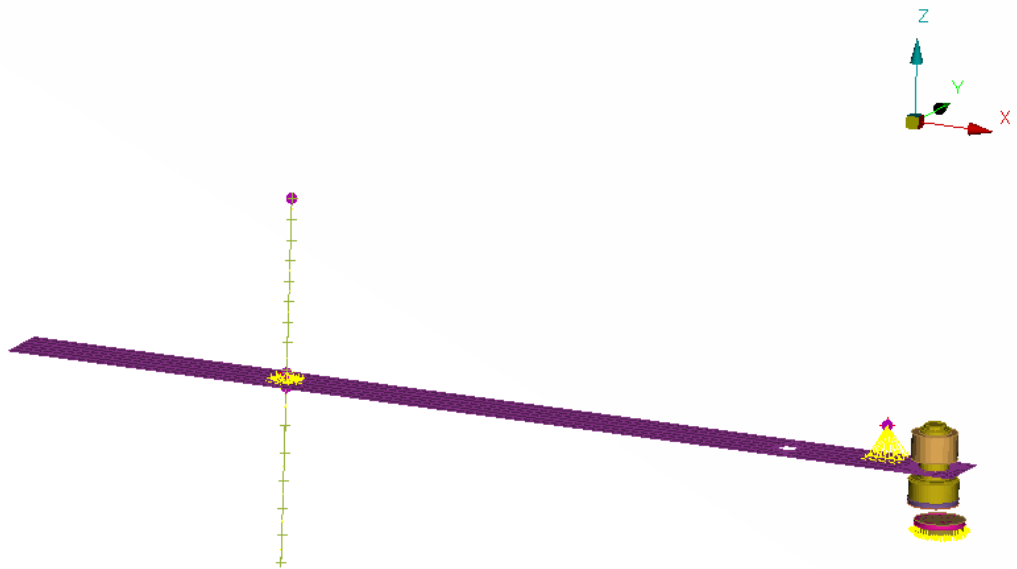




CHALMERS
UNIVERSITY OF TECHNOLOGY



Rattle Predictions using Linear and Non-Linear Structural Dynamics Simulations

Master's thesis in Applied Mechanics

SURAJ SRINATH MUTHYA
SRIKANNAN SELVARAJ

Department of Mechanics and Maritime Sciences

CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2021
www.chalmers.se

MASTER'S THESIS 2021:12

Rattle predictions using linear and non-linear structural dynamics simulations

SURAJ SRINATH MUTHYA
SRIKANNAN SELVARAJ



CHALMERS
UNIVERSITY OF TECHNOLOGY

Department of Mechanics and Maritime Sciences
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2021

Rattle Predictions using linear and non-linear structural dynamics simulations
SURAJ SRINATH MUTHYA
SRIKANNAN SELVARAJ

© SURAJ SRINATH MUTHYA, 2021.
© SRIKANNAN SELVARAJ, 2021.

Supervisor: Mohsen Bayani, Solidity group, Volvo Car Corporation
Examiner: Thomas Abrahamsson, Mechanics and Maritime Sciences, Chalmers University of Technology

Master's Thesis 2021
Department of Mechanics and Maritime Sciences
Chalmers University of Technology
SE-412 96 Gothenburg
Telephone +46 31 772 1000

Cover: Rattle FE-model

Typeset in L^AT_EX
Printed by Chalmers Reproservice
Gothenburg, Sweden 2021

Rattle predictions using linear and non-linear structural dynamic simulations
SURAJ SRINATH MUTHYA
SRIKANNAN SELVARAJ
Department of Mechanics and Maritime Sciences
Chalmers University of Technology

Abstract

The purpose of this project is to study the parameters in the already developed metrics for the Squeak & rattle, which are annoying sounds in a passenger car, as a result of two surfaces sliding or due to normal impact respectively. These are indicators of poor quality and burden warranty costs to the car manufacturers. The rattle event is predicted by validating against experimental data and to compare the accuracy of the S&R risk severity metrics calculated from the linear and non-linear structural dynamics simulations. Though there is a lack of accurate simulations and post-processing methods, the methods used here helps in the evaluation of these issues in the product development process itself by virtual simulations and parameter study. A simple mechanical system is studied for the impact study and various output parameters from the experimental rattle setup are correlated with the simulation results in order to reduce the computational costs of complex setups and mass production.

Keywords: Rattle, Implicit and Explicit, modal analysis, frequency response or steady state dynamics, linear and non-linear, structural dynamics, model correlation

Acknowledgements

We would like to thank our supervisor Mohsen Bayani for giving us this opportunity and for his patient guidance, enthusiastic encouragement and useful critiques of this project work. His advice and assistance in keeping our progress on schedule and motivating us throughout by providing useful references and keeping us in the right direction.

Furthermore, we would like to thank our examiner Thomas Abrahamsson for providing us with a lot of valuable insights and be a part of this project on 'Rattle predictions using linear and non-linear structural dynamics simulations'.

A special mention to Anoob Basheer, for his support and valuable inputs during the project work.

Suraj Srinath Muthya, Gothenburg, June 2021
Srikannan Selvaraj, Gothenburg, June 2021

Nomenclature

Abbreviations:

S&R - Squeak and rattle
VCC - Volvo Car Corporation
CAE - Computer aided engineering
FE - Finite element
FFT - Fast Fourier transform
IFFT - Inverse fast Fourier transform
SDE - Structural dynamics equation
SSD - Steady state dynamics
FEM - Finite element method
FEA - Finite element analysis
MPC - Multi-Point Constraint
Al-Al - Aluminum-Aluminum
St-St - Steel-Steel
St-P1 - Steel-Polypropylene
P2-P1 - Polypropylene-Polypropylene
CP - Contact pair

α - Mass proportional material damping

β - Stiffness proportional material damping

El-El - Element-based modelling of the surfaces of the material pairs

Node-El - Node based on one surface and Element based on the other surface of the material pairs

Symbols:

M - Mass matrix [kg]

K - Stiffness matrix [N/m]

u - Load [m]

ω - Angular frequency [rad/s]

Contents

List of Figures	xi
List of Tables	xiii
1 Introduction	1
1.1 Background	1
1.2 Objective	2
1.3 Limitations	2
2 Theory	5
2.1 CAE Rattle predicting method	5
2.2 Solver Settings	5
2.3 Inverse Fourier Transform	6
2.4 Linear and non-linear in FEM	6
2.5 Modal Analysis	7
2.6 Frequency response analysis	7
2.7 Impact analysis	9
2.8 Material Damping	9
2.9 Time Domain Analysis	10
3 Methods	11
3.1 Physical Rattle setup	11
3.2 Mesh	12
3.3 Boundary Conditions	13
3.4 Signal processing	14
3.5 Omega Arithmetic	15
3.6 Material Properties	16
3.7 Contact definition	17
3.8 Excitation signal	18
3.9 FE model setup for simulation	19
3.10 Initial Conditions	20
3.11 Gapsizes Determination	23
3.12 Implicit and Explicit solvers	24
3.13 Design of Experiments	25
3.14 Model Correlation	26
3.14.1 Model tuning	26
3.15 Linear Analysis	27

4	Results & Discussions	29
4.1	St-P1 Material Pair - Implicit	29
4.1.1	Experimental Results	29
4.1.2	Simulation results - Implicit Solver	30
4.1.3	Simulation results - Explicit Solver	31
4.2	Parameter study for different cases	32
4.3	Linear Simulations	35
4.4	Correlation Between Linear and Non-linear Simulations	36
4.5	Discussions	37
5	Concluding Remarks	39
5.1	Conclusion	39
5.2	General Recommendations	39
5.3	Future Work	39
	Bibliography	41
	Bibliography	41

List of Figures

1.1	Squeak and rattle visualization	1
1.2	Physical model setup	2
2.1	Linear and non-linear characteristics of a spring	7
2.2	Steady state dynamics frequency spread interval	9
3.1	Squeak and rattle visualization	11
3.2	FE model of the Rattle setup	13
3.3	Boundary conditions of the FE model	14
3.4	Signal obtained by using omega arithmetic	15
3.5	Signal without using omega arithmetic	15
3.6	Contact between the sample beam (highlighted in red) and the sample hammer (highlighted in blue)	18
3.7	Smoothened Amplitude(y-axis) vs Time(x-axis) plot used as the input signal for the model	19
3.8	Node set 2-26 from the clamped end	20
3.9	Node set 27-33 near the shaker connector hole area	21
3.10	Node set 34-93 after the hole area till the impact area	21
3.11	Node set for the impact accelerometer area	22
3.12	Nodes selected for measuring the output parameters	22
3.13	Gap between beam and the hammer	24
4.1	Experimental data for St-P1 material pair with 0 mm gap size which includes the reaction force(N), velocity(m/s) and acceleration(g) and displacement(mm) in z-direction Vs time(s)	29
4.2	Simulation data for St-P1 using Implicit Solver for reaction force(N), velocity(m/s) and acceleration(g) and displacement(mm) Vs time(s) .	30
4.3	Displacement in z-direction(mm) Vs time(s) for St-P1 material pair using Implicit solver	30
4.4	Simulation data for St-P1 using Explicit Solver for reaction force(N), velocity(m/s) and acceleration(g) and displacement(mm) in z-direction Vs time(s)	31
4.5	Displacement in z-direction(mm) Vs time(s) for St-P1 material pair using Explicit solver	31
4.6	Comparison of velocity after impact (m/s) from the experiment with implicit and explicit simulation values for various material pairs . . .	32

4.7	Comparison of velocity before impact (m/s) from the experiment with implicit and explicit simulation values for various material pairs . . .	33
4.8	Comparison of acceleration before impact(g) from the experiment with implicit and explicit simulation values for various material pairs	33
4.9	Comparison of acceleration at impact(g) from the experiment with implicit and explicit simulation values for various material pairs . . .	34
4.10	Comparison of reaction force(N) from the experiment with implicit and explicit simulation values for various material pairs	34
4.11	simulated parameters for linear case alal-5g	35
4.12	Correlation for velocity before impact	36
4.13	correlation for acceleration before impact	37

List of Tables

3.1	Name and dimensions for the experimental parts	12
3.2	Material properties for sample beam	16
3.3	Material properties for sample hammer	17
3.4	Important output nodes and its usages	23
3.5	Material pairs for simulation with priority	25
4.1	Parameter correlation between experiment and simulation for stp1 0mm gap size.	32
4.2	Parameters obtained for linear simulations without contact	36

1

Introduction

Solidity as an attribute within personal driving experience centre is responsible for squeak and rattle (S&R) sounds and solid feeling in the car. The main responsibility is setting requirements and verification of S&R problems from early phases to after production launch. One of the major challenges of the Solidity Department is to improve the competence for attribute verification in Pre-design-freeze phases to enable upfront design concept changes driven by CAE and to meet shorter lead time with a reduced number of physical complete vehicle prototypes.

1.1 Background

Squeak and Rattle are two non-stationary sounds that occurs when two adjacent parts come into contact, either impacting or sliding. A car with in-cabin S&R problems are considered as as quality deficient by the passengers. Absence of S&R in passenger cars has been in focus since the advent of the automotive industries. Recent improvements in the quality of the stationary sounds, besides the advancements in electrification and autonomous driving, will result in further drawing the attention of non-stationary sounds inside the cabin, such as S&R. Volvo cars strategy of shifting engineering activities to early phases of product development, means every quality needs to be evaluated upfront and common practice is to use computer simulation tools (CAE). In order to secure an upfront robust verification. There is a need to improve the CAE methods we use today. Studying and comparing linear and non linear methods to address this problem is the focus of this project. The schematic representation of S&R is shown in the figure 1.1

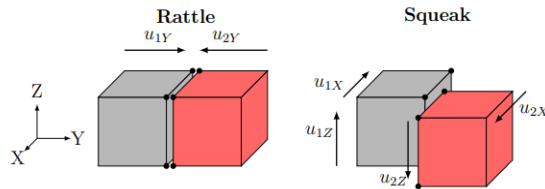


Figure 1.1: Squeak and rattle visualization

The project comprises of two parts. In the first part, a comparison between the linear and the non-linear response of a simple mechanical system concerning the parameter contributing to squeak and rattle generation will be made. The mechan-

ical systems of interest are the rattle producing machine that was used in previous studies and the stick-slip test machine that was simulated in a previous project. The experiment data are gathered in the previous project and will be used in this work. The CAE models are also developed in previous projects and the non linear simulations results are post-processed. However, some of the linear simulations are needed to be calculated and if needed the non-linear simulations are required to be partially calculated. The response in the form of S&R risk severity will be compared with respect to the objective risk values from the sound for equivalent events rather that were collected before. In the second part of the project, the findings from the first part will be evaluated for a side door assembly. The last activity is to do linear and non-linear simulation of the experimental event of the side door assembly and then comparing the results with the previously collected system response. The analysis is performed in the physical rattle setup used in Volvo cars is shown in Figure 1.2. The FE model, boundary conditions used for the rattle setup assembly is described in section 3.3.

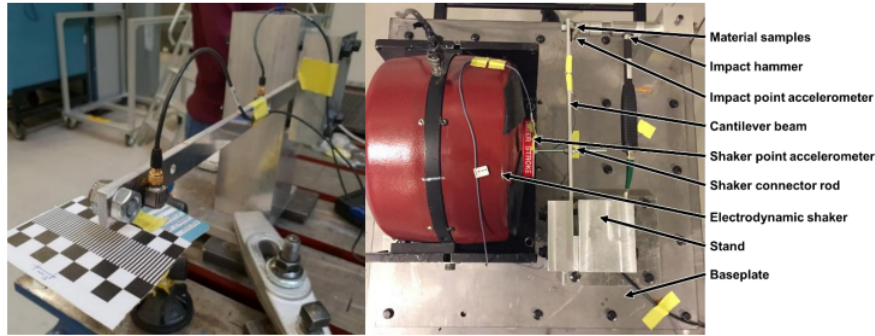


Figure 1.2: Physical model setup

1.2 Objective

The main objectives of the work are:

- To study the parameters in the already developed metrics for squeak and rattle prediction by validating against experimental data.
- To compare the accuracy of S&R risk severity metrics calculated from linear and nonlinear S&R simulations.

1.3 Limitations

The meshed CAE model is provided by VCC, so the mesh convergence study is not done for the CAE model. The experiments are carried out before the thesis work, therefore the correlation study is done only for the captured events by using the feedback from the simulation.

The gap size used between the material pair is not the desired gap size, it is adjusted based on the output from the experimental signals. The FE model is not validated

due to time constraint.

