td>0.424</td>
</tr>
<tr>
<td>13</td>
<td>210</td>
<td>2710</td>
<td>8802.88</td>
<td>1.70</td>
</tr>
<tr>
<td>14</td>
<td>3360</td>
<td>2710</td>
<td>35211.54</td>
<td>6.79</td>
</tr>
<tr>
<td>15</td>
<td>13440</td>
<td>2710</td>
<td>70423.08</td>
<td>13.58</td>
</tr>
<tr>
<td>21</td>
<td>3.28</td>
<td>7810</td>
<td>648.05</td>
<td>1/8</td>
</tr>
<tr>
<td>22</td>
<td>13.1</td>
<td>7810</td>
<td>1295.12</td>
<td>1/4</td>
</tr>
<tr>
<td>23</td>
<td>210</td>
<td>7810</td>
<td>5185.42</td>
<td>1</td>
</tr>
<tr>
<td>24</td>
<td>3360</td>
<td>7810</td>
<td>20741.69</td>
<td>4</td>
</tr>
<tr>
<td>25</td>
<td>13440</td>
<td>7810</td>
<td>41483.38</td>
<td>8</td>
</tr>
<tr>
<td>31</td>
<td>3.28</td>
<td>124960</td>
<td>162.01</td>
<td>1/32</td>
</tr>
<tr>
<td>32</td>
<td>13.1</td>
<td>124960</td>
<td>323.78</td>
<td>1/16</td>
</tr>
<tr>
<td>33</td>
<td>210</td>
<td>124960</td>
<td>1296.36</td>
<td>1/4</td>
</tr>
<tr>
<td>34</td>
<td>3360</td>
<td>124960</td>
<td>5185.42</td>
<td>1</td>
</tr>
<tr>
<td>35</td>
<td>13440</td>
<td>124960</td>
<td>10370.84</td>
<td>2</td>
</tr>
<tr>
<td>41</td>
<td>3.28</td>
<td>499840</td>
<td>81.01</td>
<td>1/64</td>
</tr>
<tr>
<td>42</td>
<td>13.1</td>
<td>499840</td>
<td>161.89</td>
<td>1/32</td>
</tr>
<tr>
<td>43</td>
<td>210</td>
<td>499840</td>
<td>648.18</td>
<td>1/8</td>
</tr>
<tr>
<td>44</td>
<td>3360</td>
<td>499840</td>
<td>2592.71</td>
<td>1/2</td>
</tr>
<tr>
<td>45</td>
<td>13440</td>
<td>499840</td>
<td>5185.42</td>
<td>1</td>
</tr>
</tbody>
</table>

Notes:

1- Case 23 is the original case (bolt/spacer are modeled with steel mechanical properties.

2- The ratio of $\sqrt{E/\rho}$ (last column) is obtained by dividing $\sqrt{E/\rho}$ of each model to $\sqrt{E/\rho}$ of the original model.
Table 6-7. Comparison of Hat section acceleration for FE models with different bolt/spacer material properties

