DYNAMIC ANALYSIS OF STEEL FRAMES SUBJECTED TO IMPULSE LOADS

NUR UZMAIZZATI BINTI MUHAMMAD YUSOF

B. ENG(HONS.) CIVIL ENGINEERING

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

DECLARATION OF THESIS AND COPYRIGHT

Author's Full Name : NUR UZMAIZZATI BINTI MUHAMMAD YUSOF

Date of Birth : 17TH AUGUST 1996

Title : ______________________________

Academic Session : ______________________________

I declare that this thesis is classified as:

☐	CONFIDENTIAL	(Contains confidential information under the Official Secret Act 1997)*
☐	RESTRICTED	(Contains restricted information as specified by the organization where research was done)*
☑	OPEN ACCESS	I agree that my thesis to be published as online open access (Full Text)

I acknowledge that Universiti Malaysia Pahang reserves the following rights:

1. The Thesis is the Property of Universiti Malaysia Pahang
2. The Library of Universiti Malaysia Pahang has the right to make copies of the thesis for the purpose of research only.
3. The Library has the right to make copies of the thesis for academic exchange.

Certified by:

_____________________ (Student's Signature)

_____________________ (Supervisor's Signature)

____
New IC/Passport Number
Date:

____
Name of Supervisor
Date:

NOTE : * If the thesis is CONFIDENTIAL or RESTRICTED, please attach a thesis declaration letter.

THESIS DECLARATION LETTER (OPTIONAL)

Librarian,
Perpustakaan Universiti Malaysia Pahang,
Universiti Malaysia Pahang,
Lebuhraya Tun Razak,
26300, Gambang, Kuantan.

Dear Sir,

CLASSIFICATION OF THESIS AS RESTRICTED

Please be informed that the following thesis is classified as RESTRICTED for a period of three (3) years from the date of this letter. The reasons for this classification are as listed below.

Author's Name
Thesis Title

Reasons (i)

(ii)

(iii)

Thank you.

Yours faithfully,

____
(Supervisor's Signature)

Date:

Stamp:

Note: This letter should be written by the supervisor, addressed to the Librarian, *Perpustakaan Universiti Malaysia Pahang* with its copy attached to the thesis.

(Please take out if not related)

SUPERVISOR’S DECLARATION

I/We* hereby declare that I/We* have checked this thesis/project* and in my/our* opinion, this thesis/project* is adequate in terms of scope and quality for the award of the Bachelor Degree of Civil Engineering

____

(Supervisor’s Signature)

Full Name :

Position :

Date :

____

(Co-supervisor’s Signature)

Full Name :

Position :

Date :

STUDENT'S DECLARATION

I hereby declare that the work in this thesis is based on my original work except for quotations and citations which have been duly acknowledged. I also declare that it has not been previously or concurrently submitted for any other degree at University Malaysia Pahang or any other institutions.

____

(Student's Signature)

Full Name : NUR UZMAIZZATI BINTI MUHAMMAD YUSOF

ID Number : AA15160

Date : 24 MAY 2019

DYNAMIC ANALYSIS OF STEEL FRAMES SUBJECTED TO IMPULSE LOADS

NUR UZMAIZZATI BINTI MUHAMMAD YUSOF

Thesis submitted in fulfillment of the requirements
for the award of the
B.Eng. (Hons) Civil Engineering

Faculty of Civil Engineering and Earth Resources
UNIVERSITI MALAYSIA PAHANG

MAY 2019

ACKNOWLEDGEMENTS

The utmost grateful to Allah S.W.T for His permission that this thesis has been successfully completed.

First of all, special thanks to my supervisor, Dr. Aizat Bin Alias that had been helped me in finding and completing this study. I am also really grateful and thanks for his encouragement and full support.

I am also would like to thank my beloved family that always encouraged and gave support to complete this thesis. Not to forget, special thanks to all my friends that always have helped me in getting ideas and completing this thesis. Next, I would like to take this opportunity to thank to all FKASA lecturers that have been helping me by giving the guideline in order to provide and completed a good thesis.

Last but not least, I also would like to express my gratitude to University Malaysia Pahang for providing all the necessary facilities and gave the opportunity to gain knowledge and experienced here by learning the course offered that can be really useful in the future. Thank you again.

ABSTRAK

Beban impuls adalah beban yang tertakluk pada struktur dalam masa yang sangat singkat. Jenis beban ini biasanya dikaitkan dengan beban letupan, yang dihasilkan daripada tekanan ledakan dari letupan kimia. Respon struktur yang tertakluk pada beban impuls boleh dikaji melalui eksperimen dan pendekatan berangka *(numerical)*. Walaubagaimanapun, eksperimen memerlukan prosedur keselamatan yang sangat ketat dan kos yang tinggi. Oleh kerana itu, permodelan berangka *(numerical modelling)* adalah perdekatan alternatif yang boleh diambil. Jadi, kajian ini adalah untuk mengkaji respon stuktur bingkai keluli yang tertakluk kepada beban impuls dengan menggunakan unsur terhingga *(Finite Element).* Salah satu objektif untuk kajian ini ialah untuk membangunkan satu kod finite *element* menggunakan OCTAVE untuk meramalkan respon stuktur bingkai keluli apabila menggunakan pelbagai beban impuls. Jadi, kertas ini membentangkan tentang analisis dinamik bingkai keluli apabila tertakluk kepada beban impuls. Analisa ini dijalankan menggunakan kaedah unsur terhingga *(Finite Element Method).* Kod Unsur Terhingga *(Finite Element)* dibangunkan dengan menggunakan satu pengisian bahasa pengaturcaraan untuk mengkaji kelakuan dinamik bingkai keluli. Hasi; dari kod sumber yang dibangunkan akan disahkan dengan perisian unsur terhingga komersil, ABAQUS. Daripada kajian ini, perbezaan jenis sokongan mepengaruhi kelakuan dinamik bingkai keluli apabila stuktur tertakluk kepada pelbagai berat letupan di pelbagai jarak menentang.