The biggest limitation was that the experiment was already conducted and the test signals were not repeated, therefore only the available data was used in the correlation work.

CAE software ANSA 18.1.4 was used for pre-processing, META 18.1.4 for post-processing, and ABAQUS 6.14 and 2019 as solver which is limited to what is used at VCC. There was a possibility to use Nastran solver, but due to the limitations in time, this did not happen.

The biggest limitation was time, however most of the correlation work for predicting the rattle event is solved and these findings will be useful for evaluating the side door assembly for the future work at VCC.

2

Theory

In this chapter, the theory used during the thesis work is described. Firstly, the solvers and the experimental physical setup used for the work is described and the types of analysis used for simulation with different material pairs is described in this section.

2.1 CAE Rattle predicting method

Two different methods are used for predicting the rattle method using CAE, one of them is SAR line or E-line method([1]) and the other method is predicted using 'Virtual Sensors'([2]). The E-line method is used in time domain and use the user defined function 3D-lines to make the displacement calculations, whereas the virtual sensor method is mainly operated in frequency domain and evaluate the displacement by using super elements. E-line method is followed by VCC to predict the rattle sounds.

2.2 Solver Settings

CAE solvers use two different solver techniques, Implicit and Explicit solvers. These two methods differ in the approach used by time incrementation (Δt). These two solvers are mainly used for the convergence criteria. Implicit solvers have a higher time increment when compared to Explicit solvers. Implicit solver is similar to an iteration technique which calculates the value for both the current and future states of the system, whereas explicit calculates only the future state of the system based on the current one.

The implicit solvers is unconditionally stable and work well for the coupled equations, while the decoupled equations are solved by explicit method. The implicit and explicit method can be chosen in the ABAQUS/Standard or ABAQUS/Explicit modules in the software before the analysis. In this project most of the analysis are done in implicit solvers, due to time consumption. The number of runs required are very high for a material pair to correlate with the experiment, so implicit solver is used.

2.3 Inverse Fourier Transform

Conversion of time domain to frequency domain is called as Fourier Transform, whereas Inverse Fourier Transform used in this project to convert the system from frequency domain to time domain.

The inverse fourier transform is represented as

$$y(t) = \int_{-\infty}^{\infty} Y(\omega) e^{-j\omega t} d\omega \quad (2.1)$$

where ω represents the angular frequency, time domain signal is represented by y , frequency domain is represented by Y and time is represented by t .

There is a another function also called as inverse fast fourier transform (IFFT) that also converts the system from frequency domain to time domain. The inverse fast fourier transform (IFFT) is represented in the equation as follows,

$$y_n = \frac{1}{N} \sum_{k=0}^{N-1} Y_k \cdot e^{i(\frac{2\pi}{N})kn} \quad (2.2)$$

Where N is the length of discrete data.

2.4 Linear and non-linear in FEM

Both linear and non-linear analysis have been used in this project. In both the analysis the stiffness matrix plays the major role, both the material and geometrical features are characterized by element stiffness matrix. The basic FEM equation is represented as follows,

$$K \cdot u = F \quad (2.3)$$

where the stiffness matrix is represented by K , u represents the load vector and F is the force vector.

In linear analysis, the stiffness matrix always tends to remain constant throughout the analysis, in this analysis when the element undergoes deformation it will be assumed that there will be a negligible change in stiffness.

In non-linear analysis, the stiffness is not constant it will be changed based on the deformation and loading of the material. Iterative techniques are used in non linear analysis to update the element stiffness matrix for each iteration. There are different aspect of non-linearity which includes, large displacements, material behaviour, geometry and with contact.

An example of linear and non-linear characteristics of a spring is shown in figure 2.1

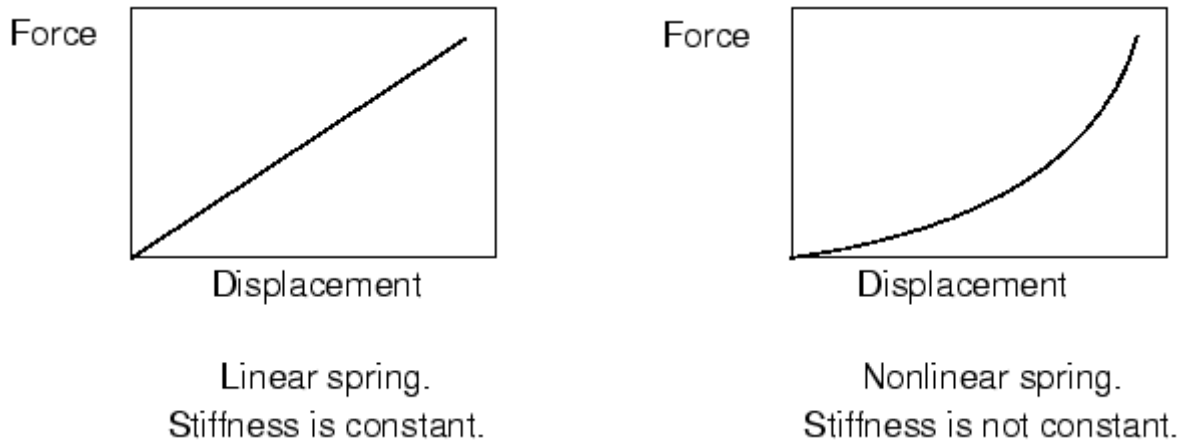


Figure 2.1: Linear and non-linear characteristics of a spring

In this project, the non-linearity lies in the contact modelling between the beam and the hammer which can be done by preventing or eliminating the penetration of the parts, these methods are called the kinematic and the penalty method respectively.

2.5 Modal Analysis

The natural frequency and different mode shapes of the system is done by modal analysis. The modal behaviour of the system is mainly affected by the boundary conditions and material properties of the system, So it is vital to do modal analysis for a system. The structural dynamic equation is given by,

$$M\ddot{u} + Ku = 0 \quad (2.4)$$

The eigenfrequencies are extracted for the first three eigenmodes of the system in ABAQUS/Standard for all the material pair combinations and it is needed for the frequency response analysis or the steady state dynamics, explained in the next section 2.6.

For this, we disable the shaker connector to let the beam vibrate freely. The first three eigenfrequencies obtained are 32.408 Hz, 104.91 Hz and 225.83 Hz. This is required as a preceding step for the steady state dynamics modal method analysis explained in the next section 2.6.

2.6 Frequency response analysis

In order to find the response of the system based on the harmonic excitation steady state dynamic analysis is used. It is a specialized type of transient response analysis that is extremely efficient to solve a very specific type of model. There are two methods used in the steady state dynamic analysis one is direct solution based and the other is mode based analysis([2]). The description of the direct solution based analysis and the mode based analysis is shown below.

The direct solution method is done as a frequency sweep by applying loads at different frequencies and recording the response of the system, here the solution is calculated directly from physical DOFs of the model using the mass, stiffness and damping matrices of the system[1].

The mode-based method is also a linear perturbation procedure similar to the methods above calculating the response using the system's eigen frequencies and modes, therefore requires a eigen frequency extraction performed before the analysis, is computationally cheaper than the other two methods but less accurate when damping is involved. It is capable of biasing the excitation frequencies toward the values that generate a response peak.

The primary difference between direct solution and the mode-based is the unknown variables, where the former considers the nodal displacements as variables which might result in large system of equations while the latter considers model coefficients or general coordinates as the unknown variables, which is the equal to the number of eigenmodes used in the simulation.

The excitation frequency that is applied to the signal is given in the form of frequency spread as in ABAQUS module. The steps that are involved in the frequency spread is shown below.

- Lower limit of frequency range or a single frequency, in cycles/time or Hz, which is calculated as (1-spread) multiplied by the frequency of the respective material pair model shown in 2.2.
- Upper limit of frequency range, in Hz or cycles/time. The calculation for the upper frequency limit is (1+spread) multiplied by the frequency of the respective material pair model shown in 2.2..
- Number of equally spaced points around the eigenfrequency at which results should be given, including the eigenfrequency and end points. 5 calculation points are used for the analysis.
- Bias parameter. If the value given is different than 1.0, the bias parameter is reset to 1.0. Further discussed below.
- Frequency scale factor. All the frequency points are scaled by this factor. The default frequency scale factor of 1 is used.
- Frequency spread. Determines the spread as a fractional value of each eigenfrequency in the specified range. The frequency spread of 0.1 is selected.

The frequency spread interval used for the steady state dynamics direct solution analysis is shown in the figure 2.2

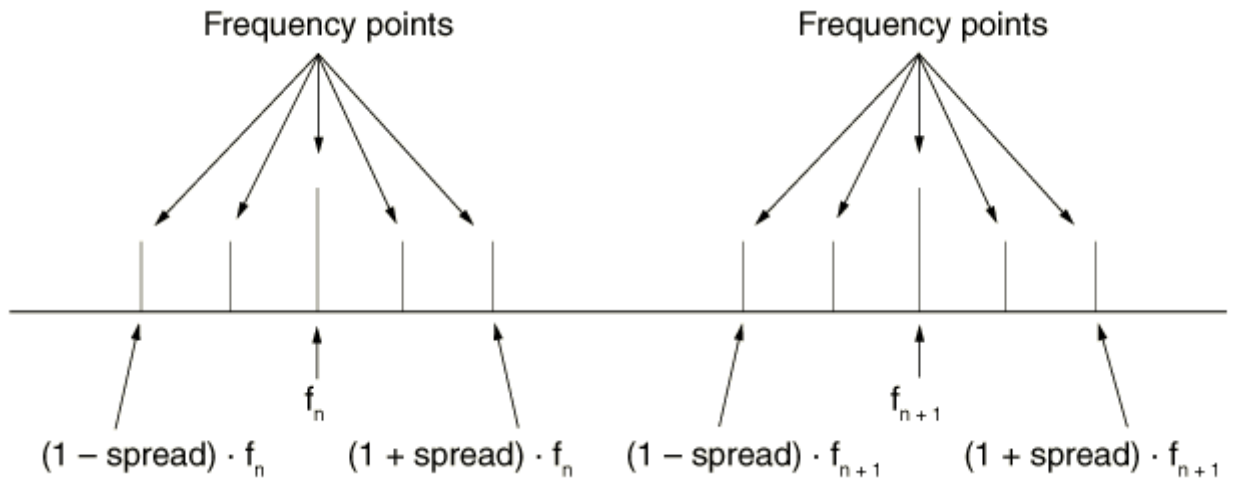


Figure 2.2: Steady state dynamics frequency spread interval

The load that is used for the excitation is the sinusoidal load taken from the experiment. It can be given either as a base motion or displacement boundary condition based on the analysis. The significant advantage of frequency response analysis is in the selection of frequencies where one component of the spectrum is extracted and the corresponding system response at that frequency can be measured with great precision.

2.7 Impact analysis

The rattle event is a normal impact of two surfaces whereas squeak sound is developed when two parts slide against each other, both are very unpleasant sounds in a car.

A physical impact analysis setup is expensive and if the structure fails, it is difficult to determine the cause of failure and what design changes need to be made in order to rectify it as there is very limited information, and therefore virtual simulations (CAE) plays an important role in these situations where the experimental setup is modelled in the pre-processing software ANSA. In impact analysis various parameters have been used for correlation such as reaction force, velocity before and after impact, acceleration before and after impact.

2.8 Material Damping

Rayleigh damping is used for the beam material. In this, it is assumed that the damping is distributed as a combination of mass and stiffness.

$$C = \alpha M + \beta K \quad (2.5)$$

where

α is the mass proportional Rayleigh damping coefficient

β is the stiffness proportional Rayleigh damping coefficient

M is the system structural mass matrix

K is the system structural stiffness matrix

This results in different ratios for different response frequencies,

$$\xi = \frac{1}{2} \left(\frac{\alpha}{\omega} + \beta\omega \right) \quad (2.6)$$

where

ξ is the damping ratio (a value of 1 corresponds to critical damping)

ω is the response frequency in rad/s.

From the equation, it is seen that the stiffness proportional term gives the damping ratio linearly proportional to response frequency and the mass proportional term gives the damping ratio inversely proportional to response frequency response.

2.9 Time Domain Analysis

In this project work, time domain analysis is carried out for most of the simulations, the excitation is done based on the frequency response and it is converted into time domain by using FFT (Fast Fourier transform) function which integrates the frequency domain to time domain[3]. Most of the measurements are taken based on the response of the system at the specific time and it is used for correlation. The parameters used for correlation and the results obtained by both experiment and simulation for time domain system is seen in the methods and the results section below.

3

Methods

This section describes the methods used in the project work, such as experimental setup, mesh, gap size determination, contact analysis, boundary condition, initial conditions, excitation signal, omega arithmetic, material pairs and properties, renumbering of nodes, design of experiments, FE-model setup, model validation and correlation.

3.1 Physical Rattle setup

The physical test setup consists of a rattle test rig. It consists of a cantilever beam with cross section in rectangular shape. The electrodynamic shaker is connected to a beam, the beam is also connected to a cylindrical connecting rod and is fastened to a shaker. One end of the beam is clamped with bolts to a stand, While the other end is drilled and a bolt is placed in it, A material sample is attached to a bolt side by using glue. Similarly another material sample is attached to a bolt on the other end at hammer side. The gap size between the two material sample varies between 2mm to 10mm. Different types of material samples are used in this project, These material samples can be changed by removing the bolts. The physical setup of an experimental rattle setup is shown in the figure 3.1

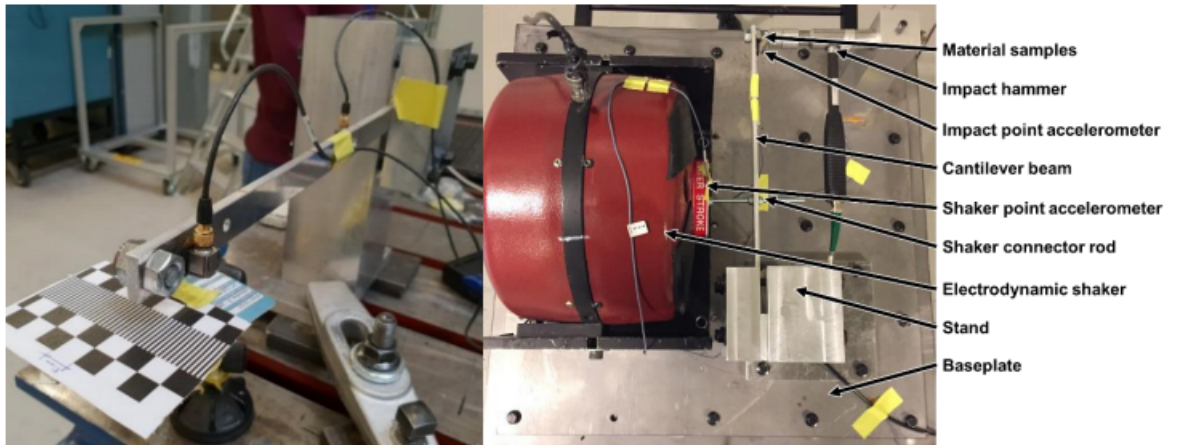


Figure 3.1: Squeak and rattle visualization

The materials used and their dimensions are shown in the table 3.1.