<table>
<thead>
<tr>
<th>Case</th>
<th>E (GPa)</th>
<th>ρ (kg/m³)</th>
<th>Regular Method (Average Acc)</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acceleration</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0th 1st 2nd</td>
<td>0th 1st 2nd</td>
<td>0th 1st 2nd</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>3.28</td>
<td>2710</td>
<td>32</td>
<td>44.6</td>
<td>32.3 36 37</td>
<td>33</td>
<td>34.2</td>
</tr>
<tr>
<td>12</td>
<td>13.1</td>
<td>2710</td>
<td>20.3</td>
<td>30.9</td>
<td>20.6 20.8 19.8</td>
<td>26</td>
<td>22.1</td>
</tr>
<tr>
<td>13</td>
<td>210</td>
<td>2710</td>
<td>24.5</td>
<td>26</td>
<td>24.6 25.7 25.8</td>
<td>32.8</td>
<td>27.4</td>
</tr>
<tr>
<td>14</td>
<td>3360</td>
<td>2710</td>
<td>26.88</td>
<td>25.85</td>
<td>26.9 28.5 28.9</td>
<td>34.54</td>
<td>29.8</td>
</tr>
<tr>
<td>15</td>
<td>13440</td>
<td>2710</td>
<td>26.57</td>
<td>26.26</td>
<td>26.6 28.3 28.7</td>
<td>34.98</td>
<td>29.8</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>21</td>
<td>3.28</td>
<td>7810</td>
<td>31.6</td>
<td>44</td>
<td>31.9 35.6 37</td>
<td>34.3</td>
<td>34.4</td>
</tr>
<tr>
<td>22</td>
<td>13.1</td>
<td>7810</td>
<td>20.1</td>
<td>29</td>
<td>20.4 20.6 19.6</td>
<td>20.3</td>
<td>20.3</td>
</tr>
<tr>
<td>23</td>
<td>210</td>
<td>7810</td>
<td>24.2</td>
<td>26.9</td>
<td>24.7 26.1 26.3</td>
<td>30.1</td>
<td>26.9</td>
</tr>
<tr>
<td>24</td>
<td>3360</td>
<td>7810</td>
<td>26</td>
<td>27.1</td>
<td>26 28 28.5 37</td>
<td>36.3</td>
<td>29.5</td>
</tr>
<tr>
<td>25</td>
<td>13440</td>
<td>7810</td>
<td>25.6</td>
<td>26.9</td>
<td>25.6 27.3 27.8</td>
<td>36.3</td>
<td>29.5</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>31</td>
<td>3.28</td>
<td>124960</td>
<td>32.6</td>
<td>42.7</td>
<td>32.8 36.6 38.4</td>
<td>37.5</td>
<td>34.4</td>
</tr>
<tr>
<td>32</td>
<td>13.1</td>
<td>124960</td>
<td>20</td>
<td>29.3</td>
<td>20.3 21.1 20.6</td>
<td>23.5</td>
<td>20.3</td>
</tr>
<tr>
<td>33</td>
<td>210</td>
<td>124960</td>
<td>26.4</td>
<td>29.6</td>
<td>26.7 29.8 31.1</td>
<td>32.3</td>
<td>26.9</td>
</tr>
<tr>
<td>34</td>
<td>3360</td>
<td>124960</td>
<td>25.7</td>
<td>29.3</td>
<td>26.2 29.2 30.4</td>
<td>31.1</td>
<td>30.1</td>
</tr>
<tr>
<td>35</td>
<td>13440</td>
<td>124960</td>
<td>25.7</td>
<td>29.4</td>
<td>25.9 28.9 30.1</td>
<td>30.7</td>
<td>29.5</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>41</td>
<td>3.28</td>
<td>499840</td>
<td>30.4</td>
<td>40.3</td>
<td>30.7 34.6 36.5</td>
<td>31.9</td>
<td>36.0</td>
</tr>
<tr>
<td>42</td>
<td>13.1</td>
<td>499840</td>
<td>21.7</td>
<td>32.7</td>
<td>22.1 23.4 23</td>
<td>30.1</td>
<td>21.5</td>
</tr>
<tr>
<td>43</td>
<td>210</td>
<td>499840</td>
<td>20</td>
<td>29.2</td>
<td>20.3 23 24.1 33.4</td>
<td>29.8</td>
<td></td>
</tr>
<tr>
<td>44</td>
<td>3360</td>
<td>499840</td>
<td>24</td>
<td>29.2</td>
<td>24.3 28 29.6 38.7</td>
<td>29.1</td>
<td></td>
</tr>
<tr>
<td>45</td>
<td>13440</td>
<td>499840</td>
<td>25</td>
<td>30</td>
<td>25.5 29.3 31.2</td>
<td>37.6</td>
<td>28.7</td>
</tr>
</tbody>
</table>
Figure 6.10. Comparison of hat section acceleration using zeroth moment (Effect of modulus and density)

Figure 6.11. Comparison of hat section acceleration using first moment (Effect of modulus and density)
Table 6-8 shows the quantitative comparison of plate acceleration simulations. For better illustration these differences are plotted in Figure 6.13, Figure 6.14 and Figure 6.15. There are four curves in each plot, which are corresponding to four different values for density. As it can be seen in the dissimilarity factor plots, the minimum DF calculated where E is greater than 210 GPa. The models with E=13.1 GPa, with better results in hat section, show worse dissimilarity factor on the plate acceleration.
Table 6-8. Comparison of plate acceleration for FE models with different bolt/spacer material properties