ABSTRACT

Impulse loads are loading that are subjected on structures in a very short time. These types of loads are commonly associated with blast loads, which are generated from blast pressures from chemical explosions. The response of structures subjected to impulse loads can be studied using experimental and numerical approaches. However, experimental studies require stringent safety procedure and can be costly. Therefore, numerical modelling is an alternative approach, which is the aim of this study. This study is interested investigating the response of steel frames subjected to impulsive loading using finite element (FE) analysis. One of the objectives of this study is to develop a finite element codes using OCTAVE to predict the response of steel frames when subjected to various impulse loads. So, this paper present the dynamic analysis of steel frames subjected to impulse loads. The analysis is performed using finite element method. Finite element source codes were developed using OCTAVE, a programming language software, to investigate the dynamic behaviour of the steel frames. The results from the developed source codes are validated against the result from a commercial finite element software, ABAQUS, where satisfactory results are obtained. From this study, the types of support influenced the dynamic behaviour of the steel frames when subjected to different explosive weights at various standoff-distances

.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

θ	Rotation
[M]	Global Mass Matrix
[K]	Global Stiffness Matrix
{F}	Global Load Vector
{U}	Displacement
$\{\dot{U}\}$	Velocity
$\{\ddot{U}\}$	Acceleration

LIST OF ABBREVIATIONS

FE	Finite Element
FEM	Finite Element Method
FEA	Finite Element Analysis
rc	Reinforced concrete
RK4	Rugge-Kutta 4^{th} Order
FD	Finite Differential

CHAPTER 1

INTRODUCTION

1.1 Background Study

Bombing and explosions are one of the easy ways for terrorist to commit the violence. All the attacks are usually for the terrorist to achieve and justify their blood act on the economic, social and political unfairness and sometime they inspired from the religious beliefs or spiritual principles. Over the past decade, this attacks have been caused thousands of death, damages to development and increased racism over different races, cultures and religions.

For the past decade, the bombing and explosions attack became worst year to year. For the recent bombing attacks in Sri Lanka on 21st April 2019 where the explosions is occurred in eight different location of churches and hotels in Sri Lanka. This attacks have been caused about more than 250 people died and more than 500 people were injured and damages to public buildings.

Usually, the iconic and public buildings is a target for the terrorist attacks. Most of the building have been built or are built without consideration of this unexpected event. All this incident raise the concern the whole world and researcher to study about the response of various structure subjected to impulse load (blast loading).

1.2 Problem Statement

The response of the structure can be studied by performed experimental and numerical approaches. However, experimental studies requires stringent safety procedure and costly. Therefore, numerical modelling is an alternative approach, which is the aim of this study. This study is interested investigating the response of the steel frames subjected to impulse loading using finite element (FE).

Other than that, to perform the numerical analysis, usually researcher will be used the commercial software that available in the market. However, the commercial software is restricted, having some problem with the licensing issues and sometimes costly to be used and learned for beginner designer. So, this study is focusing in develop the FE open source code using the open software to analyse the response of the steel frames subjected to impulse load.

1.3 Objectives

The objectives of this study are:

i. To develop a FE code using OCTAVE to predict the response of steel frames when subjected to impulse loads.

ii. To compare the develop finite element source code with ABAQUS.

iii. To investigate the response of steel frames subjected to different impulse loads and different boundary conditions using finite element method (FEM).

1.4 Scope of Study

The scope of this study:

i. This study is focusing on elastic analysis only.

ii. This study is analysed for different boundary condition which is simple-supported and fixed steel frames and also for different impulse loads.

iii. This study is identified the behaviour of steel frames subjected to impulse loads.

1.5 Significance of Study

The significance of this study are:

i. To provide an open source code software that focus on simple analysis of frames so that can be used for designer engineer.

ii. To provide a platform for the beginner designer and can be used for educational purposes to get more understanding for the fundamental of the FE analysis.

1.6 Summary

This thesis will contain of five chapters which in Chapter 1 shall discuss about the introduction of this study and give some overview about developed FE code.

Chapter 2 which is Literature Review will be discussed about some cases study, that almost similar to this study. It is also elaborate the different types of explosion, comparison between commercial software and develop source code.

Chapter 3 which is Methodology will be discussed on approach and frame work have been used in this study. It will cover the FEM of frames, two different time integration approach and other techniques that have been used for the analysis. The methodology will be discussed in detail in this chapter.

Chapter 4 will be shown and discussed the results from the developed FE source code using OCTAVE. This chapter also will be discussed on the validation with commercial software and verification other method.

Chapter 5 argue on conclusion of dynamic analysis of steel frames subjected to impulse loads. This is meaningful for the whole description of this project.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

In this chapter will be discussed based on the similar study of dynamic analysis subjected to impulse loads using numerical approach. There are several studies using the commercial software such as ABAQUS, ANSYS and more. All related study will be discussed and compared for the behaviour of the frames. Other than that, in this chapter also will be discussed about the type of blast loads, effect the blast loading profile and explosive threats.

Figure 2.1 Example of explosion incident

2.2 Blast Loading and Explosion

According to (T. Ngo, et al., 2007) explosion can be classified based on their nature which physical explosion, chemical explosion, nuclear and electrical explosion. Besides, the blast and explosion can be divided to two categories which is unconfined and confined explosions.

Table 2.1 Categories of Explosion

Category	Description
Unconfined Explosion	i. **Air Blast** – Takes place at a given distance and at a certain height away from structure. ii. **Surface Blast** – Explosion take place close to or on the ground surface.
Confined Explosion	If the explosions occur within the structures

2.3 Effect of Blast Loads

According to (Agrawal, 2015), the effects of blast load can be defined as below:

i. Direct ground shock where the buried explosive can cause ground shock. The results will produced horizontal vibration that similar to earth quake

ii. Heat is a part of the explosive energy is converted to heat. Due to higher temperature, the building material tend to weakened and damage. However, if the temperature is higher enough, it can cause fire.

iii. Primary Fragments from the explosive source which thrown into the air at high velocity may destroy windows and glass facades and cause injury to surrounding.

iv. Progressive collapse is blast that directly destroys a structure that caused failure to whole structures.

2.4 Explosive Threats

Based on (Smith & Hetherington, 1994), explosive threats can be defined as shown Table 2.2.

Table 2.2 Explosive Threats

Threat	Description
Nuclear Device	Bomb releasing massive quantities of energy.
Vehicle Bomb	Common terrorist weapon designed to cause blast damage to structures.
Package Bomb	Common hard-portable terrorist device that can caused high level of damage.
Gas and vapour cloud explosion	Military fuel/air munitions

From the Table 2.2 shown above, there several of explosive threat that commonly happened recently. According to (United States Bomb Data Center (USBDC), 2017), incidents involving packaging/luggage/ bag increasing from year 2016 to 2017 which is about 55 percent. This shows that this hand-portable bomb is most preferable way for terrorists as the bag or luggage is small size, portable and it really easy to bring anywhere. For example, Sri Lanka attacks where the terrorists placed the bomb in the bag/luggage at eight different places. From the attacks, it caused more than 250 deaths and 500 injuries and also damages to public buildings. Because of that, this study is performed based on FEMA 426 [reference] where the chosen blast scenario is based on explosives stored in a luggage (FEMA 426, December 2003).