Part	Material	Dimensions
Cantilever beam	Aluminium 6061	320x20x6 mm
Shaker Connector Rod	steel grade 8.8	M6 diameter, 129mm length

Table 3.1: Name and dimensions for the experimental parts

Two different analysis have been used in this experimental setup

- Modal analysis was performed in this setup to determine the natural frequency of the system. The shaker rod is disconnected and the modal analysis is done only for the beam section. Only acceleration is detected from this experiment.
- The important aspect of this experiment is used to capture the rattle noise. The rattle noise is detected at the impact points between the hammer section and beam section. The load is given to the beam section by electrodynamic shaker through connector rod. This makes the beam to vibrate and causes the impact between two material pairs. During rattle analysis, important parameters such as displacement, velocity, acceleration and reaction force are captured.

The experiments are done in a semi-anechoic chamber at room temperature. The parameters required are evaluated from two accelerometers (Bruel & Kjer Type 4524B triaxial CCLD piezoelectric accelerometer) placed in the beam. One accelerometer is glued at the cantilever beam near to the bolt section and the other is glued at the tip of the connecting rod. The accelerometers are labelled in the figure 3.1. The accelerometer near to the bolt section is called as impact point accelerometer, and the accelerometer near to connecting rod is called as shaker point accelerometer.

3.2 Mesh

The finite element model for the rattle setup developed in ANSA software is shown in the figure 3.2. The finite element model consists of first order shell, solid and beam elements. The element size varies from 1mm to 2mm it is selected based on the previous analysis in NVH field. The model consists of 21 1-D beam elements, 1040 2-D shell elements and 2742 3-D solid elements. The elements used for different modelling parts is described below.

- The beam in the rattle setup is modelled fully with 2d shell elements(S3, S4), using this elements reduces the computational time with no change in accuracy.
- The bolt and nut pair holding the samples and the samples itself are modelled with 3D brick (C3D8) and 3D penta (C3D6) elements.
- The connecting rod that connects the beam to the shaker is modelled with 1D beam elements.

- Small parts with negligible geometry distribution such as accelerometers and nuts of the connector rod are modelled with point masses. The equivalent mass value are based on the measurements and they are validated on the previous study.
- The connection between the shaker connector rod and the beam was modelled with rigid elements. The mass points representing the accelerometer are connected with MPC (Multi Point Constraint).

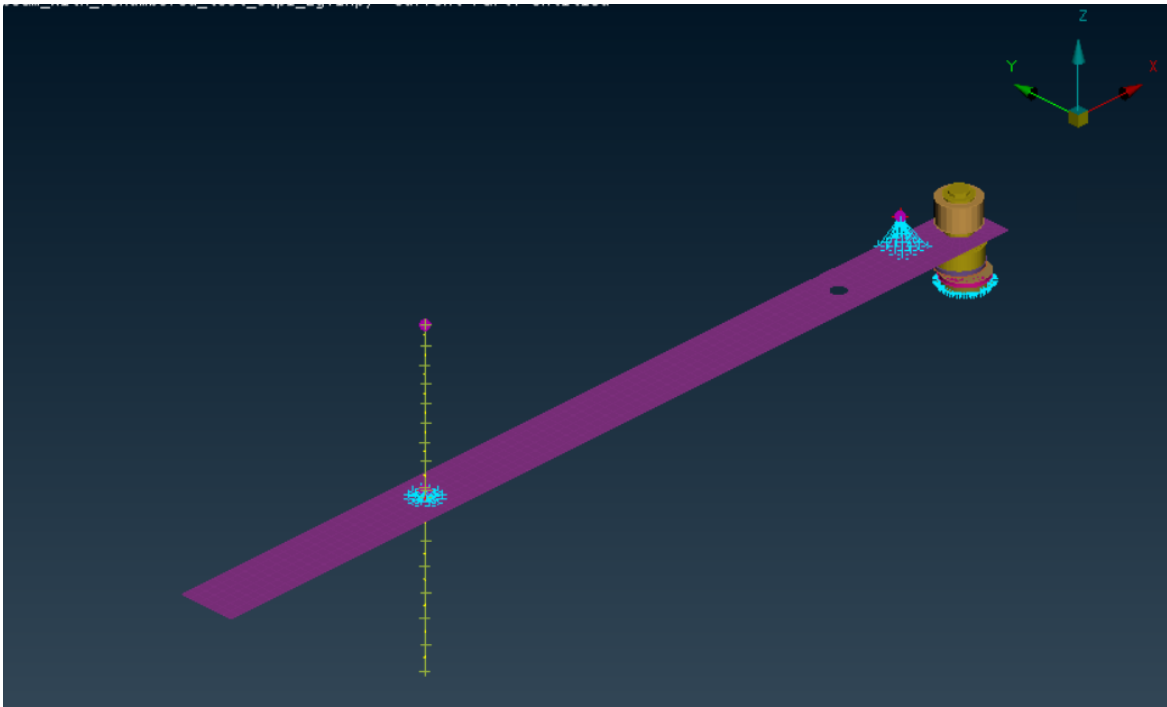


Figure 3.2: FE model of the Rattle setup

3.3 Boundary Conditions

The boundary conditions for the FE model was defined according to the experimental setup. The boundary conditions used in the model are shown in the figure 3.3.

- The free end of the beam near to the shaker connecting rod is constrained in all 6 directions, since it was rigidly clamped in the test setup.
- The excitation signal that passes from the shaker to the end of the connecting rod is constrained in all directions except Z. The Z direction has the prescribed displacement due to the signal from the shaker.
- The nodes of the counterpart sample were constrained in all 3 DOF (solid elements). The effects due to elasticity from the hammer support was investigated during previous study and it was neglected due to its minor effect on

the results.

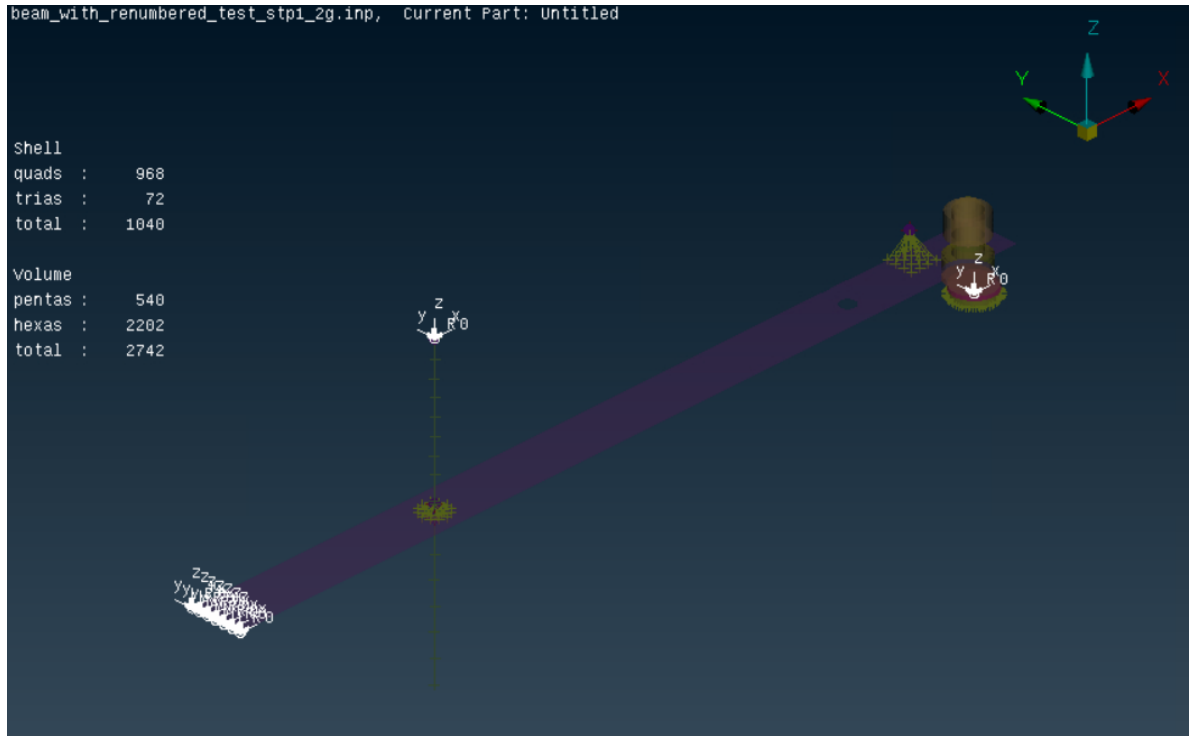


Figure 3.3: Boundary conditions of the FE model

3.4 Signal processing

In the experiment, the accelerometers used at the shaker region and impact region are used to evaluate the output parameters. Acceleration is detected from the accelerometers and integrated to displacement and velocity for the shaker region and impact region. As the analysis is done mainly in time domain, displacement plays a vital role for the correlation part.

Due to the integrated signals there might be a drift and noises occurring in both velocity and displacement from the measurement data. These drifts and noises are needed to be eliminated from the signal. Those noises and drift in the signal need to be filtered without affecting the effect on the results[8].

Fast Fourier Transform has been used as a transfer function in this work to filter the data. There are several methods that have been used for filtering such as Mean removal, Butterworth filter, Moving average filtration and Omega Arithmetic. In this project, omega arithmetic has been used for filtration is discussed in the next section. The main advantages of the signal processing is discussed below:

- Mainly eliminate noises and drift
- The curve shapes are improved.

3.5 Omega Arithmetic

Omega arithmetic is a filtration technique used to eliminate the noises and drift in the signal. It work on the basis of FFT algorithm which integrate the signal to frequency domain. As we are working in the time domain, Inverse FFT algorithm is used to filter the signal in time domain.

The omega filtration is mainly used because of its freedom in filtering. The FFT algorithm used will divide the signal into frequency components, so it will be easier to altered or eliminate each of these components. The alteration or elimination of a component does not have the effect on the neighbouring component or it causes a phase shift in the inversed signal.

Certain frequency ranges have been tried to eliminate the noise and drift in the signal. Finally after several runs, it is found that for low frequency 0 Hz to 8 Hz is used and for higher frequency 28 to 60 Hz is used. It is seen that for these frequency ranges the important data of the impact events does not change when eliminating the drift and the noise. The Matlab code used for omega arithmetic is seen in the appendix and for further details regarding the omega arithmetic is seen in the reference.

From the figures 3.4 and 3.5 it is clearly seen that there is a drift in the signal when omega arithmetic is not used.

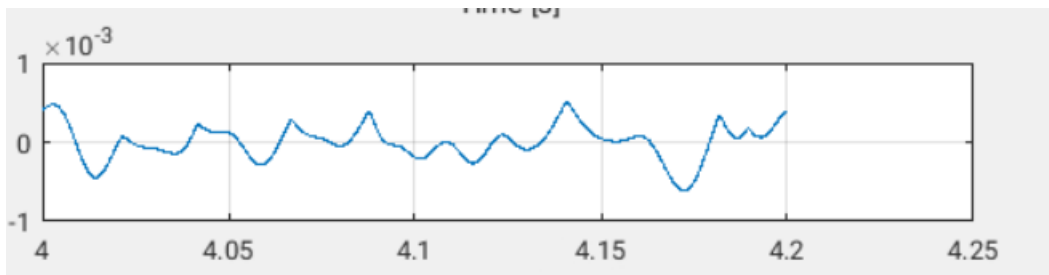


Figure 3.4: Signal obtained by using omega arithmetic

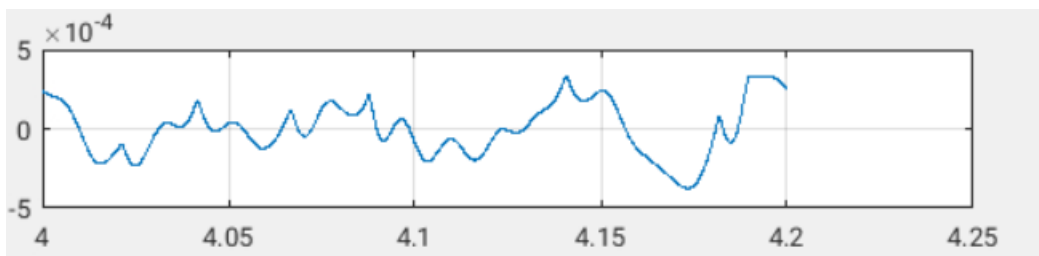


Figure 3.5: Signal without using omega arithmetic

3.6 Material Properties

The material properties used for the different material pairs such as Steel-Steel, Steel-Plastic p1, plastic p2 - plastic p1, Aluminum-aluminum and different gap sizes such as 2 mm, 5 mm and 10 mm between the sample beam and the sample hammer is shown in the table 3.2 and table 3.3. These material properties are changed in the simulation depending on the material pairs used for the sample beam and sample hammer.

The steel grade used for the combination has a Young's modulus of 210000 MPa, density of 7.85×10^{-9} tonnes/mm³, Poisson's ratio of 0.3, mass of the sample beam of 0.17 kg and thickness of the beam is 0.01 mm.

The plastic p1 material used for the sample hammer has a Young's modulus of 1850 MPa, density of 1.02×10^{-9} tonnes/mm³, Poisson's ratio of 0.42, mass of the sample beam of 0.165 kg and thickness of the hammer is 0.01 mm.

The plastic p2 material used for the sample beam has a Young's modulus of 1400 MPa, density of 1.11×10^{-9} tonnes/mm³, Poisson's ratio of 0.4, mass of the sample beam of 0.166 kg and thickness of the beam is 0.01 mm.

The Aluminum material used for the sample beam has a Young's modulus of 68900 MPa, density of 2.609×10^{-9} tonnes/mm³, Poisson's ratio of 0.33, mass of the sample beam of 0.166 kg and thickness of the beam is 0.01 mm.