<table>
<thead>
<tr>
<th>Case</th>
<th>E (GPa)</th>
<th>( \rho ) (kg/m³)</th>
<th>Regular Method (Average Acc.)</th>
<th>Moment Method</th>
<th>Max Peak Acceleration</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Mean Square Value</td>
<td>0th</td>
<td>1st</td>
<td>2nd</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>2nd</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>3.28</td>
<td>2710</td>
<td>31.8</td>
<td>56.9</td>
<td>32.1</td>
<td>38</td>
</tr>
<tr>
<td>12</td>
<td>13.1</td>
<td>2710</td>
<td>20.4</td>
<td>44.2</td>
<td>20.8</td>
<td>25.3</td>
</tr>
<tr>
<td>13</td>
<td>210</td>
<td>2710</td>
<td>13</td>
<td>62.8</td>
<td>13.1</td>
<td>17.7</td>
</tr>
<tr>
<td>14</td>
<td>3360</td>
<td>2710</td>
<td>12.93</td>
<td>59.6</td>
<td>12.92</td>
<td>17.99</td>
</tr>
<tr>
<td>15</td>
<td>13440</td>
<td>2710</td>
<td>13.09</td>
<td>58.87</td>
<td>13.09</td>
<td>18.22</td>
</tr>
<tr>
<td>21</td>
<td>3.28</td>
<td>7810</td>
<td>35.6</td>
<td>54.3</td>
<td>35.9</td>
<td>40.9</td>
</tr>
<tr>
<td>22</td>
<td>13.1</td>
<td>7810</td>
<td>22.6</td>
<td>40</td>
<td>22.9</td>
<td>26.3</td>
</tr>
<tr>
<td>23</td>
<td>210</td>
<td>7810</td>
<td>13.89</td>
<td>59.1</td>
<td>14.4</td>
<td>19</td>
</tr>
<tr>
<td>24</td>
<td>3360</td>
<td>7810</td>
<td>13.8</td>
<td>56.18</td>
<td>13.8</td>
<td>18.8</td>
</tr>
<tr>
<td>25</td>
<td>13440</td>
<td>7810</td>
<td>13.8</td>
<td>56.7</td>
<td>13.8</td>
<td>18.6</td>
</tr>
<tr>
<td>31</td>
<td>3.28</td>
<td>124960</td>
<td>38.5</td>
<td>48.6</td>
<td>38.8</td>
<td>35.2</td>
</tr>
<tr>
<td>32</td>
<td>13.1</td>
<td>124960</td>
<td>26.3</td>
<td>45.3</td>
<td>26.6</td>
<td>26.7</td>
</tr>
<tr>
<td>33</td>
<td>210</td>
<td>124960</td>
<td>17.3</td>
<td>51.4</td>
<td>17.7</td>
<td>20.3</td>
</tr>
<tr>
<td>34</td>
<td>3360</td>
<td>124960</td>
<td>17.4</td>
<td>51.6</td>
<td>17.9</td>
<td>20.1</td>
</tr>
<tr>
<td>35</td>
<td>13440</td>
<td>124960</td>
<td>17.9</td>
<td>50.5</td>
<td>18</td>
<td>20.5</td>
</tr>
<tr>
<td>41</td>
<td>3.28</td>
<td>499840</td>
<td>51</td>
<td>56.6</td>
<td>51.2</td>
<td>50.1</td>
</tr>
<tr>
<td>42</td>
<td>13.1</td>
<td>499840</td>
<td>36</td>
<td>46.3</td>
<td>36.2</td>
<td>33</td>
</tr>
<tr>
<td>43</td>
<td>210</td>
<td>499840</td>
<td>17</td>
<td>53.2</td>
<td>17.3</td>
<td>14.1</td>
</tr>
<tr>
<td>44</td>
<td>3360</td>
<td>499840</td>
<td>18.4</td>
<td>57.8</td>
<td>18.7</td>
<td>17.3</td>
</tr>
<tr>
<td>45</td>
<td>13440</td>
<td>499840</td>
<td>18.5</td>
<td>57.6</td>
<td>19</td>
<td>18.1</td>
</tr>
</tbody>
</table>

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Figure 6.13. Comparison of plate acceleration using zeroth moment (Effect of modulus and density)

Figure 6.14. Comparison of plate acceleration first moment (Effect of modulus and density)
Figure 6.15. Dissimilarity Factor for plate acceleration (Effect of modulus and density)

Ultimately, the original case has the best results. This can be proved by considering both hat section and plate errors. On the conclusion we can say that modifying the bolt/spacer material properties either does not change the results or gives a better result on hat section and worse on plate.

6.3.2 Orthotropic Modulus of Bolt and Spacer

The shock generated from impact uses the bolt and spacer to pass from hat section to plate. It propagates along the bolt’s shank and bolt diameter. The shock transmission is depended on $\sqrt{E/\rho}$ where E is the modulus of elasticity and $\rho$ is density. Assigning different modulus elasticity in axial and radial direction of bolt and spacer might improve the result. This means the bolt and spacer should be modeled as orthotropic martial. According to knowledge of the author and pervious literature survey this has never been done before. By modeling bolt as orthotropic material we have control on both longitudinal and transverse shock transmission through the joint.