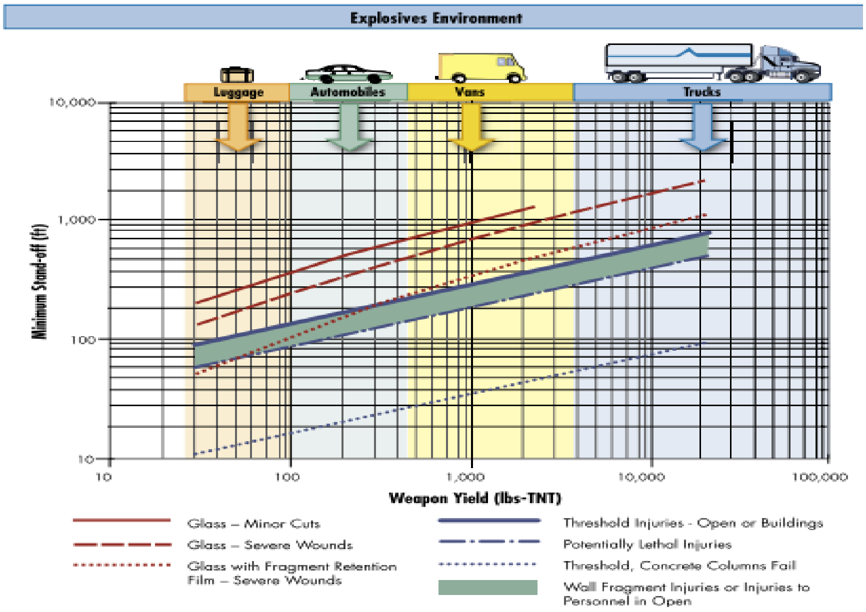


Figure 2.2 Sources: (FEMA 426, December 2003)

2.5 Blast Waves

According to (Smith & Hetherington, 1994), blast loading profile can be expressed as shown in Figure 2.3:

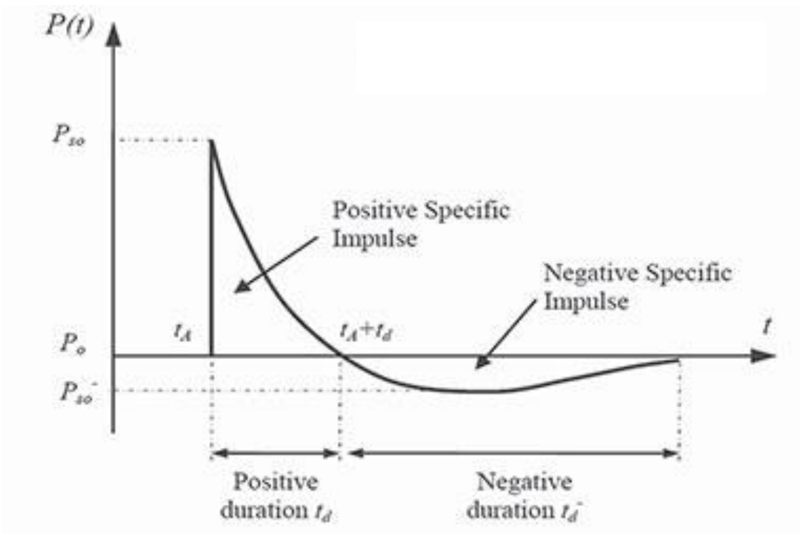


Figure 2.3 Blast Loading Profile

From the figure above, idealised profile of the pressure in relation to time. The negative impulse wave usually not taken into account of design as verified that high damage level occurred in positive phase. Additionally, the pressure from negative impulse will produced lower than positive phase.

Thus, this blast loading profile also can be known as force function. Force function is used in the dynamic analysis. However, this blast profile can be simplified to rectangular and triangular force shape function.

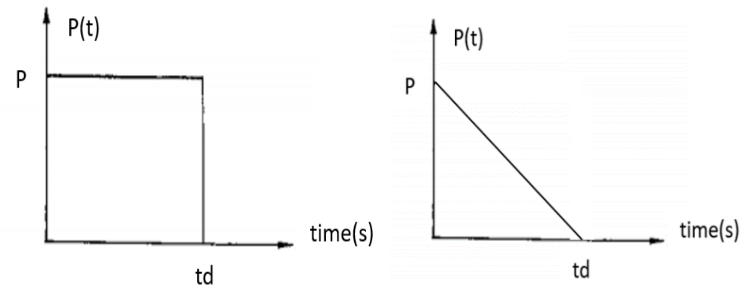


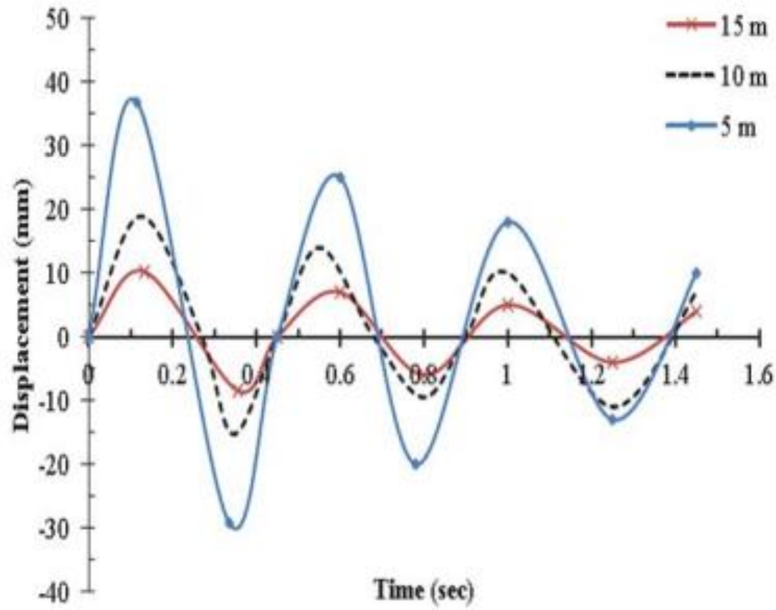
Figure 2.4 Force Shape Function (a) Rectangular (b) Triangular