	Sample Beam					
Material pair	Material	Young's Modulus [MPa]	Density [t/mm ³]	Poisson's ratio	mass [kg]	sample thickness [mm]
stst 2mm	Steel	210000	7.85E-09	0.3	0.17	1.6
. stst 5mm	Steel	210000	7.85E-09	0.3	0.17	1.6
stst 10mm	Steel	210000	7.85E-09	0.3	0.17	1.6
stp1 2mm	Steel	210000	7.85E-09	0.3	0.17	1.6
stp1 5mm	Steel	210000	7.85E-09	0.3	0.17	0.01
stp1 10mm	Steel	210000	7.85E-09	0.3	0.17	0.01
p2p1 2mm	Plastic p2	1400	1.11E-09	0.4	0.166	0.01
p2p1 5mm	Plastic p2	1400	1.11E-09	0.4	0.166	0.01

Table 3.2: Material properties for sample beam

Sample hammer					
Material	Young's Modulus [MPa]	Density [t/mm ³]	Poisson's ratio	mass [kg]	sample thickness [mm]
Steel	210000	7.85E-09	0.3	0.17	0.01
Steel	210000	7.85E-09	0.3	0.17	0.01
Plastic p1	1850	1.02E-09	0.42	0.165	0.01
Plastic p1	1850	1.02E-09	0.42	0.165	0.01
Plastic p1	1850	1.02E-09	0.42	0.165	0.01
Plastic p1	1850	1.02E-09	0.42	0.165	0.01
Plastic p1	1850	1.02E-09	0.42	0.165	0.01

Table 3.3: Material properties for sample hammer

3.7 Contact definition

The contact definition for the model is kept the same from the previous thesis work, which is defined by four settings, the contact type, the contact surfaces, the mechanical constraint and the interaction.

The solver ABAQUS has two types of contact, one is the 'contact pair' which requires the user to select the surfaces which come in contact manually whereas the 'general contact' option allows the selection of an area or the whole model and helps locate the contact within the defined region.

A surface can appear in any number of contact pairs. Since the two surfaces which come in contact are known in our FEA model, we use the 'contact pair' option for the simulations.

Next, the “master” and “slave” in CAE, the two surfaces that come in contact, have to be defined. The master chosen can only be element-based(El), but the slave can be either element(El) or node-based(node). Using either of the two is dependent on the problem under investigation. A slave surface can be either element-based(El) or node-based(node), while the master can only be element-based(El). In this project the contact is defined as element-element(El-El) based for all the simulations.

The contact between the sample beam and the sample hammer is shown in figure 3.6 is a surface to surface contact pair.

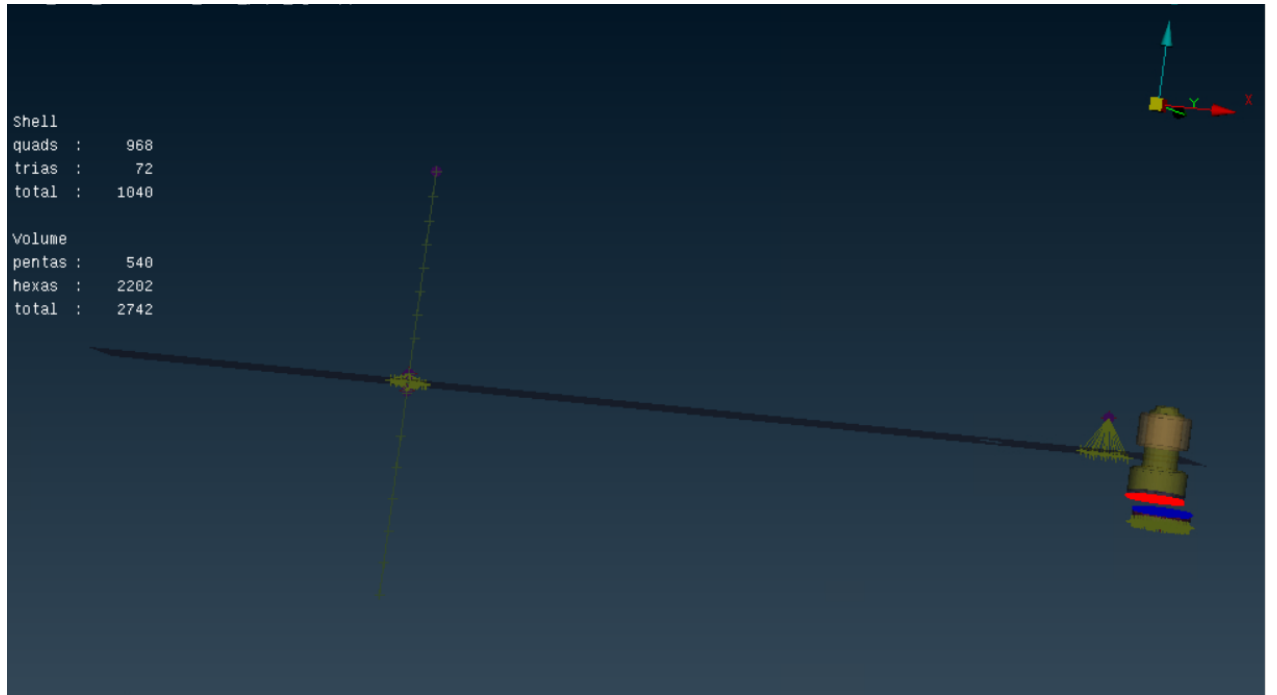


Figure 3.6: Contact between the sample beam (highlighted in red) and the sample hammer (highlighted in blue)

3.8 Excitation signal

The signal that can be used for excitation is shown in figure 3.7. This signal is generated from the experiment based on the time interval and the amplitude obtained in the sine wave before the impact. Matlab scripts are used to generate the signal by using the time interval and amplitudes obtained from the experiment. The generated signal is used as an input signal to the beam and the simulation is run for the length of the signal.

The excitation signal is given as input through the shaker point shown in figure 3.3 this makes the beam to vibrate and is used to check the contact happening between two material pairs. The signals are generated for different material pairs based on the experimental time and amplitude.

The signal curve is added in ANSA Pre-Processing as an amplitude for the ABAQUS/-Standard or ABAQUS/Explicit solver based on the input file and the simulation being run.

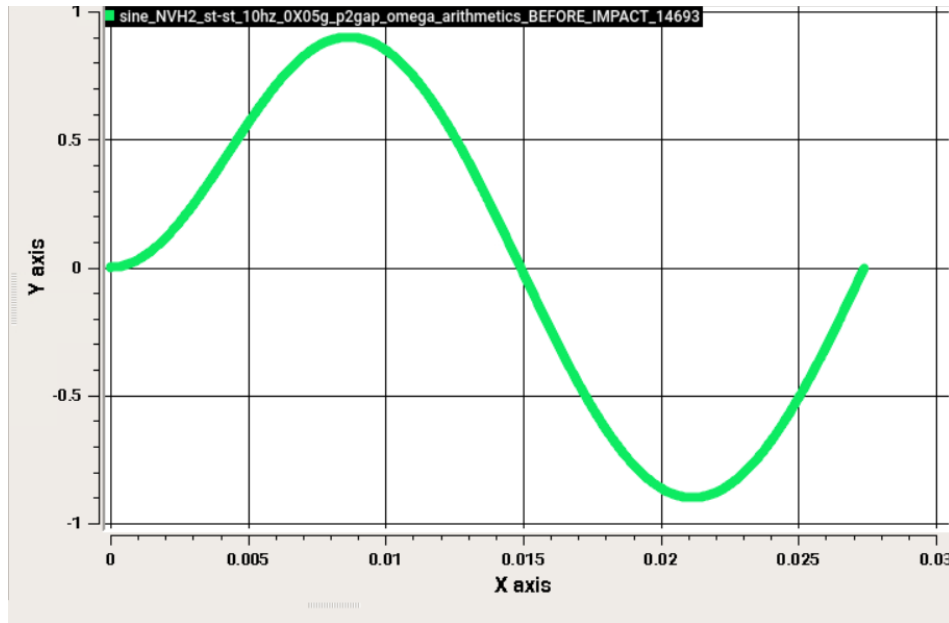


Figure 3.7: Smoothened Amplitude(y-axis) vs Time(x-axis) plot used as the input signal for the model

3.9 FE model setup for simulation

The FE model used for the thesis is that of the cantilever beam as shown in the physical rattle setup 3.1.

The simulation deck (Fe-model) is set up in ANSA pre-processing involves certain steps that need to be done in order to setup, which include the changing of material pairs and their properties, changing node numbers and creating new node sets to make it easier for ramping the velocity which is the initial conditions which is edited in the input file for each of the simulations. The time step is changed according to the length of the input signal for the different sets of simulations gathered from the experiment.

The linear and non-linear analysis for the model are run in different solvers based on the simulation, ABAQUS/Standard and ABAQUS/Explicit is used for the implicit and explicit simulations respectively.

The results are then checked in META post using an automated session script which outputs the reaction force, displacement, velocity and the accelerations as a text file which are the desired outputs for the simulation needed for the correlation and the plots are generated in MATLAB using a script to check for the contact in the non-linear simulation and the correlation with experimental results which is explained in the section 3.14.

3.10 Initial Conditions

The signals used for excitation is already processed, but it cannot be applied directly to the FE model as a boundary condition, since it would result in a jerk because part of the signal would produce either an initial velocity or displacement at the start of the curve, To overcome this, initial velocities are used, two points are taken from the experiment, one is the impact velocity and the other is the shaker velocity, the values are ramped from impact to the shaker based on the number of nodes using a MATLAB script. These values are then used as initial conditions in the beam. The initial conditions are used in the beam by splitting the beam into 4 different parts, start region, shaker region, middle region and impact region.

The nodes are renumbered from 10001 to 10093 to have the proper order to use the initial conditions in the beam. The node numbers from 10001 to 10026 is used as a start region shown in figure 3.8 where the velocities are used as ramped from 0 to shaker velocity from the experiment. The node numbers from 10027 to 10033 is used as a shaker region is shown in figure 3.9 where the shaker velocity obtained from the experiment is used for all the nodes. The node numbers from 10034 to 10093 is used as a middle region where the velocities are ramped from shaker value to impact value is shown in figure 3.10. The node numbers from 10094 to 10097 is used as an impact where the impact velocity value is used for the whole region as shown in figure 3.11. After the impact region the values are interpolated by multiplying with the scale factor of 5 and used for the entire region after impact. The output nodes selected to obtain the desired outputs such as the reaction force, displacement, velocity and the acceleration are chosen in the beam, hammer, impact point accelerometer and the shaker point accelerometer as seen in 3.12.