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
By definition, an orthotropic material has at least 2 orthogonal planes of symmetry, where material properties are independent of the direction within each plane [51]. Such materials require 9 independent variables (i.e. elastic constants) in their constitutive matrices. These 9 independent variables are the 3 Young's modulus \((E_a, E_b, E_c)\), the 3 Poisson's ratios \((\nu_{bc}, \nu_{ca}, \nu_{ab})\) and the 3 shear modulus \((G_{bc}, G_{ca}, G_{ab})\). Figure 6.16 shows the orientation of the local axis in orthotropic model of a bolt. Axis 'a' is along the bolt axial direction. Axes 'b' and 'c' are along the bolt radial directions.

![Figure 6.16. Orientation of local axis in orthotropic bolt model](image)

The bolt can be modeled as a special case of an orthotropic solid is one that contains a plane of isotropy (this implies that the solid can be rotated with respect to the loading direction about one axis without measurable effect on the solids response). Then, transverse isotropy requires that [51]:

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
\[ E_b = E_c \]
\[ \nu_{ca} = \nu_{ab} \]
\[ G_{ca} = G_{ab} \]

Four finite element models made with using orthotropic material properties for bolt and spacer. Table 6-9 includes material properties consistent with that coordinate system shown in Figure 6.16.

<table>
<thead>
<tr>
<th>Case</th>
<th>Finite Element Model</th>
<th>( E_a ) (GPa)</th>
<th>( E_b = E_c ) (GPa)</th>
<th>( \nu_{ab} = \nu_{ca} )</th>
<th>( \nu_{bc} )</th>
<th>( G_{ab} = G_{ca} ) (GPa)</th>
<th>( G_{bc} ) (GPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 23</td>
<td>Isotropic Model</td>
<td>210</td>
<td>---</td>
<td>0.3</td>
<td>---</td>
<td>---</td>
<td>81</td>
</tr>
<tr>
<td>Case 101</td>
<td>Orthotropic Model</td>
<td>210</td>
<td>13.1</td>
<td>0.0244</td>
<td>0.3</td>
<td>5</td>
<td>81</td>
</tr>
<tr>
<td>Case 102</td>
<td>Orthotropic Model</td>
<td>210</td>
<td>70</td>
<td>0.0244</td>
<td>0.3</td>
<td>27</td>
<td>81</td>
</tr>
<tr>
<td>Case 103</td>
<td>Orthotropic Model</td>
<td>210</td>
<td>40</td>
<td>0.0244</td>
<td>0.3</td>
<td>15.4</td>
<td>81</td>
</tr>
<tr>
<td>Case 104</td>
<td>Orthotropic Model</td>
<td>140</td>
<td>70</td>
<td>0.0244</td>
<td>0.3</td>
<td>27</td>
<td>70</td>
</tr>
</tbody>
</table>

The results of investigations connected with orthotropic modeling of bolts and spacers are presented in Table 6-10. The results indicate that the cases show smaller discrepancy on that hat section, and larger discrepancy on the plate acceleration. The same trend observed in the pervious section for the cases with smaller modulus. The orthotropic bolt modeling will change the result, but it gives a better result on hat section and worse result on plate acceleration. The objective of this project is to compare the
accelerations on both before and after the joints. For this objective, the original model gives the best answer.

Table 6-10. Comparison of finite element models with orthotropic bolt and spacer

<table>
<thead>
<tr>
<th>Case</th>
<th>Structure</th>
<th>Regular Method (Average Acc.)</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acc.</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0th</td>
<td>1st</td>
<td>2nd</td>
</tr>
<tr>
<td>23</td>
<td>Hat</td>
<td>24.2</td>
<td>26.9</td>
<td>26.4</td>
<td>26.3</td>
<td>30.1</td>
</tr>
<tr>
<td></td>
<td>Plate</td>
<td>13.9</td>
<td>59.1</td>
<td>14.4</td>
<td>19</td>
<td>21.3</td>
</tr>
<tr>
<td>101</td>
<td>Hat</td>
<td>20.2</td>
<td>29.4</td>
<td>20.5</td>
<td>20.8</td>
<td>19.8</td>
</tr>
<tr>
<td></td>
<td>Plate</td>
<td>23.3</td>
<td>40.3</td>
<td>23.6</td>
<td>26.3</td>
<td>27.7</td>
</tr>
<tr>
<td>102</td>
<td>Hat</td>
<td>22.7</td>
<td>25.8</td>
<td>23</td>
<td>23.6</td>
<td>23.2</td>
</tr>
<tr>
<td></td>
<td>Plate</td>
<td>14.2</td>
<td>58.4</td>
<td>14.5</td>
<td>19.5</td>
<td>22.3</td>
</tr>
<tr>
<td>103</td>
<td>Hat</td>
<td>20.9</td>
<td>26.3</td>
<td>21.2</td>
<td>21.8</td>
<td>21.3</td>
</tr>
<tr>
<td></td>
<td>Plate</td>
<td>17.3</td>
<td>53.3</td>
<td>17.6</td>
<td>21.7</td>
<td>23.7</td>
</tr>
<tr>
<td>104</td>
<td>Hat</td>
<td>22.5</td>
<td>25.8</td>
<td>22.8</td>
<td>23.4</td>
<td>22.9</td>
</tr>
<tr>
<td></td>
<td>Plate</td>
<td>14.4</td>
<td>57.8</td>
<td>14.8</td>
<td>19.7</td>
<td>22.5</td>
</tr>
</tbody>
</table>