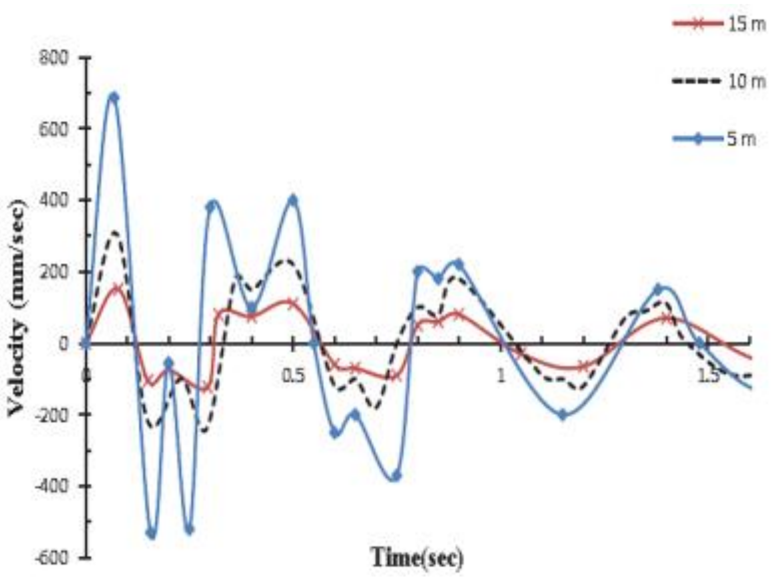
For this study, finite element (FE) source code is developed based on the rectangular and triangular force shape functions. The triangular force shape function should produce half of rectangular force shape function. This function will reduced the maximum displacement when the pressure applied.

2.6 Dynamic Behaviour of Structures Subjected to Impulse Loads.

Dynamic analysis is load that varies with time. Load is applied over time or frequency, and time and inertial effects are relevant. Next, earthquake, blast, wave are several example of the dynamic load. From the dynamic analysis, behaviour of structures can be determined where the failure mode of the structures when low or high pressure is applied, even different material and boundary condition could affect the structures. There are several previous studies that study about the behaviour of structures when subjected to blast loading.

One of the previous study by (Bhosale, et al., 2016), the numerical modelling using Staad Pro is used for analyse five-storey reinforced concrete (rc) frame structures to investigate the dynamic analysis subjected blast loading. The parameters used in this study are determined by adopting wave scaling laws given from (UFC, 2008). This study stated that explosive load or blast load are important for designing structure. Then, this study is presented and simulated the numerical model using Staad Pro to analyse the rc frame for various blast weight and different standoff distance. After the validation, the results from the Staad Pro is presents as shown in Figure 2.5





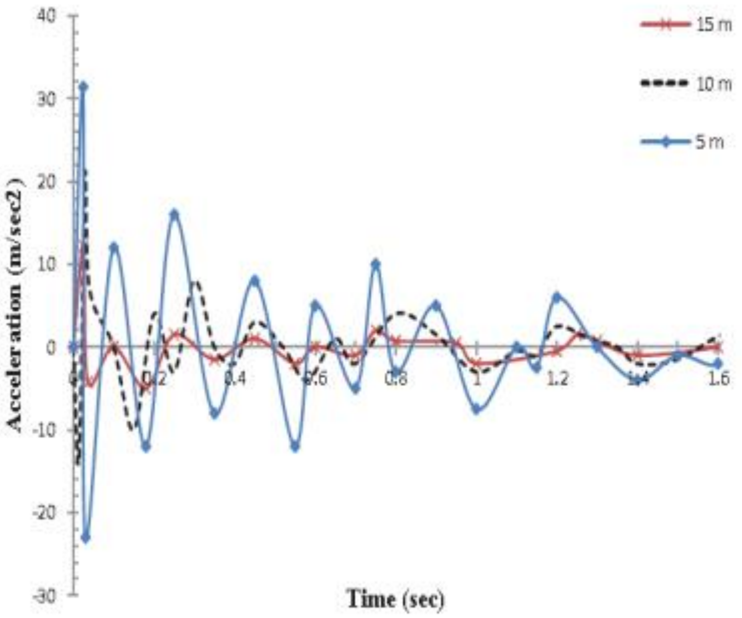
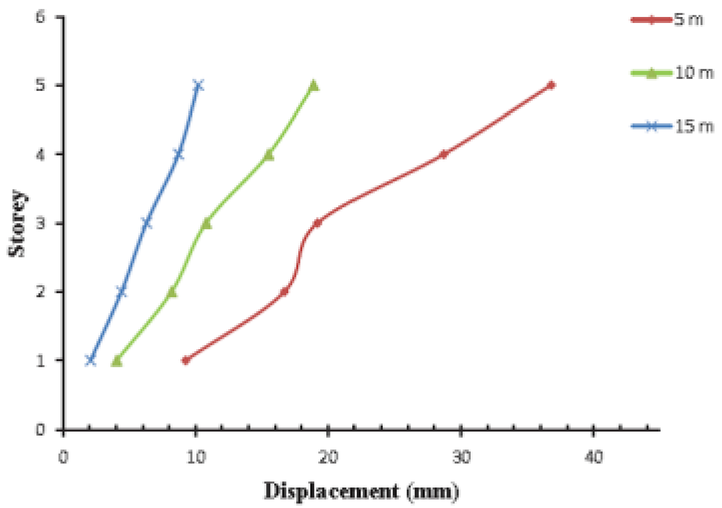
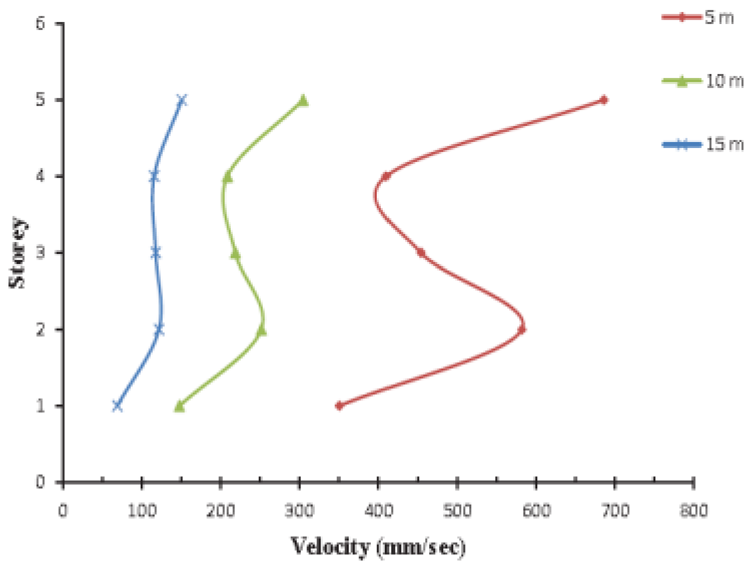