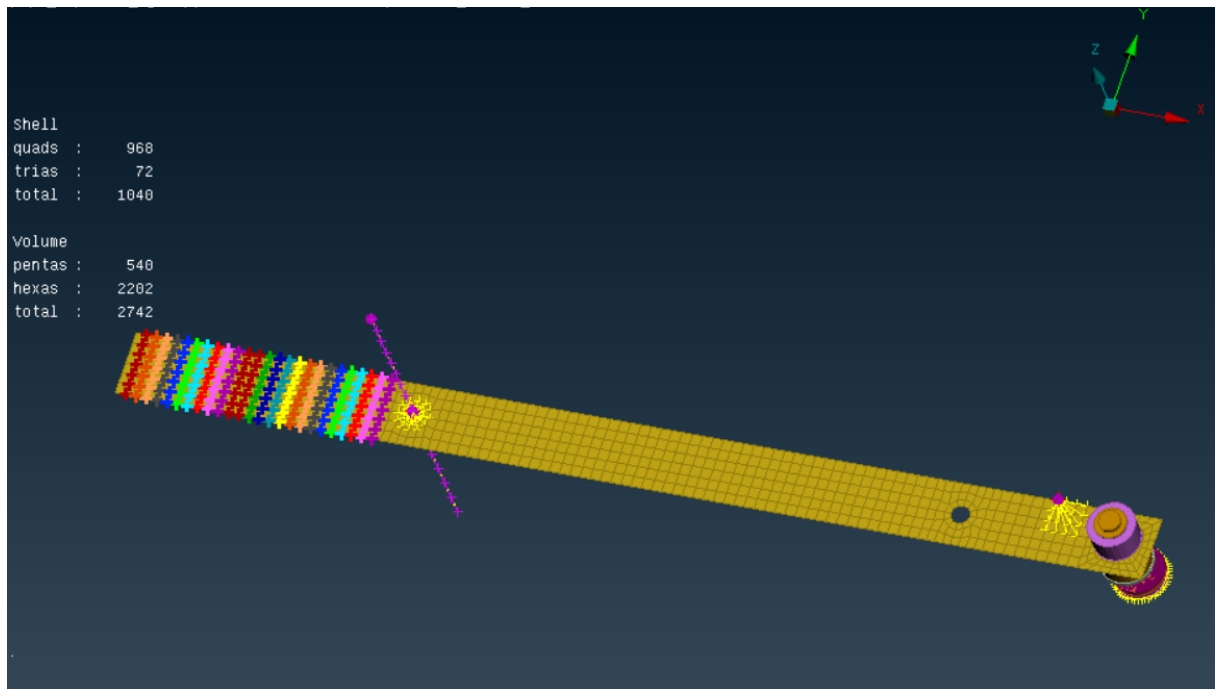


Figure 3.8: Node set 2-26 from the clamped end

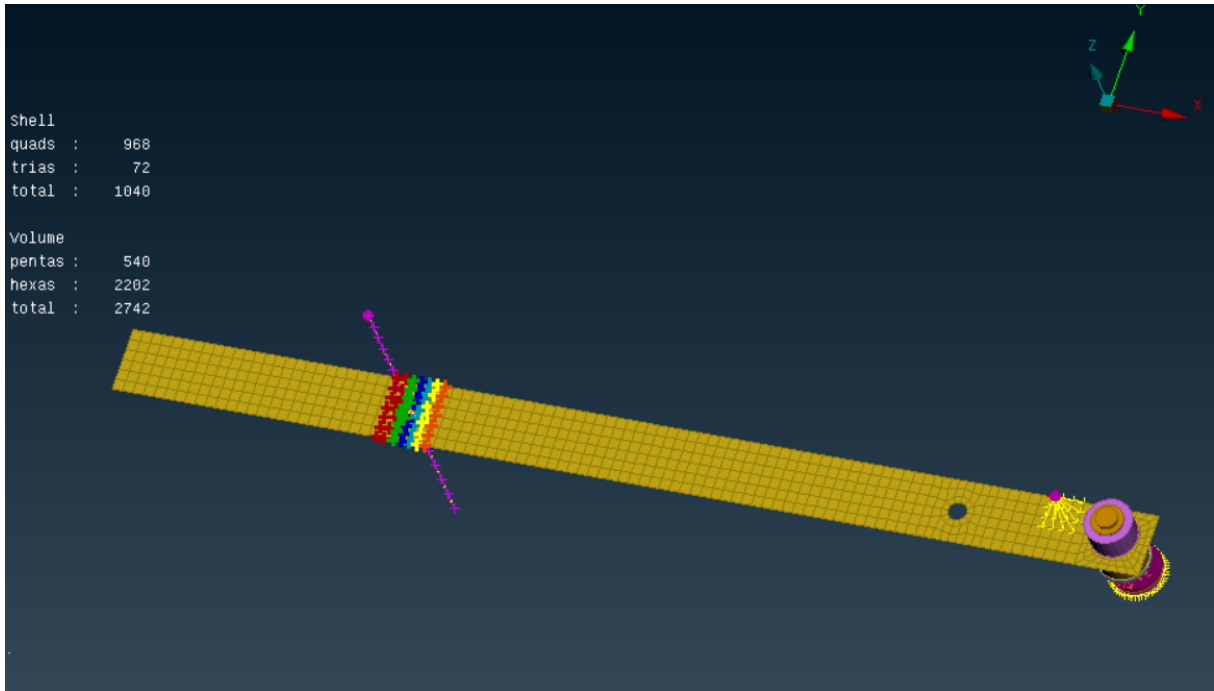


Figure 3.9: Node set 27-33 near the shaker connector hole area

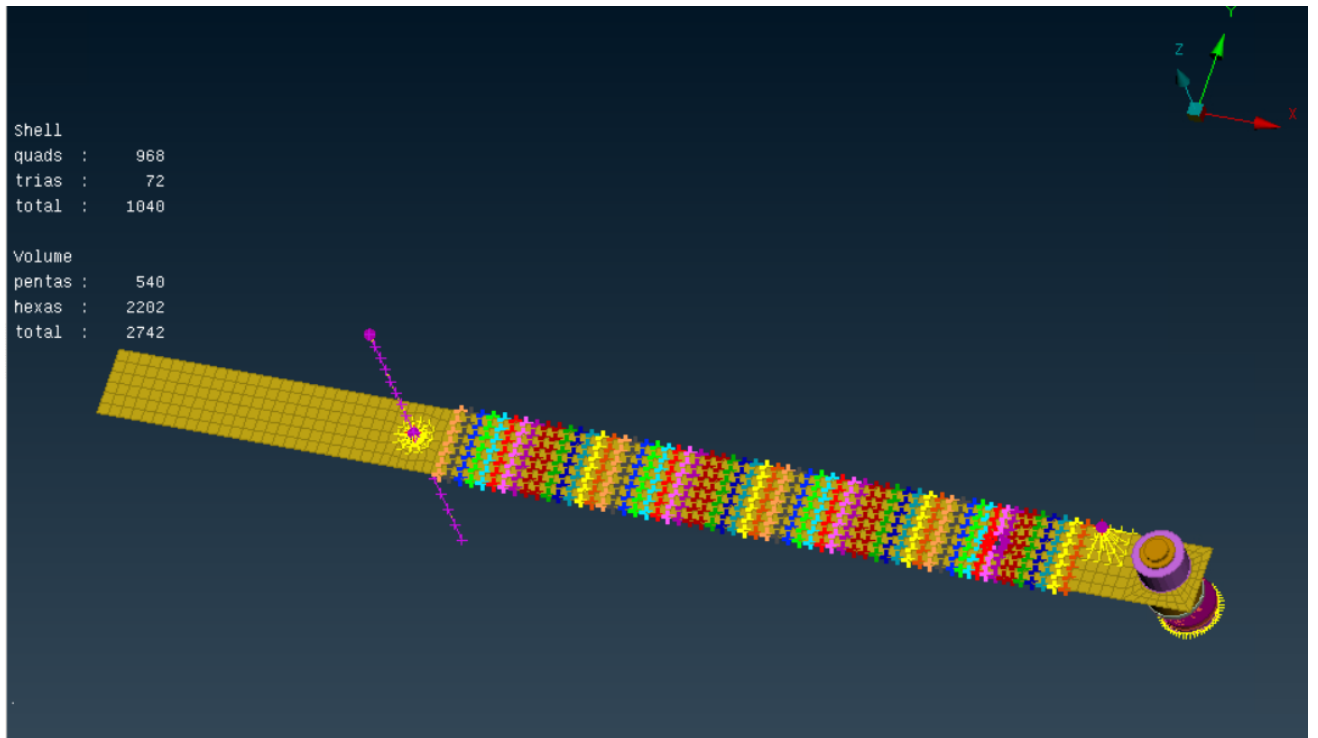


Figure 3.10: Node set 34-93 after the hole area till the impact area

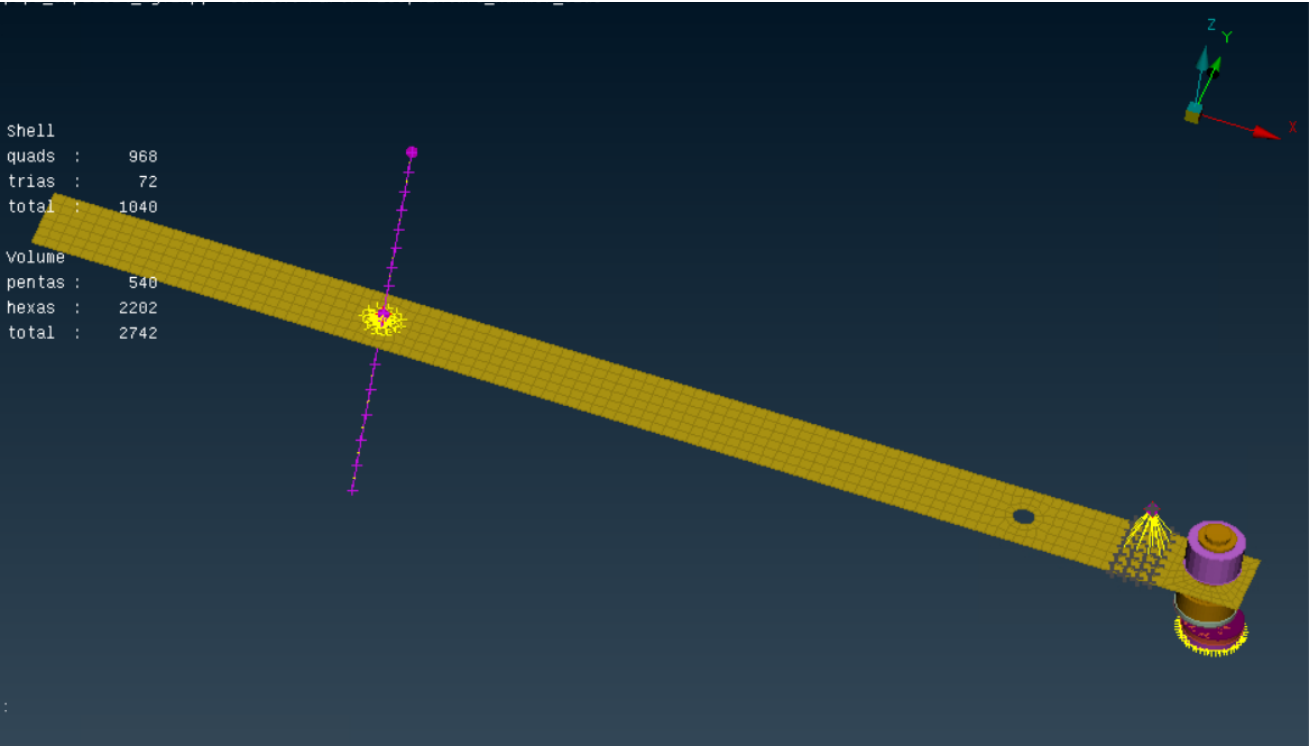


Figure 3.11: Node set for the impact accelerometer area

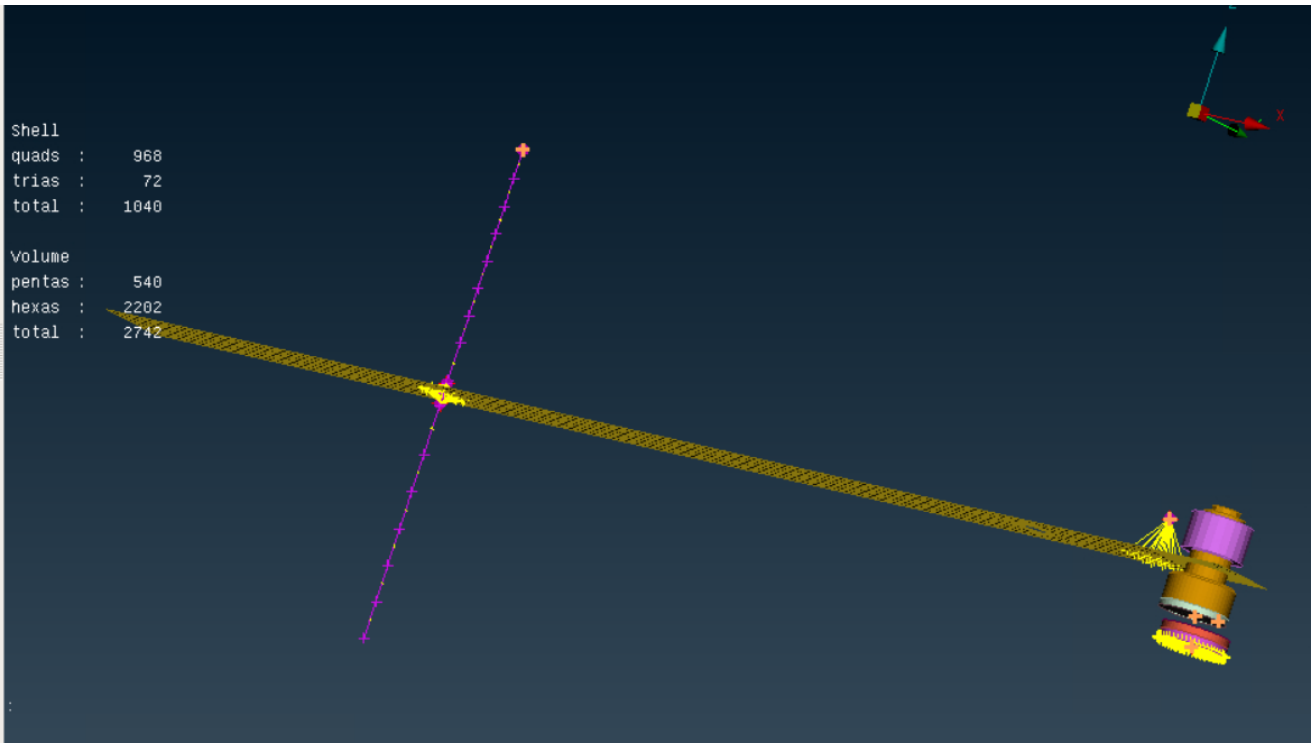


Figure 3.12: Nodes selected for measuring the output parameters

The values obtained from the simulation for all the parameters are noted from the impact accelerometer placed at the impact region. The important output nodes and

their information is shown in the table 3.4.

Node number	Usage
1496	Shaker excitation signal
9510	Calculates the required parameters from the accelerometer
19	Detects the contact between beam and hammer
77089	Used to detect the penetration between beam and hammer
2889	Checks whether the signal excited the beam

Table 3.4: Important output nodes and its usages

3.11 Gapsize Determination

The main aim of the correlation is to simulate all the parameters based on the experiment. The gap size between the sample beam and the sample hammer is obtained from the experiment but the gap size used does not have desired precision. The displacement values are taken from the experiment based on the amplitude it reaches the impact point, which is used as the gap between material samples for the simulation.

The determined gap size differs based on the material pairs and the experimental signal, number of simulations are run in order to achieve the displacement to correlate with experiment with model tuning explained in section 3.14.1. The parameters that are used to change the setup in the simulation was mainly the magnitude of the amplitude.

In the experiment, impact at positive z-direction is considered as closing the gap whereas in simulation the negative Z-direction is closing the gap between two material pairs. In order to correlate both simulation and experiment, signs are changed in the experimental values.

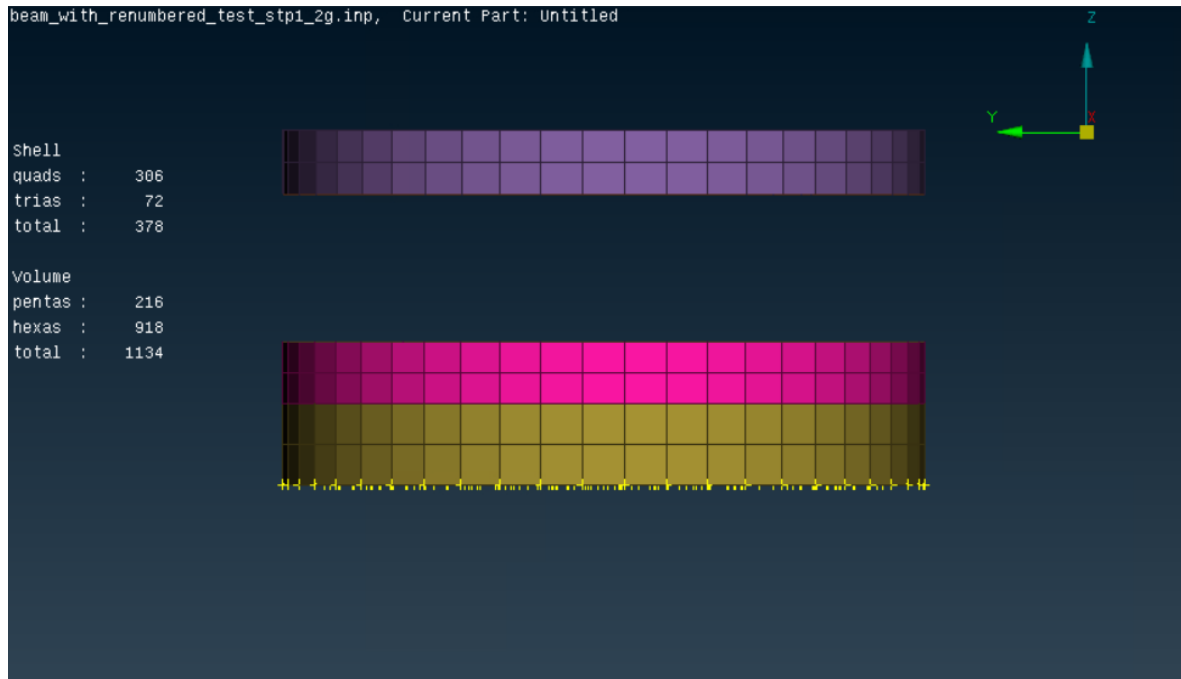


Figure 3.13: Gap between beam and the hammer

3.12 Implicit and Explicit solvers

In static analysis, there is no effect of mass (inertia) or of damping. In dynamic analysis, nodal forces associated with mass/inertia and damping are included.