6.4 Discretization of Finite Element Response

Breaking the results into time regions will help us to understand where in the simulation error is accumulating. The time regions are 0-2, 2-4, 4-6, 6-8 and 8-10 ms. These time regions are chosen based on the impact time. The objective to this study would be to quantify the time at which the error goes from acceptable, to unacceptable. Figure 6.17 and Figure 6.18 compare the hat section and plate acceleration from finite element and experiment.
Figure 6.17. Hat section acceleration versus time obtained from experiment and FEA

Figure 6.18. Plate acceleration versus time obtained from experiment and FEA
Table 6-11 shows hat section dissimilarity factor for the predefined time spans. Figure 6.19 is the dissimilarity factor of hat section. Based on the recommendation of Army Research Laboratory engineers, 20% dissimilarity factor is acceptable for the design purposes. Based on regular method, the finite element answer is acceptable up to 4 ms and it is not reliable beyond that time. The zeroth moment method shows 4 ms as the acceptable time span, but the first moment method decreases the acceptable time span to 2.5 ms. It is necessary to mention that the complete conclusion cannot be withdrawn without looking at the plate errors. Like hat section, the finite element simulation is never reliable for our application, if the root mean square is considered as the only error criteria. The regular method and zeroth moment method show that the plate error always stay below 20%. The first moment method shows 8 ms as the acceptable time span, where the error is below the threshold limit.

Considering both hat section and plate errors, the acceptable range is about 4 ms based on regular method. The zeroth and first moment method show 4 ms as the acceptable range and based on root mean square method the answer always is not acceptable.

<table>
<thead>
<tr>
<th>Time Span</th>
<th>Regular Method</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acc.</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0th</td>
<td>1st</td>
<td>2nd</td>
<td>0-2 ms</td>
<td>9</td>
</tr>
<tr>
<td>0-2 ms</td>
<td>9</td>
<td>103.45</td>
<td>8.3</td>
<td>13.77</td>
<td>16.93</td>
</tr>
<tr>
<td>0-4 ms</td>
<td>16.61</td>
<td>54.43</td>
<td>17.14</td>
<td>20.92</td>
<td>23.29</td>
</tr>
<tr>
<td>0-6 ms</td>
<td>19.66</td>
<td>49.61</td>
<td>20.17</td>
<td>23.1</td>
<td>24.44</td>
</tr>
<tr>
<td>0-8 ms</td>
<td>23.74</td>
<td>27.49</td>
<td>24.22</td>
<td>26.57</td>
<td>27.45</td>
</tr>
<tr>
<td>0-10 ms</td>
<td>24.26</td>
<td>26.93</td>
<td>24.74</td>
<td>26.14</td>
<td>26.29</td>
</tr>
</tbody>
</table>

Table 6-11. Discritzed hat section dissimilarity factor on impact time span
Table 6-12. Discritzed plate dissimilarity factor on impact time span

<table>
<thead>
<tr>
<th>Time Span</th>
<th>Regular Method</th>
<th>Mean Square Value</th>
<th>Moment Method</th>
<th>Max Peak Acc.</th>
<th>DF</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>0th</td>
<td>1st</td>
<td>2nd</td>
</tr>
<tr>
<td>0-2 ms</td>
<td>4.74</td>
<td>111.32</td>
<td>4.07</td>
<td>8.97</td>
<td>8.04</td>
</tr>
<tr>
<td>0-4 ms</td>
<td>4.42</td>
<td>75.05</td>
<td>3.76</td>
<td>5.25</td>
<td>5.2</td>
</tr>
<tr>
<td>0-6 ms</td>
<td>5.69</td>
<td>63.68</td>
<td>6.29</td>
<td>9.95</td>
<td>13.04</td>
</tr>
<tr>
<td>0-8 ms</td>
<td>12.17</td>
<td>64.6</td>
<td>12.73</td>
<td>18.5</td>
<td>22.9</td>
</tr>
<tr>
<td>0-10 ms</td>
<td>13.89</td>
<td>59.14</td>
<td>14.44</td>
<td>19.03</td>
<td>21.34</td>
</tr>
</tbody>
</table>