Figure 2.5 Results from Staad Pro. Souces: (Bhosale, et al., 2016)

The results is presented for displacement, velocity and acceleration versus time. This figure present the behaviour of the rc frames when subjected the blast loads for different standoff distance of 15m, 10m and 5m. In addition this study also presented the maximum displacement, velocity and acceleration as shown in Figure 2.6





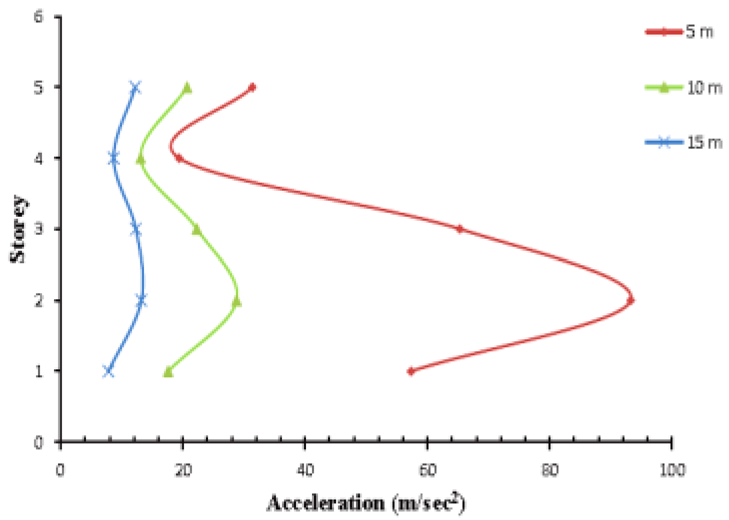


Figure 2.6 Maximum displacement, velocity and acceleration. Sources: (Bhosale, et al., 2016)

According to the study, it can be concluded that, peak reflected overpressure is increases when the weight of blast and decreases as increase in standoff distance. Next, based on this study, the blast pressure is inversely proportional to blast scaled distance and the maximum nodal displacement, the maximum velocity and the maximum acceleration decreases as standoff distance increases.

2.7 Numerical Modelling

Numerical modelling is mathematically present and simulate the behaviour of structures. It is commonly used in analyse earthquake and blast loading. Most usually, the numerical models will be describes the experimental methods and compared the results. After the validation, parametric studies can be performed.

There are several method can be used to perform the analysis such as finite element method (FEM), finite difference method (FDM), discrete element method (DEM) and more. The result obtained from the numerical modelling, will helps the designer to design the structure that mitigate the catastrophic and progressive collapse.

Based on the previous study by (Ruwan, et al., 2011), this study present of the impact of near field explosions on the rc structural framing system to analyse the component response subjected to blast loading. This paper cover from the linear elastic and non-linear plastic response. Next, this study using numerical approach to simulate the framing system and the results obtained will be validated and compared to experimental results. There are two stage finite element modelling where in the first stage, the analysis carried out using SAP2000 to verify response of rc framing system and LY-DYNA for the second stage to investigate the non-linear response. Figure 2.7 shown the validation of experimental, present study and experimental residual displacement. Then, the damage mechanism is obtained by plotting the stress with plastic-strain diagram shown in Figure 2.9.