The implicit solver uses iterations to establish equilibrium within a certain tolerance to give the solution for each step with respect to the loading or displacement applied. The unknown displacements are solved through matrix inversion and when the problem is non-linear in terms of contact or material or the geometry, inverting the matrix becomes computationally expensive or sometimes impossible for large models as the solution is derived through small time increments and the current time step considers the value from the previous time step and goes on. This means that each increment has to converge and if the solution is found, it is unconditionally stable and accepts large time steps to be solved at once. Implicit transient analysis has no inherent limit on the size of the time step. As such, implicit time steps are generally several orders of magnitude larger than explicit time steps([8]).

In the explicit solver the nodal accelerations are solved directly as the inverse of the diagonal mass matrix times the net force which includes the contact, damping, element stress, etc and therefore no iterations are required. Here, the kinematic conditions for one small time step is used to calculate the conditions for the next and the end of increment considers the displacement, velocity and the accelerations at the beginning of the same time increment and therefore has no relevance to the previous value and hence no convergence issue. The time step in explicit analysis must be less than the Courant time step (time it takes a sound wave to travel across

an element).

3.13 Design of Experiments

A design of experiments (DOE) is setup for the rattle predictions using the linear and non-linear FE impact simulations for the mechanical system considered.

The study mainly focuses on how rattle sounds affect the quality of the car and this problem is addressed by comparing the linear and non-linear virtual analysis to the experimental data collected from the physical rattle setup.

The experiment is designed by considering different material pairs like Steel-plastic p1 (St-p1), Plastic p2 - plastic p1 (p2-p1), Aluminum-Aluminum (Al-Al) and for different gap sizes of 0 mm, 2 mm and 5 mm respectively with the different outputs like the displacements between the sample beam and the sample hammer, velocity at zero displacement, accelerations before impact and the impact force or the contact force required in order to correlate with the simulation results. The choice of the solver is based on the material pair used and if the dynamic problem has low velocity or high velocity from the experiment.

There are 2 events selected for each material pair from the experiment .

In Event 1, considering a value from the average impact force range, meaning the high occurrence of these forces which is slightly on the lower side and the start and end time adjusted to capture the event with contact bumps which are evident in the displacement plots and the velocity and acceleration values just before impact and the peak after impact is noted with a priority set to the choice of solver that is implicit or explicit, based on these values and this event is considered as the first in the priority list.

In Event 2, the impact force values chosen are on the higher side which are less occurring and are evident in the signal and the start and end time for this event is noted along with the velocity and acceleration values just before impact and the peak after impact. This event is considered as second on the priority list.

The table 3.5 below shows the important events selected based on their material combinations and the gap sizes and the solver used to run the simulations. Most of the results are achieved in the implicit solver whereas explicit solver is used for some cases to check the contact bumps that occur during the simulation.

Material pair	Gap Size	Priority	Solver
St-P1	0mm	1	Implicit/Explicit
P2-P1	0mm	1	Implicit/Explicit
Al-Al	5mm	1	Implicit/Explicit

Table 3.5: Material pairs for simulation with priority

The correlation for the implicit and the explicit simulations for the non-linear and linear cases are discussed in the results section 4.1.

3.14 Model Correlation

The model is evaluated for the output structural dynamics parameters by comparing the simulation results obtained with the experimental data gathered previously and the calculated risk severity numbers.

The correlation done in the thesis work for the parameters such as contact force at impact, displacement of the beam at impact, velocity and the acceleration values before impact from the rattle impact experiment and the simulation results are extracted using the ABAQUS/Standard solver and the risk values are compared.

The experimental results are plotted using a Matlab script with varying frequency bandwidths within the sampling frequency at different time durations for respective material pairs with the varying gap sizes. This data is collected for the a lower frequency range of 0 - 8 Hz and a higher bandwidth of 28 - 40 Hz which gave data with less noise and better contact bumps.

The reaction force is recorded first for a certain time which is used to record the other data such as the impact velocity before and after impact, acceleration before and after impact, velocity at the start, that is, at zero displacement and the displacements before and after impact with delta (value at the highest peak minus the value at the lowest trough) also being recorded. The shaker velocities, accelerations and the displacements are recorded too with the values of the sine curve used from the shaker displacement used to create the input signal for the simulation setup.

The rattle event is predicted using both the linear and non-linear analysis. The non-linear simulations are done with a contact between the sample beam and the sample hammer and the linear simulations are done without contact.

A frequency domain response analysis is also simulated to get the relative velocities and accelerations. First, the eigenfrequencies are extracted from modal analysis and a frequency response analysis , also called steady state dynamics explained in section 2.6 is done by giving the amplitude in terms of frequency as a displacement boundary condition used for the linear and non-linear analysis. The results are evaluated for different parameters and described in the results section.

3.14.1 Model tuning

In order to compare the results from the simulations to the experimental data, the model needs to be tuned to get the desired closeness to the risk severity parameters like the reaction forces at impact, velocity just before impact, velocity after impact,

acceleration before impact, acceleration after impact, displacements.

Model tuning plays a vital role in this correlation study. The main parameter used for correlation is the magnitude of the amplitude. The amplitude parameter is tuned to get the reaction force that is closer to the experiment. The time at the reaction force is then used for taking out the parameters to correlate such as velocity and acceleration before impact and after impact.

3.15 Linear Analysis

As discussed in the previous section the linear simulation is done for the same model as shown in figure 3.2 is done without contact between the sample beam and the hammer. During this analysis the displacement due to penetrations, the velocity and acceleration before impact and the relative maximum displacement and velocity is taken from the simulation and it is then compared with the results obtained from the non-linear analysis. In linear analysis there is no option of getting a reaction force as there is no contact between the sample beam and hammer. The simulations are done for some of the material pairs and the results are described in Chapter 4.

4

Results & Discussions

This section gives the complete results obtained during the thesis work, the main thing is to correlate the structural dynamic parameters with experiment and simulations. Two solvers are used for this study one is Implicit and the other is Explicit. The results obtained for different signals for non-linear simulations are shown below:

4.1 St-P1 Material Pair - Implicit

This section shows the results for St-P1 material pair for non linear simulation using the implicit solver where the material pair steel is placed on the beam side and plastic is placed on the hammer side.

4.1.1 Experimental Results

Figure 4.1 shows the experimental data captured for the St-P1 material pair with 0 mm gap size. The values extracted from the experiment are used for correlation with the simulated values.

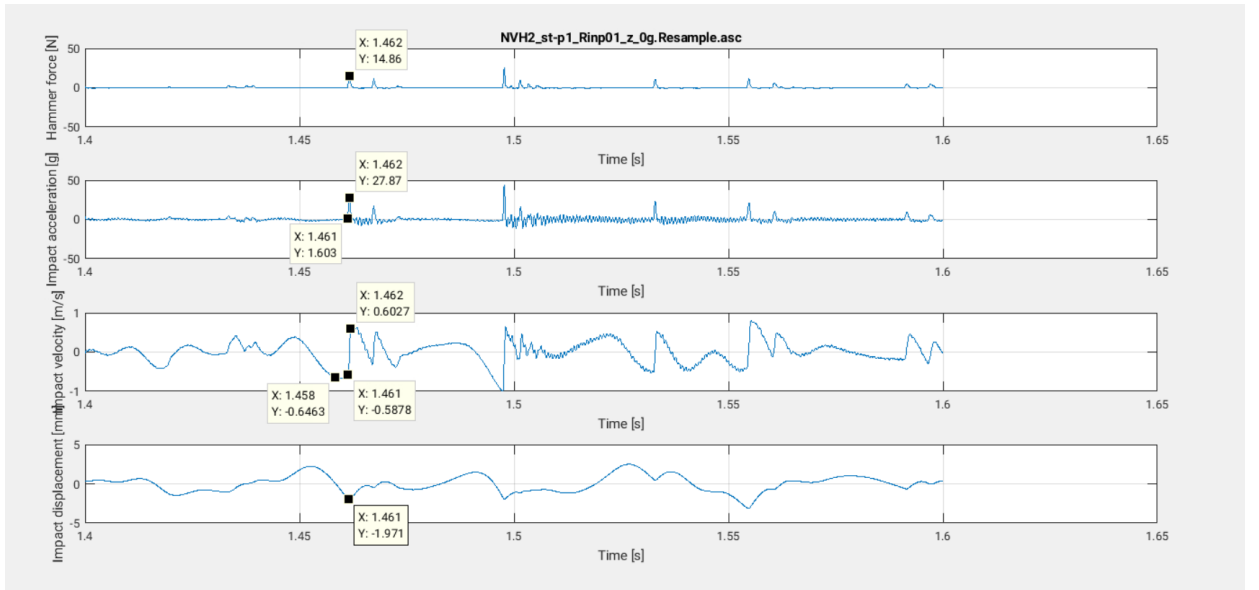


Figure 4.1: Experimental data for St-P1 material pair with 0 mm gap size which includes the reaction force(N), velocity(m/s) and acceleration(g) and displacement(mm) in z-direction Vs time(s)

4.1.2 Simulation results - Implicit Solver

Figure 4.2 and 4.3 shows the simulated results for St-P1 material pair with 0mm gap size using Implicit solver.

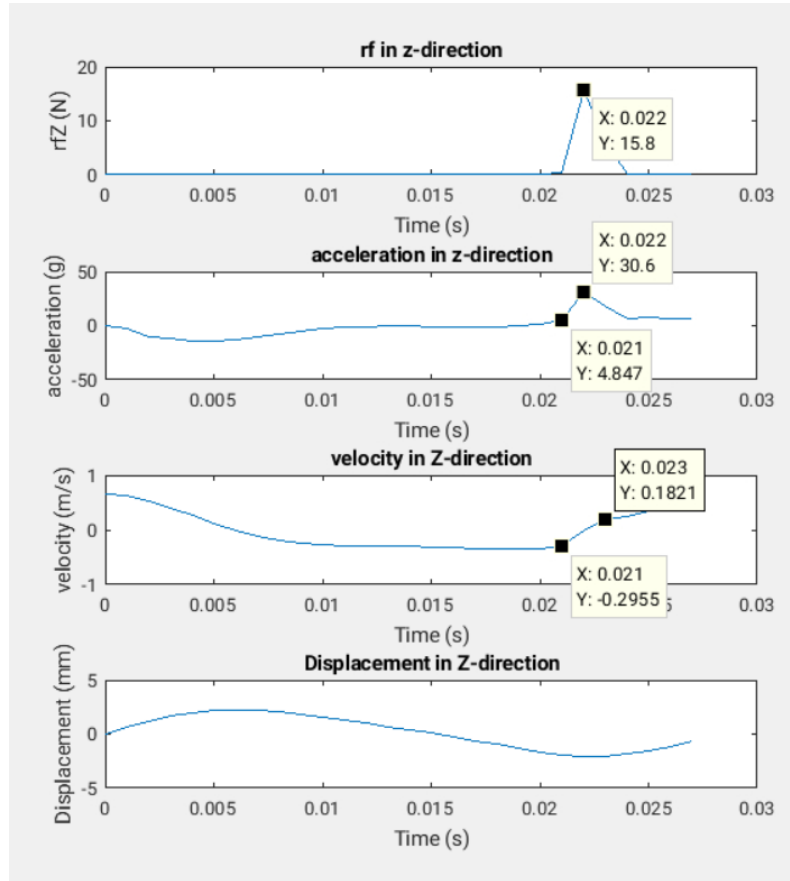


Figure 4.2: Simulation data for St-P1 using Implicit Solver for reaction force(N), velocity(m/s) and acceleration(g) and displacement(mm) Vs time(s)

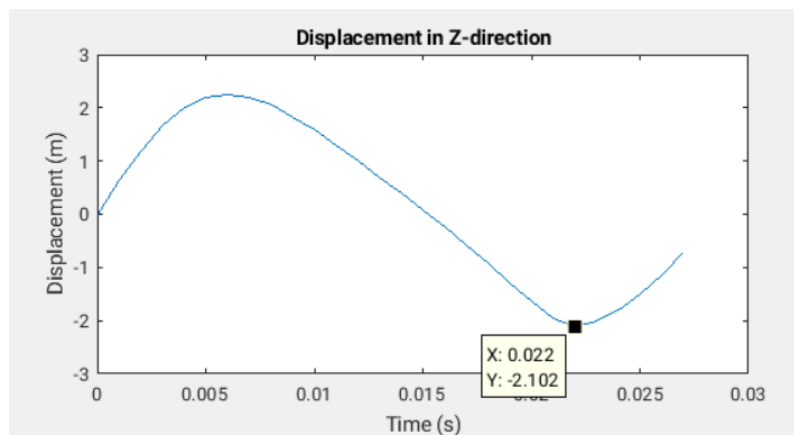


Figure 4.3: Displacement in z-direction(mm) Vs time(s) for St-P1 material pair using Implicit solver

4.1.3 Simulation results - Explicit Solver

Figure 4.4 and 4.5 shows the simulated results for St-P1 material pair with 0mm gap size using Implicit solver.