Figure 6.19. Hat section dissimilarity factor versus time span

Figure 6.20. Plate dissimilarity factor versus time span
6.5 Summary

This chapter looks at the effect of different aspects of modeling in the transient response of bolted structure. The modeling parameters investigated in this chapter are: Contact types and parameters, friction, preload, vibration characteristics of bolt and spacers, mesh refinement of bolt and spacer and output frequency.

Among these parameters contact type, friction coefficients and output frequency have absolutely no effect on the transient response and the response is exactly identical. Modifying the contact area will change the transient response, but these changes are insignificant. Moreover, as long as the bolts were not very loose, the change in preload is not going to affect the transient response. It is not expected to have much better result with having finer mesh in bolted joint structure. Changing the bolt material properties or modeling bolt with orthotropic material, either does not change the results or gives a better result on hat section and worse on plate.

The last part of chapter discusses about acceptable time range where the error stays below 20%. Considering both hat section and plate errors, the acceptable range is about 4 ms based on regular method. The zeroth and first moment method show 4 ms as the acceptable range and based on root mean square method the answer always is not acceptable.
CHAPTER 7

SUMMARY, CONCLUSIONS, AND RECOMMENDATIONS

7.1 Summary

An experimental and numerical investigation of simple and bolted joint structure was conducted in this research project. There is a need to quantify the time history responses, because there are a lot of experimental and numerical cases must be compared to each other. All the possible error criteria for comparing two time histories were reviewed and explained. Their application, strongness and weaknesses in quantifying time histories were discussed. Calibration is the first step on any experimental conduct. A solid bar was chosen to calibrate our experimental equipment. In addition, a finite element model of solid bar was made to obtain the numerical responses. Dynamic response of a simple structure was obtained numerically and experimentally. The comparison of responses verifies that the method of approach works for a simple structure. Before moving to jointed structure, the experiment and analysis were performed on two simple structures without any joints. A steel hat section and a plate were the subjects of the study in this part of the project. There are two purposes for this part of structure. Firstly, it helps us to understand the shock transmission through structures without joints. Secondly, it determines whether there is an identical match between analysis and the experiment or not, and if they do not match how much is the error between the numerical and experimental responses.
A bolted joint structure was tested under shock loading. The bolted joint structure consisted of a steel hat section and a steel plate which are bolted together. The structure was tested under impact loads. Horizontal impacts were applied to the side of hat section in order to simulate the shock transmission through the structure. The load duration is very short (about 1.5 ms) which puts the loading condition under the ‘shock excitation’ category. A finite element model based on explicit dynamic formulation was developed for the analysis of bolted joint structure. Material and geometric nonlinearity, and the contact area between the surfaces were included in the model. The finite element model was used to simulate the modal analysis and dynamic responses of the structure. Since the solution strategy in the explicit formulation does not involve iteration, the analysis was completed without any numerical difficulty. In general, excellent agreement was observed between experimental and numerical modal analysis. The time history of the bolted joint structure was predicted well with the finite element model, but the predicted response was not very satisfactory compare to experimental result.

The discrepancies between the analysis and the experimental transient response can be minimized by modifying different modeling parameters. The parameters describing the behavior of bolted joint structure subjected to shock excitation. A parametric study was conducted to identify the effect of some of the main parameters on the structure transient response. These parameters are contact types, contact surface area, friction between parts, preload, mesh refinement, spacer and bolt material properties and finite element output frequency. Moreover, the bolt was modeled with orthotropic material properties with the purpose of having more control over longitudinal and transverse vibration.
7.2 Conclusions

A extensive literature survey showed that there is little work done on the shock transmission through bolted joints. Most of the available articles on structural dynamic analysis rely on modal analysis for comparing transient responses and only few compare the time histories. None of the published articles investigate the transient shock transmission through bolted joint in detail by comparing the time histories.