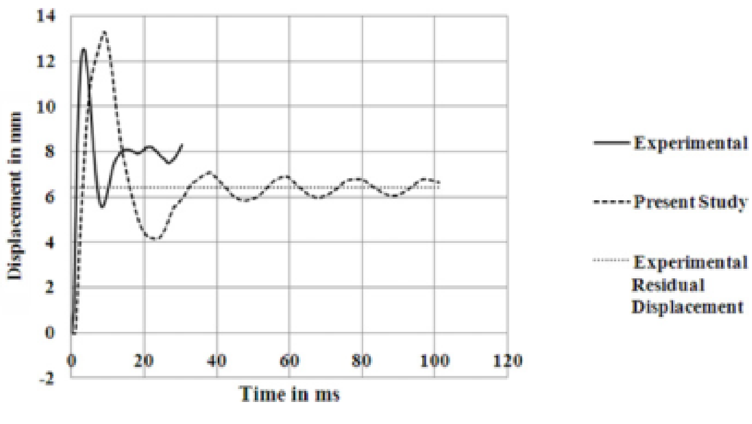


Figure 2.7 Displacement vs Time

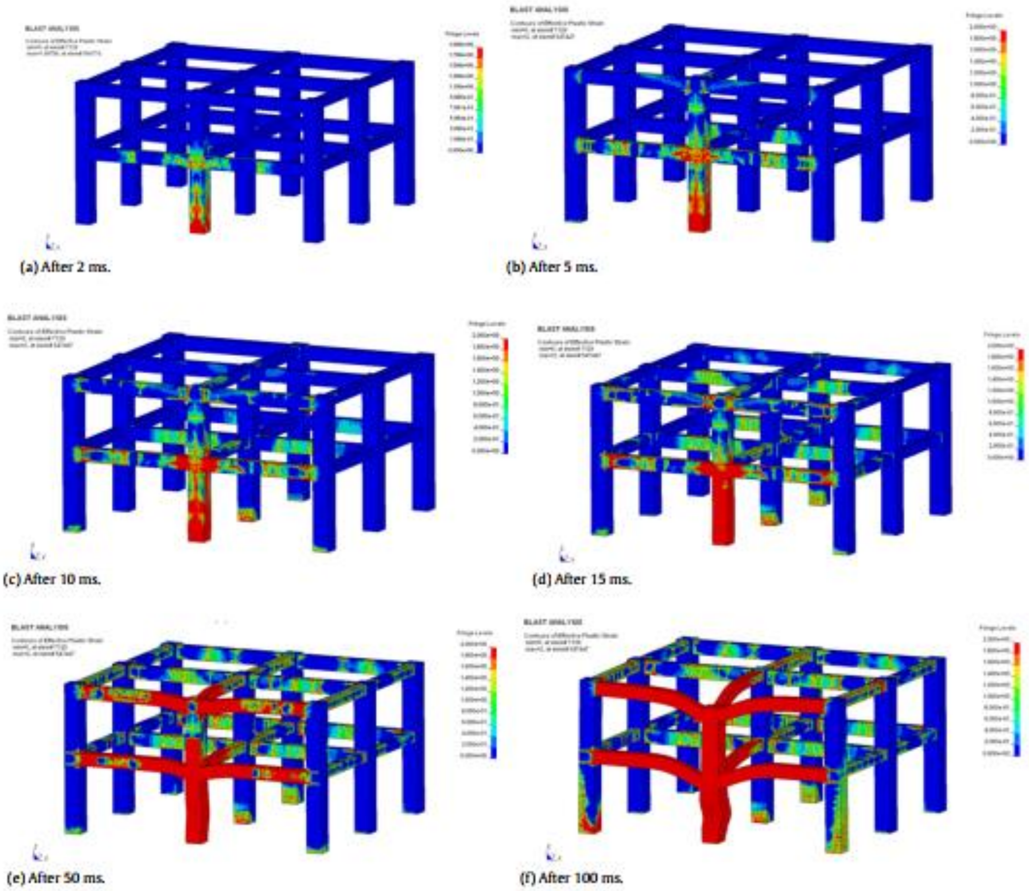


Figure 2.8 Output from SAP2000

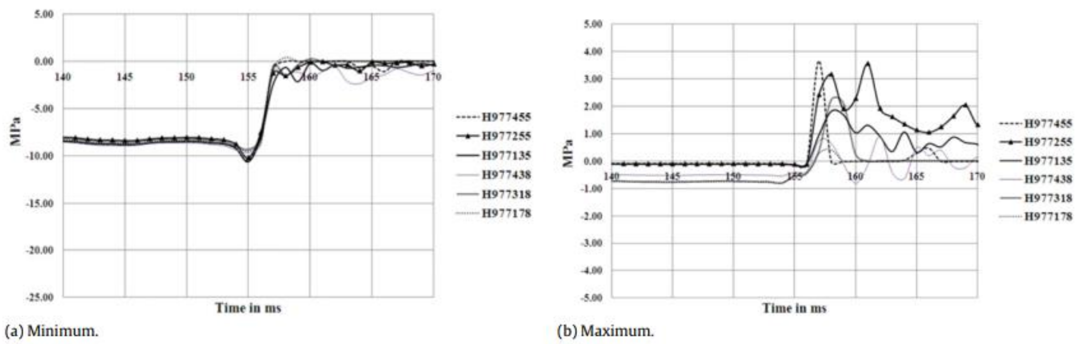


Figure 2.9 Plot of principal stress in an element

According to Figure 2.7, the result obtained from experimental with the numerical, the maximum displacement is almost same. In Figure 2.8, the output from the numerical modelling shown the effect when the impulse load is applied into the framing system. It can be concluded that by using numerical modelling, the dynamic analysis subjected to the blast loading is reliable. By using the data from Figure 2.9, the component response can be identified when subjected to blast loading.

2.8 Summary

Overall, in this chapter discussed about blast loading and explosion and past studies. The blast can be classifies into physical, chemical, nuclear and electric and also can be categorized into two which is unconfined and confined explosion. Next, the luggage and bag always being chosen by terrorist as the explosion threats. As the luggage or bag is in small size and very portable that will be easier to bring anywhere. In addition, this chapter also discussed about the blast profile and the structural dynamics from the free vibration mass-system. Lastly, this chapter will discussed about the past studies from other researchers.

CHAPTER 3

METHODOLOGY

3.1 Introduction

This chapter will be discussed about the method and approach used in developing the FE source code. This dynamic analysis is performed using finite element method (FEM) and two different time-integration which is Runge-Kutta (RK4) and Finite Central Difference (FD). So, in this chapter will be discussed in detail, the equation and approach used in this analysis.

Next, this chapter also will be discussed on the parameter of specification and impulse loads used for this analysis based on FEMA 426 and EUROCODE.

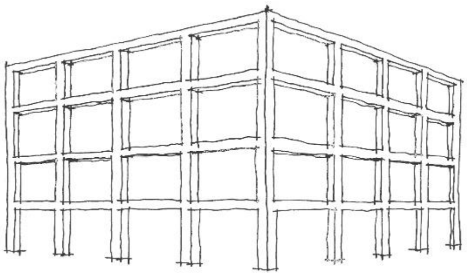


Figure 3.1 Framing System.

3.2 Finite Element Analysis (FEA) of Frames

According to (Mohd Yassin, et al., 2018) frame system is assembly of inclined beam- column elements as shown Figure 3.2.1 below.

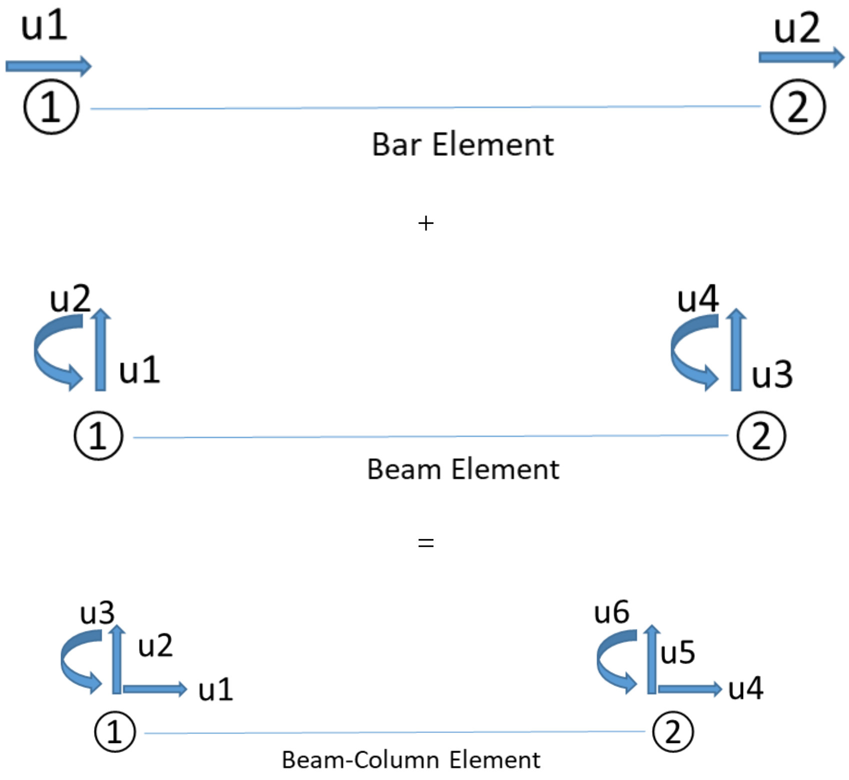


Figure 3.2 Beam-column element as a combination of bar and beam elements.