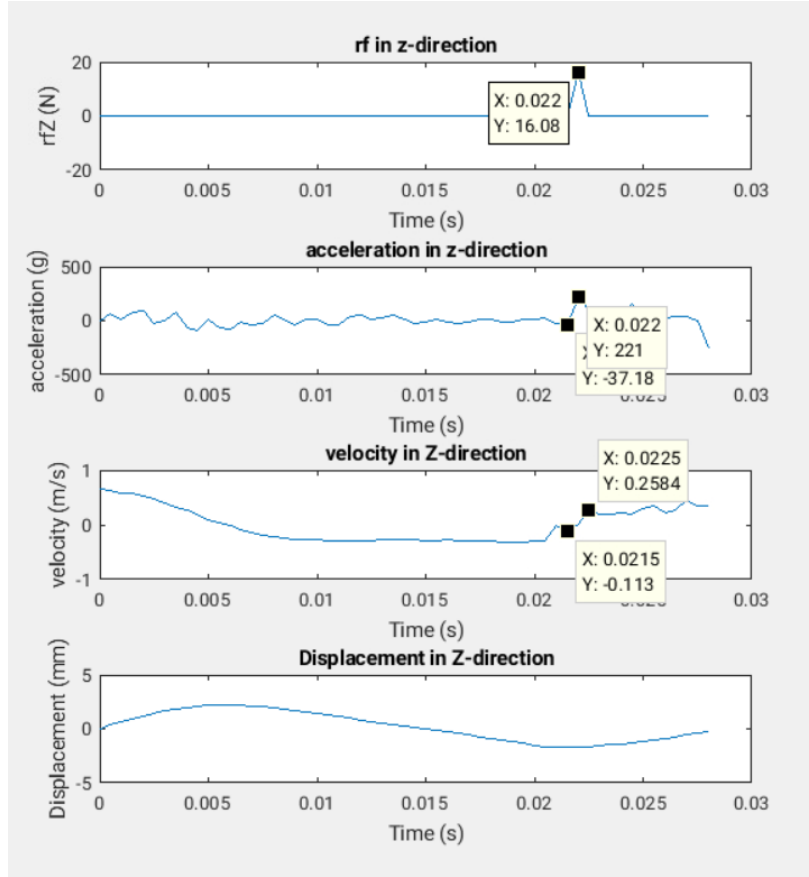


Figure 4.4: Simulation data for St-P1 using Explicit Solver for reaction force(N), velocity(m/s) and acceleration(g) and displacement(mm) in z-direction Vs time(s)

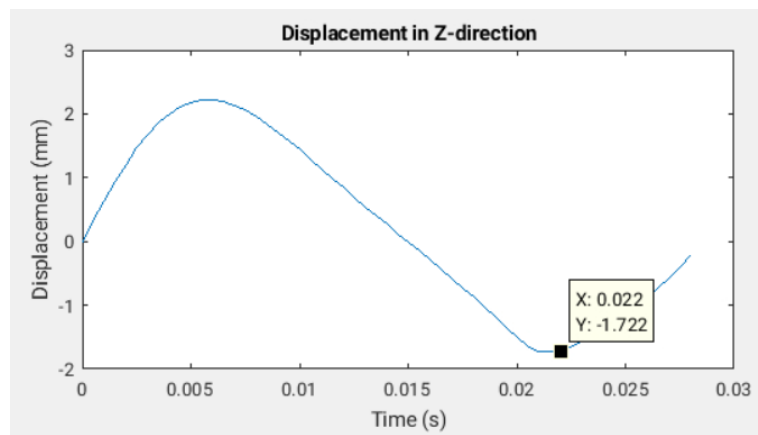


Figure 4.5: Displacement in z-direction(mm) Vs time(s) for St-P1 material pair using Explicit solver

Parameter	Experiment	Simulation-Implicit	Simulation-Explicit
Reaction Force[N]	14.86	15.8	16.08
Velocity before impact[m/s]	-0.5878	-0.2955	-0.113
Velocity peak after impact[m/s]	0.6027	0.1821	0.2584
Acceleration before impact[g]	1.603	4.847	-37.18
Acceleration peak at impact[g]	27.87	30.6	221
Displacement[mm]	-1.9	-2.1	-1.7

Table 4.1: Parameter correlation between experiment and simulation for stp1 0mm gap size.

4.2 Parameter study for different cases

Simulations for different material pairs have been done and the results are evaluated for different parameters as in section 4.1. Two events are taken for a material, one with average reaction force and the other within the range of higher reaction force. The simulation results are compared with experiment for different material pairs and the parameters evaluated for both implicit and explicit solvers are shown in the figures below.

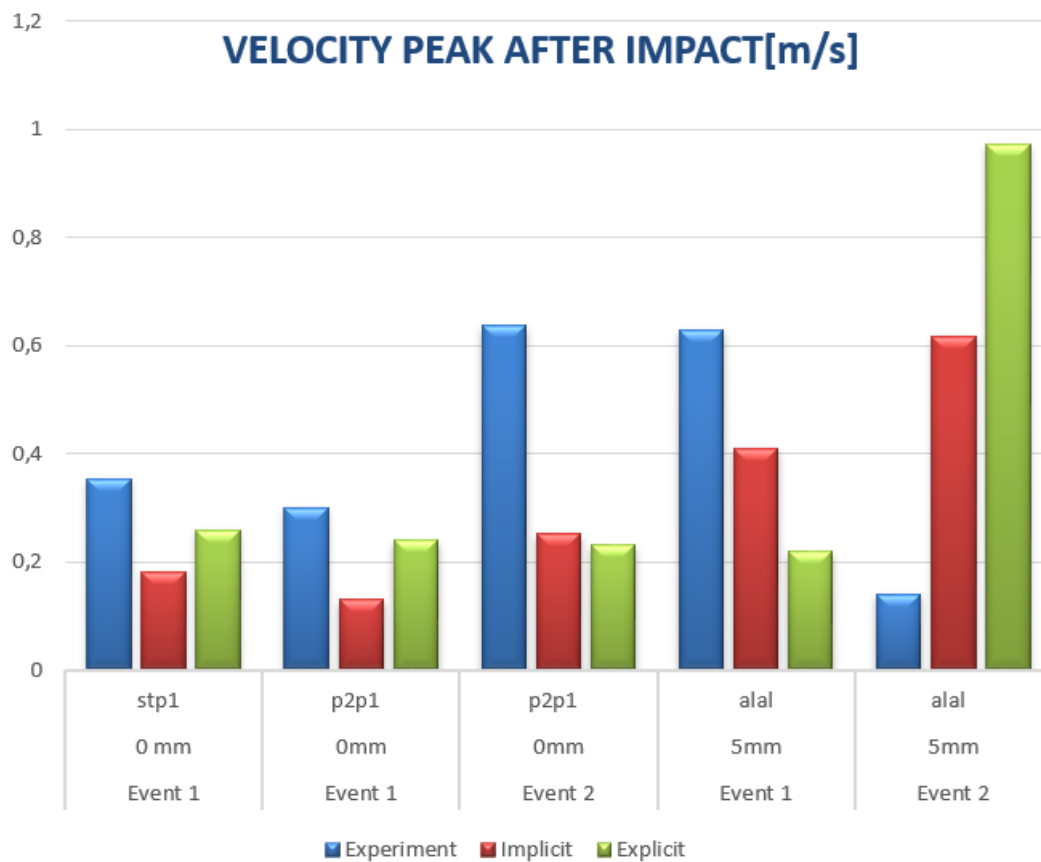


Figure 4.6: Comparison of velocity after impact (m/s) from the experiment with implicit and explicit simulation values for various material pairs

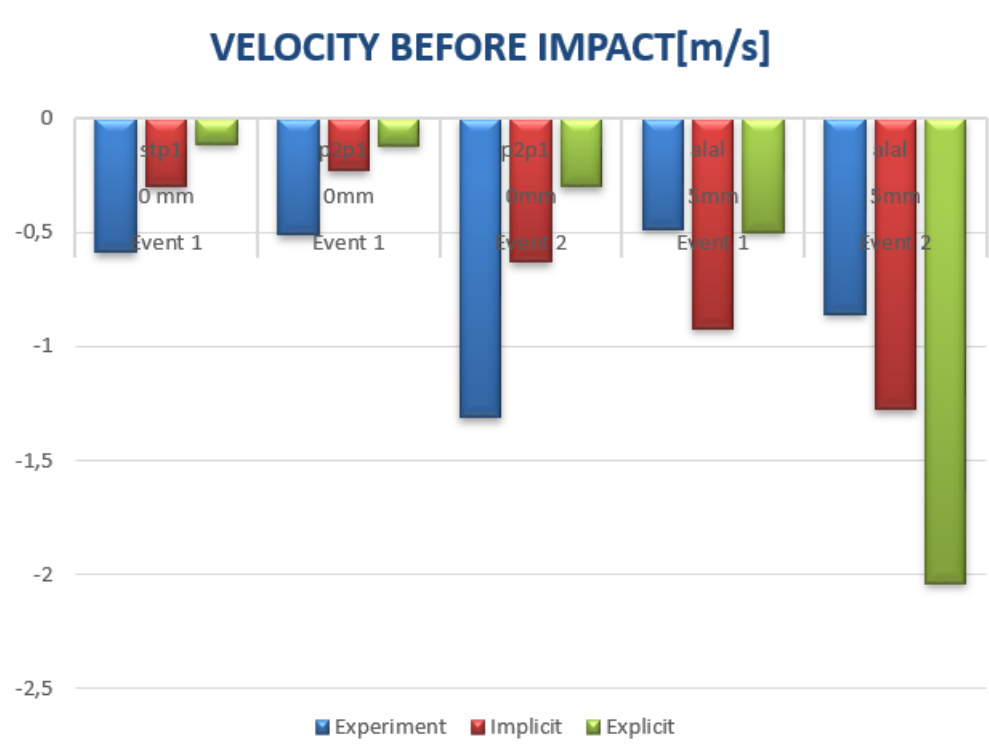


Figure 4.7: Comparison of velocity before impact (m/s) from the experiment with implicit and explicit simulation values for various material pairs

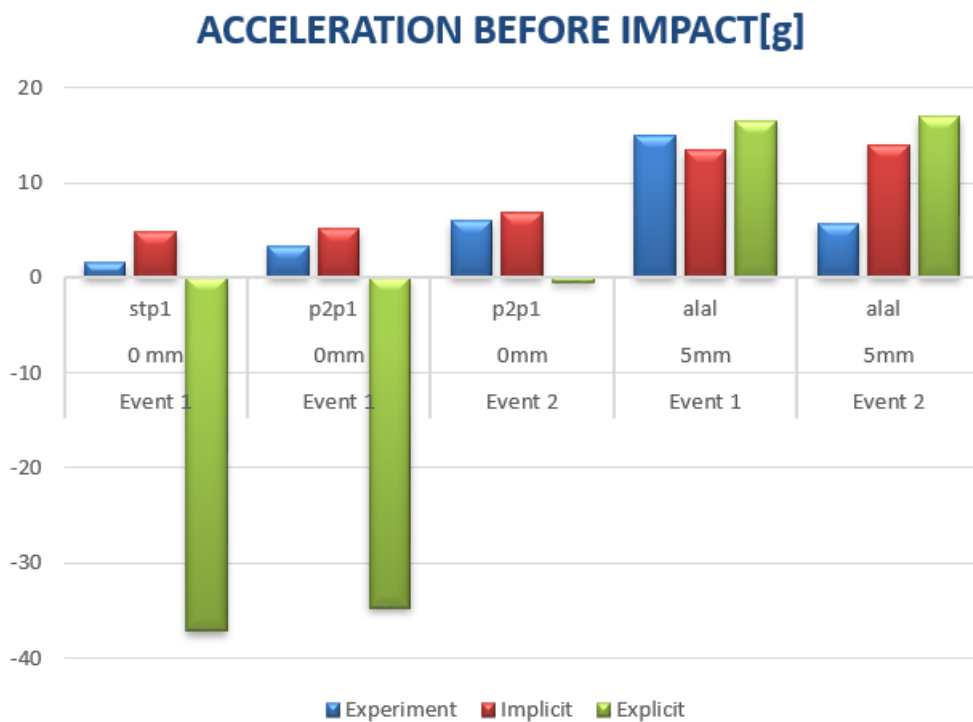


Figure 4.8: Comparison of acceleration before impact(g) from the experiment with implicit and explicit simulation values for various material pairs

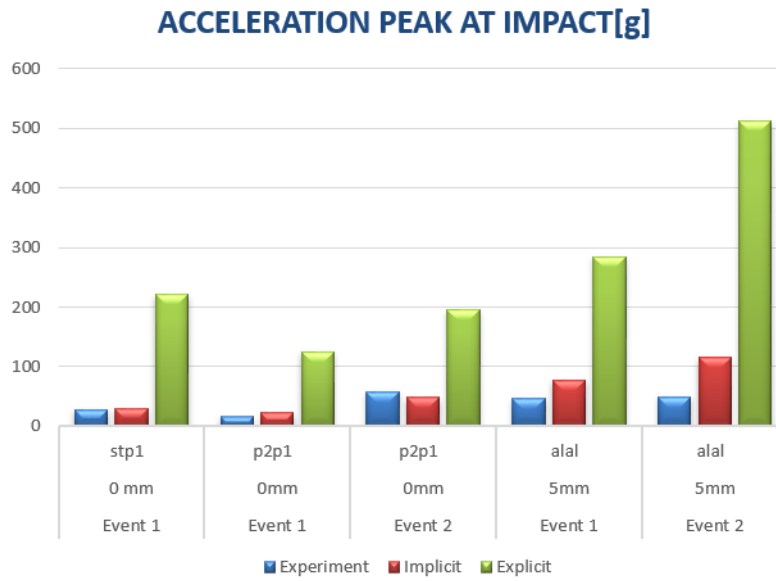


Figure 4.9: Comparison of acceleration at impact(g) from the experiment with implicit and explicit simulation values for various material pairs

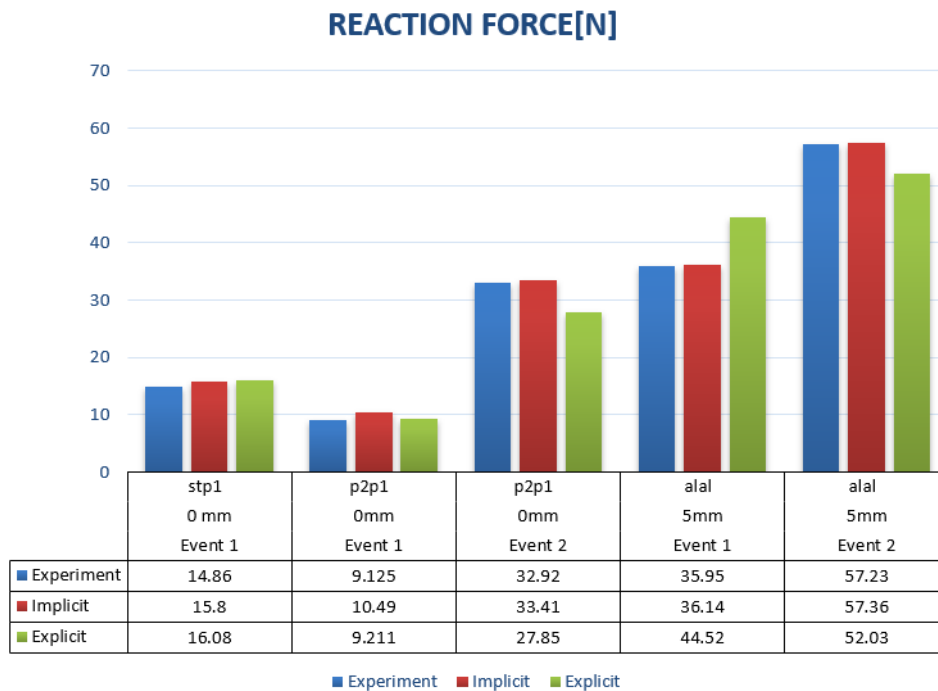


Figure 4.10: Comparison of reaction force(N) from the experiment with implicit and explicit simulation values for various material pairs

From the figures, it is seen that all the 5 material pairs gives good correlation for reaction forces and the velocities in both Implicit and Explicit solvers . It is also seen that in Figure 4.10 the acceleration peak at impact values in simulation during explicit solver is higher when compared to experiment, this is due to fact that the

initial condition values are given only for velocity and this may affect the acceleration values which tends to increase in the simulations.