This study involves finite element analysis and experimental work. Methods for comparing the transient response from experiments and analyses are investigated. The error generated between two transient responses can be the effect of phase shift or amplitude difference or both. Regular Method, Root Mean Square (RMS), Moment Method, Method of Temporal Moments and Maximum Peak and Peak Counting Method are the error criteria that can be used to quantify the difference between two time histories. Regular method is easy to use, but the error calculated with this method is time dependent. For the sake of comparison of two signals, we need a single value over a comparison time period. This makes the regular method not applicable for the objective of this study. However getting an average value of error can be chosen as criterion, but it is not a complete because it does not detect the phase shift error. The moment method determines whether the error is coming from amplitude difference or phase shift, but in order to compare two signals with the moment method they must be stationary. This property of moment method makes it suitable for most time history applications. The method of temporal moment characterizes the transient time histories, so it is not applicable for the cases that part of transient time history is in the interest of the researchers. This method is not applicable for the time histories that cannot be divided to
transient and steady state response. Both maximum peak method and peak counting method consider a particular characteristic of a curve as error criterion. Although they might be useful for particular cases that the maximum amplitude is in the interest of the researchers, they cannot be used to verify the similarity of two signals. This means they should not be used for comparison applications, because they do not represent the whole curve.

Calibration is the first and most necessary step of all experimental projects. A solid round bar with the diameter of 0.0381 m and length of 0.1968 m is used for experimental calibration of shock transmission through structures. In addition, this calibration determined that the finite element method is capable of simulating the transient responses. The natural frequencies of structure determined analytically, numerically and experimentally. The modal analysis obtained by these three methods showed that natural frequencies perfectly match each other. It is not possible to obtain the time history analytically, but numerical and experimental time histories were identical.

A steel plate and a hat section were the subject of the study of shock transmission through structure without joints. Modal analysis of the plate and single hat section shows that experimental and finite element analysis results have good agreement. The finite element analysis proves to be proficient in replicating the structural behavior of the hat sections. Both the shell and solid element models in all the cases generate almost the same frequencies. The time history results from the plate show more congruity between the finite element and experimental results when compared to the single hat sections.

The mesh refinement study of plate and hat section finite element analysis proves that the models with high number of elements show a good accuracy. If a model with
coarse mesh used for the simulation of the transient response of the vibrating systems, the results might be far away from the reality.

Two finite element models were made to explore the shock behavior through bolted joint structure. One of the models made with shell element and beam element used to model bolts. All the parts in the other model were made of solid elements. The time histories were compared with experimental data. The beam element representation of the bolt in the structures with bolted joints does not yield the desired results, the comparison between the experiment and finite element are divergent.

The solid-solid model gives a better match in both hat section and plate. This is not very surprising since from the analysis of the structures without joints we find out that solid element models always gave better match with the experimental result. Based on regular method and moment method the hat section errors are larger than 20%. The plate errors are in the range of 13 to 15%. The results were filtered to omit the high frequency oscillations. The error shell-beam model results did not change with the filtering, but in the solid-solid element, the error decreases with decreasing filtering frequency. This means, form solid-solid model, we can obtain a better match in low frequencies transient responses compare to high frequency. Filtering results showed that the finite element method could be slightly reliable in predicting the transient responses with low frequencies.

The results from the structure without joint show more congruity between the finite element and experimental results when compared to the bolted joint structure. The main reason behind this is the fact that the structures without joints are continuous structures, and the shock travels along the structure uninterrupted.
The SRS plots showed that experiment and finite element analysis predict the
damaging frequencies as long as the finite element model has a refined meshed. The SRS
plot shows small peaks, when the finite element model has coarse mesh. The designer
must have a careful consideration about small peaks in SRS plots, especially when the
plots generated by finite element model without a refined mesh.

The parts in jointed structure are two separate from each other, which are
connected to each other using spacers and bolts. The discontinuity in the structure causes
the divergence in the higher frequencies between the finite element analysis and
experimental results. The reason behind this might be from some of the modeling
parameters such as contact types, contact area, friction, preload, vibration characteristics
of bolt and spacers, mesh refinement of bolt and spacer and output frequency.

Among these parameters contact type, friction coefficients and output frequency
have absolutely no effect on the transient response and the response is exactly identical.
Modifying the contact area will change the transient response, but these changes are
insignificant. Moreover, as along as the bolts were not very loose, the change in preload
is not going to affect the transient response. It is not expected to have much better result
with having finer mesh in bolted joint structure. Changing the bolt material properties or
modeling bolt with orthotropic material, either does not change the results or gives a
better result on hat section and worse on plate.