From Figure 3.2 shown, beam-column can be determined from combination of bar element and beam element and produced six load displacements.

3.2.1 Flow Chart of Finite Element Analysis

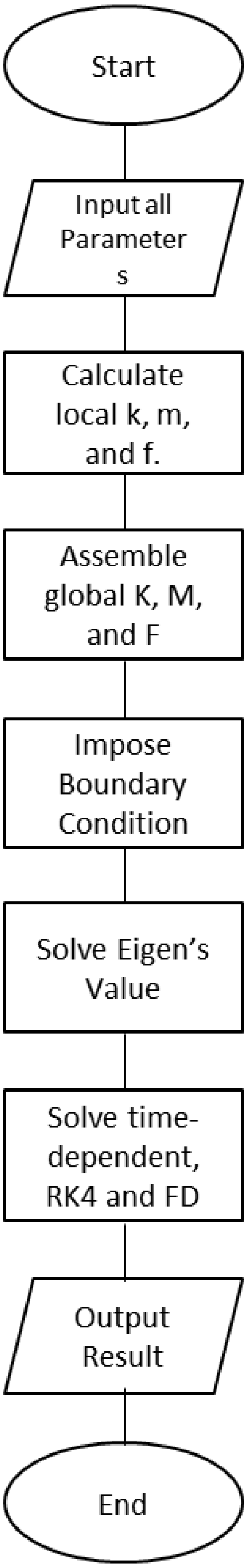


Figure 3.3 The general flowchart of FE code for dynamic analysis

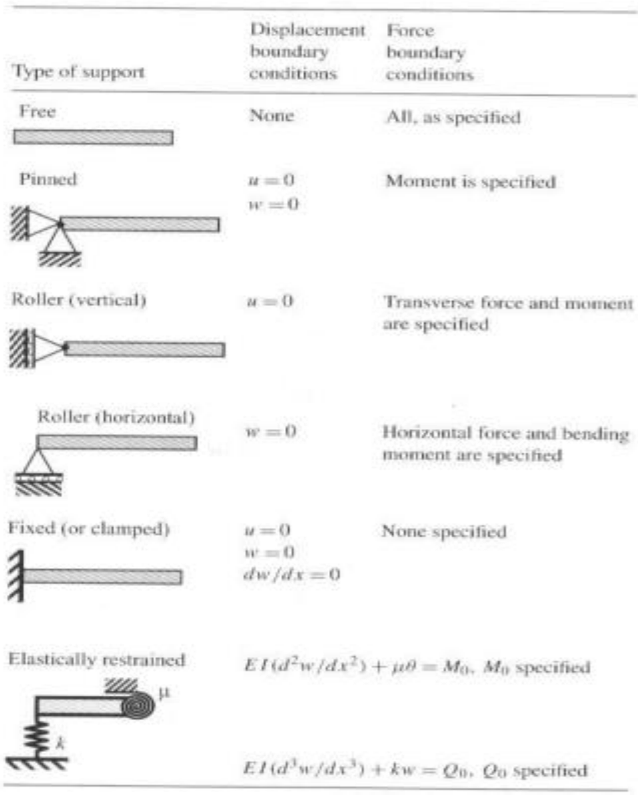
From the Figure 3.3 above, there are shown a flowchart how the FEA is performed. The steps of FEA as stated below:

i. Step 1: Input reliable parameters such as Young's Modulus, Moment of Inertia, Area of the beam column, specification of beam column and the Impulse load parameters.

ii. Step 2: Calculate the local stiffness matrix[k], mass matrix[m], load vector matrix {f} and transformation matrix [T] for every elements of the frames.

iii. Step 3: Assemble all to the global stiffness matrix [K}, mass matrix [M] and load vector matrix {F}. The size of global is based on total number of degree of freedom.

iv. Step 4: Impose boundary according the boundary condition applied into the structure. For this study, simple supported frames and fixed-fixed frames.



Type of support	Displacement boundary conditions	Force boundary conditions
Free	None	All, as specified
Pinned	$u = 0$ $w = 0$	Moment is specified
Roller (vertical)	$u = 0$	Transverse force and moment are specified
Roller (horizontal)	$w = 0$	Horizontal force and bending moment are specified
Fixed (or clamped)	$u = 0$ $w = 0$ $dw/dx = 0$	None specified
Elastically restrained	$EI(d^2w/dx^2) + \mu\theta = M_0$, M_0 specified $EI(d^3w/dx^3) + kw = Q_0$, Q_0 specified	

Figure 3.4 Displacement Boundary Condition for different type of support, Sources: (Reddy, 2006)

v. Step 5: Solve the analysis

3.2.2 The Parameters

The parameters and specification of the steel frames and impulse loads are based on (FEMA 426, December 2003) and (British Standard, 2005).

3.2.2.1 The Input Parameter of Steel Frames

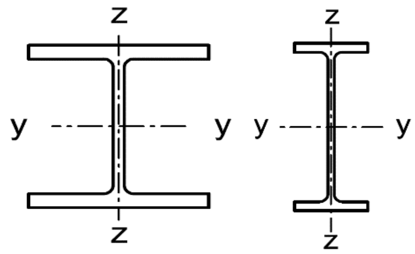


Figure 3.5 The Cross Section of Beam and Column

Table 3.1 The input parameter of steel frames

Parameter	Beam	Column
Young's Modulus, E (kN/m)	200×10^{9}	
Area, A ($\boldsymbol{m^2}$)	0.0055	0.0213
Inertia of moments, I (($\boldsymbol{m^4}$)	6.54×10^{-5}	3.00×10^{-4}
Density, $\boldsymbol{\rho}$ ($\boldsymbol{kg/m^3}$)	7860	
Length, L (m)	3.50	5.00
Cross-section (m x m)	254 x 146	254 x 254

In this study, a steel frames has been chosen include the section (UKB 254X146 and UKC 254 x 254) as mention Figure 3.5 The Cross Section of Beam and Column Figure 3.5 and Table 3.1 above.. All input is taken from BS EN 1993-1-1:2005, BS 4-1:2005

3.2.2.2 The Impulse Loads Parameter

Table 3.2 The Impulse Loads Parameter Based on (FEMA 426, December 2003)

Case	**Standoff distance,** **S (m)**	**Explosive mass,** **M (kg)**	**Scaled distance** **Z** **(m/$kg^{1/3}$)**	**Reflected Pressure P_r** **(MPa)**	**Reflected impulse,I_r** **(X 10^{-3}MPa)**
Case 1	3.4	5.4	1.93	1.17	0.66
Case 2	5.4	5.4	3.07	0.31	0.38
Case 3	3.4	45.36	0.95	9.23	3.4
Case 4	3.4	45.36	1.51	2.45	1.8

The impulse loads parameter that used in this study is based on FEMA 426 [reference] where the chosen blast scenario is based on explosives stored in a luggage. This blast scenario is chosen because luggage or begs is one of the easiest way for terrorists to carry the explosive materials to certain locations as been used in Sri Lanka blast incidents.