4.3 Linear Simulations

The linear simulations for different material pairs are done without any contact between the sample beam and sample hammer. Simulation values are obtained for parameters such as relative displacement with penetrations, velocity before impact, maximum relative velocity, acceleration before impact and maximum relative acceleration. Figure 4.11 shows the simulation results for Al-Al material pair with 5mm gap size. Similarly, the values evaluated for the other material pairs.

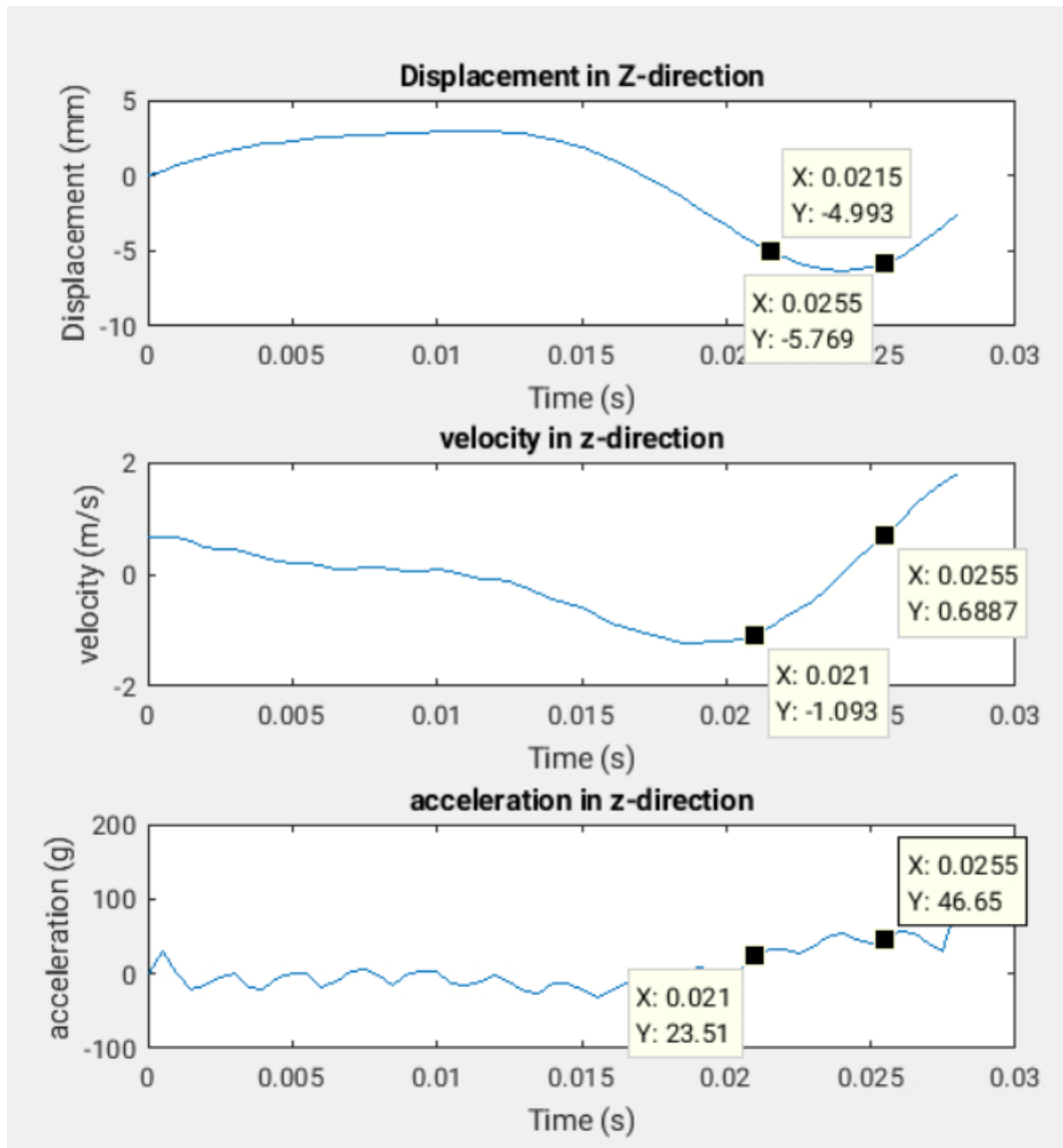


Figure 4.11: simulated parameters for linear case alal-5g

Parameters	Value
Relative Displacement with Penetrations[mm]	0.8
Velocity Before Impact[m/s]	-1.093
Relative Maximum Velocity[m/s]	0.6887
Acceleration Before Impact[g]	23.51
Relative Maximum Acceleration[g]	46.65

Table 4.2: Parameters obtained for linear simulations without contact

4.4 Correlation Between Linear and Non-linear Simulations

The correlation study for linear and non-linear simulation is done by obtaining the values for two parameters Velocity before impact and Acceleration before impact. In linear case once the beam comes into contact with the hammer it starts to penetrate so other parameters cannot be correlated. The figures represents the correlation is shown below.

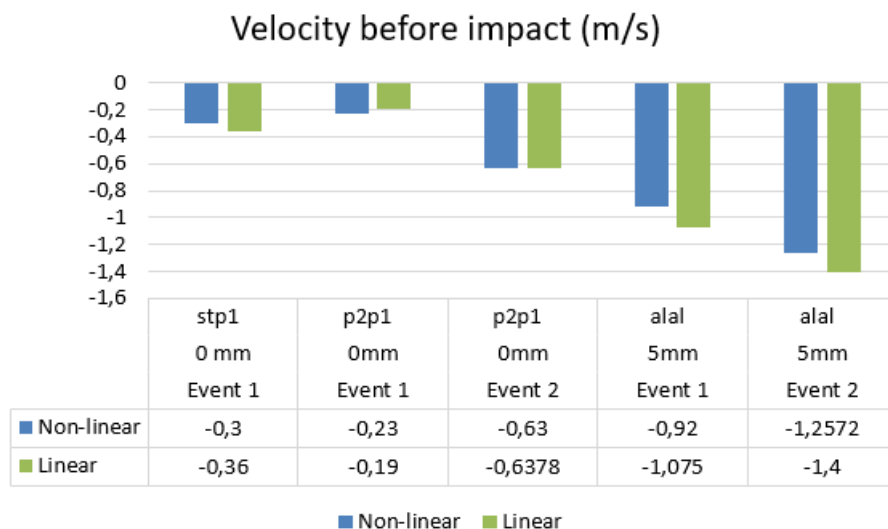


Figure 4.12: Correlation for velocity before impact

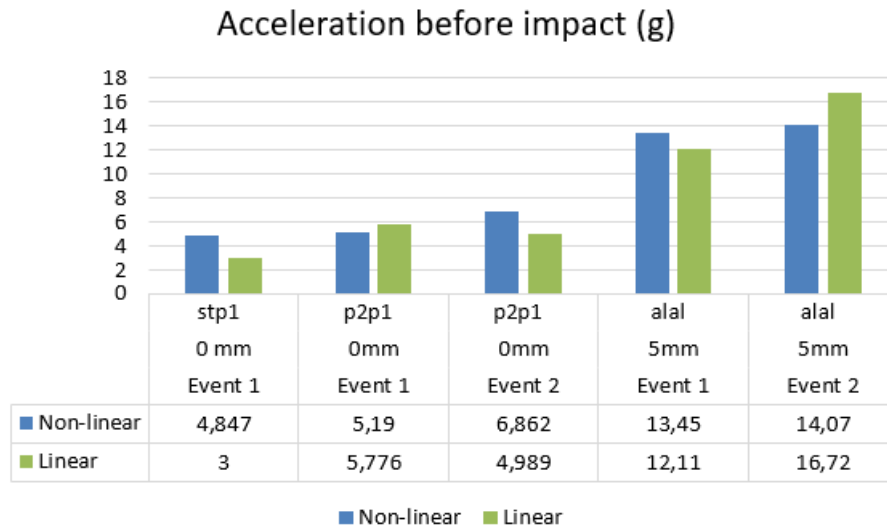


Figure 4.13: correlation for acceleration before impact

Figures 4.12 and 4.13 shows that the parameters matches closely for both the linear and non-linear simulations for the material pairs.

4.5 Discussions

From the above Results sections, it is seen that the results for the structural dynamic parameters in non-linear analysis are correlating well for the implicit and explicit solvers but in the latter it is found that the acceleration values seems to be higher when compared with the experimental one. This is due to the fact that there appears to be a contact bump before the actual point of contact in simulation, this makes the acceleration values in a higher range when compared with the implicit and experimental values. This can be controlled by running the simulation till the first point of contact and then allowing the system to vibrate freely. This can also be controlled by tuning the model to a certain extent and the acceleration value can be reduced. On the other hand with Implicit solver, the model gives good correlation for experiment and simulation for the structural dynamic parameters. It is also seen that the gap size tends to increase the value in acceleration for implicit solver when compared with the experiment. The values obtained for linear simulation without contact between the sample beam and hammer, the penetrations obtained between the beam and the hammer are recorded and the results are shown in section 4.3. The linear simulations are done only with the implicit solver and it is correlated with the non-linear parameters. It is also seen that by visualizing the plots clearly, the material pair Al-Al shows good results when correlated with experiment and simulation for both explicit and implicit cases.

5

Concluding Remarks

5.1 Conclusion

The purpose of the thesis was to predict the rattle noise in the experimental test rig at VCC by correlating the rattle induced structural dynamic parameters with experiment and simulation. The simulation results clearly shows that the structural dynamic parameters are correlating close with the experiment and simulation for different material pairs used in the work. It is important to note that the penetration values are key to evaluate the chance of rattle or squeak, not just the amplitude of movement between the two parts. The results obtained in the implicit solver gives much higher accuracy than the results obtained in the explicit cases. It is good to use implicit solver for this system as it consumes very less time and gives very good correlation for most of the parameters for both linear and non-linear simulation. At the later stages of the work, possibility of a frequency domain analysis with a sweep of frequencies in the range of study to get better visualization of the most critical frequencies which address the rattle issues, is tried to correlate with the time domain analysis obtained for the system, due to time constraints these results are not attached.

5.2 General Recommendations

It is observed that the acceleration values doesn't have a good correlation between experiment and simulation in the explicit cases. One suggested recommendation is to try by giving the initial conditions for acceleration values as velocity, so it can cause some changes to the system and reduce the acceleration values.

5.3 Future Work

The predicted results can be used for a side door speaker assembly at VCC. This can be done by selecting the parameters that gives the good correlation with experiment and implementing those parameters in the side door model. The correlation between simulation and experiment for a side door model can be checked by using the predicted results for the test rig.

This thesis work is a part of and contributes to a larger project with the aim of improving methods and tools for squeak and rattle prediction, therefore the next

5. Concluding Remarks

part of this project is to use these findings for the simple mechanical system to a complex model, that of the side door assembly at VCC and then comparing the results from the previously collected system response.

Bibliography

- [1] Direct solution Steady state dynamics "<https://abaqus-docs.mit.edu/2017/English/SIMACAEKEYRefMap/simakey-r-steadystatedynamics.htm>"
- [2] Steady state dynamics "https://abaqus-docs.mit.edu/2017/English/SIMACAEKEYRefMap/simakey-r-steadystatedynamics.htm#simakey-r-steadystatedynamics__simakey-r-steadystatedynamics-\s-OptionalAndMutuallyExclusiveParametersusedOnlyIfTheDynamic\ResponseIsNotBasedOnModalSuperposition-paramscustom1"
- [3] Székely, András Péter, and Nicole Al Hanna. Nonlinear Modelling and Simulation of Impact Events and Validation with Experimental Test. 2019. kth.diva-portal.org, "<http://kth.diva-portal.org/smash/record.jsf?pid=diva2%3A1380114&dswid=1012>"
- [4] Godborg, Filip, and Anoob Valiyakath Basheer. Model Reduction of Finite Element Models Applied to Nonlinear Squeak and Rattle Simulation. 2019. odr.chalmers.se, "<https://hdl.handle.net/20.500.12380/300325>"
- [5] Blom, Rasmus, and Jonatan Nilsson. Developing a Load Case for Analysis of Squeak and Rattle Events in Passenger Cars. 2020. odr.chalmers.se, "<https://hdl.handle.net/20.500.12380/301114>"
- [6] Eduard Caamano, Inés Lama, Andreas Rousounelos, Jordi Vinas, 2011“Improved methodology for squeak & rattle analysis with abaqus and correlation with test results”, IDIADA Automotive Technology.
- [7] Kadmiri, Younes, et al. “Experimental and Numerical Analysis of Automotive Gearbox Rattle Noise.” Journal of Sound and Vibration, vol. 331, no. 13, June 2012, pp. 3144–57. DOI.org (Crossref), doi:10.1016/j.jsv.2012.02.009.
- [8] Dr.Colin Mercer, 2006 “Acceleration, velocity and displacement spectra –Omega Arithmetic” Prosig.
- [9] “What Are the Differences between Implicit and Explicit?” Welcome

- to the LS-DYNA Support Site, "<https://www.dynasupport.com/faq/general/what-are-the-differences-between-implicit-and-explicit>. Accessed 6 June 2021."
- [10] Trapp, Martin, and Frank Chen, editors. Automotive Buzz, Squeak and Rattle: Mechanisms, Analysis, Evaluation and Prevention. 1st ed, Butterworth-Heinemann, 2012.
- [11] Implicit vs Explicit Approach in FEM – Yasin ÇAPAR. "<https://yasincapar.com/implicit-vs-explicit-approach-in-fem/>."



CHALMERS
UNIVERSITY OF TECHNOLOGY