Dividing the transient response into segments of time intervals and calculating the
error on the time spans showed that errors can be below 15% in time periods less than
simulation time. Considering both hat section and plate errors, the acceptable range is
about 4 ms based on regular method. The zeroth and first moment method show 2.5 ms as
the acceptable range and based on root mean square method the answer always is not acceptable.

7.3 Future Work

Future work involves focusing on the experimental and finite element studies of a very simple bolted lap joint with two long beams. Since the structure is simple it can be modeled with more highly refined bolt model. The impact must be applied such a way that the structure vibrates only transversally. The errors from this experiment and analysis must be smaller than values obtained in this project.

The same experimental and numerical procedure as explained in this project should be repeated with the same structure but made from other material like aluminum or composite materials, such as fiberglass composites. Another important study will be the shock transmission in a heavier structure to investigate the mass effects on shock transmission. Other test may include high impacts using the air gun available at UNLV to investigate the capability of finite element models in predicting very high frequency transient responses.

The future work in this task includes determining the various factors that reduces the shock amplitude after the joint. The different method of shock isolation can be applied numerically to the finite element model to verify the effectiveness of each method.

Recently, the researcher has started using Energy Finite Element Method (EFEM) or statistical energy Statistical Energy Methods (SEA) for mid-frequency and high
vibration transmission analysis. Using one of these methods to investigate the shock transmission through bolted joint can be a continuation of this study.
APPENDIX

Filtered Results for Shell-Beam FE Model

Bolted Joints – Shell-Beam Model – Hat Section Acceleration – Filtered at 10000 Hz

Bolted Joints – Shell-Beam Model – Plate Acceleration – Filtered at 10000 Hz

Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.
Bolted Joints – Shell-Beam Model – Hat Section Acceleration – Filtered at 8000 Hz

Bolted Joints – Shell-Beam Model – Plate Acceleration – Filtered at 8000 Hz
Bolted Joints – Shell-Beam Model – Hat Section Acceleration – Filtered at 6000 Hz

Bolted Joints – Shell-Beam Model – Plate Acceleration – Filtered at 6000 Hz
Bolted Joints – Shell-Beam Model – Hat Section Acceleration – Filtered at 4000 Hz

Bolted Joints – Shell-Beam Model – Plate Acceleration – Filtered at 4000 Hz
Bolted Joints – Shell-Beam Model – Hat Section Acceleration – Filtered at 2000 Hz

Bolted Joints – Shell-Beam Model – Plate Acceleration – Filtered at 2000 Hz
Filtered Results for Solid-Solid FE Model

Bolted Joints – Solid-Solid Model – Hat Section Acceleration – Filtered at 10000 Hz

Bolted Joints – Solid-Solid Model – Plate Acceleration – Filtered at 10000 Hz
Bolted Joints – Solid-Solid Model – Hat Section Acceleration – Filtered at 8000 Hz

Bolted Joints – Solid-Solid Model – Plate Acceleration – Filtered at 8000 Hz
Bolted Joints – Solid-Solid Model – Hat Section Acceleration – Filtered at 6000 Hz

Bolted Joints – Solid-Solid Model – Plate Acceleration – Filtered at 6000 Hz
Bolted Joints – Solid-Solid Model – Hat Section Acceleration – Filtered at 4000 Hz

Bolted Joints – Solid-Solid Model – Plate Acceleration – Filtered at 4000 Hz
Bolted Joints – Solid-Solid Model – Hat Section Acceleration – Filtered at 2000 Hz

Bolted Joints – Solid-Solid Model – Plate Acceleration – Filtered at 2000 Hz
REFERENCES


174


Reproduced with permission of the copyright owner. Further reproduction prohibited without permission.


[25] FOX5 Las Vegas (KVVU), "Weather 24-7," 2005,
http://www.kvvu.com/global/weather.asp

http://www.weather.com/


VITA

Graduate College
University of Nevada, Las Vegas

Masoud Feghhi

Home Address:
1950 Eldridge Pkwy
Apt#14202
Houston, TX 77077

Degrees:
Bachelor of Science, Mechanical Engineering, 1996
K.N. Toosi University of Technology, Tehran

Master of Science, Mechanical Engineering, 2002
Villanova University, Villanova, Pennsylvania

Dissertation Title: Experimental and Finite Element Studies of Shock Transmission through Bolted Joints

Dissertation Examination Committee:
Chairperson, Dr. Brendan J. O'Toole, Ph. D.
Committee Member, Dr. Woosoon Yim, Ph. D.
Committee Member, Dr. Douglas Reynolds, Ph. D.
Committee Member, Dr. Samir Moujaes, Ph. D.
Graduate College Faculty Representative, Dr. Samaan Ladkany, Ph. D.