3.2.3 Stiffness Matrix, Mass Matrix and Load Vector Matrix

The general FE formulation of equation of motion for structures can be given as (for one element of steel frames) (Hibbeler, 2017)

$$[M]\{\ddot{U}\} + [K]\{U\} = \{F\} \qquad 3.1$$

Where the [M], [K], {F} are represent as global mass matrix, global stiffness matrix and global load vector (force) respectively. As $\{\ddot{U}\}$ is acceleration and $\{U\}$ is known as displacements. This equation is to perform the dynamic analysis.

3.2.3.1 *Stiffness of Matrix, [k]*

$$[k] = \begin{bmatrix} \frac{AE}{L} & 0 & 0 & -\frac{AE}{L} & 0 & 0 \\ 0 & \frac{12EI}{L} & \frac{6EI}{L} & 0 & -\frac{12EI}{L} & \frac{6EI}{L} \\ 0 & \frac{6EI}{L} & \frac{4EI}{L} & 0 & -\frac{6EI}{L} & \frac{4EI}{L} \\ -\frac{AE}{L} & 0 & 0 & \frac{AE}{L} & 0 & 0 \\ 0 & -\frac{12EI}{L} & -\frac{6EI}{L} & 0 & \frac{12EI}{L} & -\frac{6EI}{L} \\ 0 & \frac{6EI}{L} & \frac{4EI}{L} & 0 & -\frac{6EI}{L} & \frac{4EI}{L} \end{bmatrix} \qquad 3.2$$

3.2.3.2 *Mass Matrix, [m]*

$$[m] = \frac{\rho AL}{420}\begin{bmatrix} 140 & 0 & 0 & 70 & 0 & 0 \\ 0 & 156 & 22L & 0 & 54 & -13L \\ 0 & 22L & 4L^2 & 0 & 13L & -3L^2 \\ 70 & 0 & 0 & 140 & 0 & 0 \\ 0 & 54 & 13L & 0 & 156 & -22L \\ 0 & -13L & 3L^2 & 0 & -22L & 4L^2 \end{bmatrix} \qquad 3.3$$

3.2.3.3 *Load Vector Matrix, {f}*

$$\{f\} = \begin{Bmatrix} \frac{qL}{2} \\ \frac{qL}{2} \\ \frac{qL}{12} \\ \frac{qL}{2} \\ \frac{qL}{2} \\ -\frac{qL}{12} \end{Bmatrix} \qquad 3.4$$

3.2.3.4 Transformation Matrix, [T]

$$T = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 & 0 & 0 \\ -\sin\theta & \cos\theta & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\theta & \sin\theta & 0 \\ 0 & 0 & 0 & -\sin\theta & \cos\theta & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \end{bmatrix} \qquad 3.5$$

3.2.4 Force Shape Function

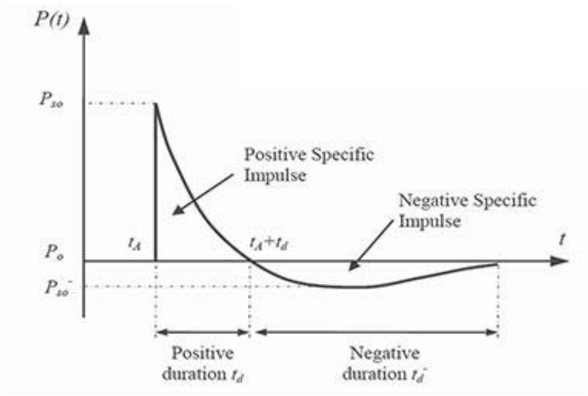


Figure 3.6 Blast Loading Profile

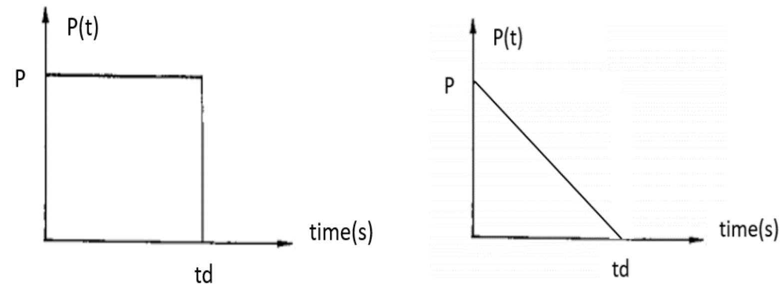


Figure 3.7 Force Shape Function

In this study, internal loading is considered to investigate the behaviour of the steel frames.

3.3 Time-Dependent

In order to solve the time-dependent problem, there several method of ordinary differential equation can be used. For this study, Finite Central Difference (FD) and Runge-Kutta 4th Order (RK4) approaches are used for solving the time-dependent problem. According to (Zill, 2018), the FD and RK4 can be obtained as shown below:

3.3.1 Finite Central Difference (FD)

Finite Central Difference (FD) is one of the simplest and oldest way to solve differential. By consider the initial time, t = 0 and x = 0, FD can be expressed as:

Table

Acceleration, a	Velocity, v	Displacement, d
$a = \ddot{U} = v\frac{d^2u}{dt^2}$	$v = \dot{U} = \frac{du}{dt}$	$d = U = \frac{du}{dx}$

$$\begin{cases} \frac{du}{dt}(t,x) - v\frac{d^2u}{dt^2}(t,x) = f(t,x) \\ u(0,x) = u_0(x) \\ u(t,0) = u(t,1) = 0 \end{cases} \quad 3.6$$

3.3.2 Runge-Kutta 4th Order (RK4)

Runge-Kutta 4th Order (RK4) method is a numerical technique used to solve ordinary differential equation of the form:

$$\frac{dy}{dx} = f(x,y), y(0) = y_0 \quad 3.7$$

3.4 Software (OCTAVE)

Octave is open software where all user can used without worry about licensing issues. Octave is easy to use and good platform to learn and perform the programming. It is free based that all beginner can use to perform any analysis. By using (Ferreira, 2009) as reference in developing this sources code.

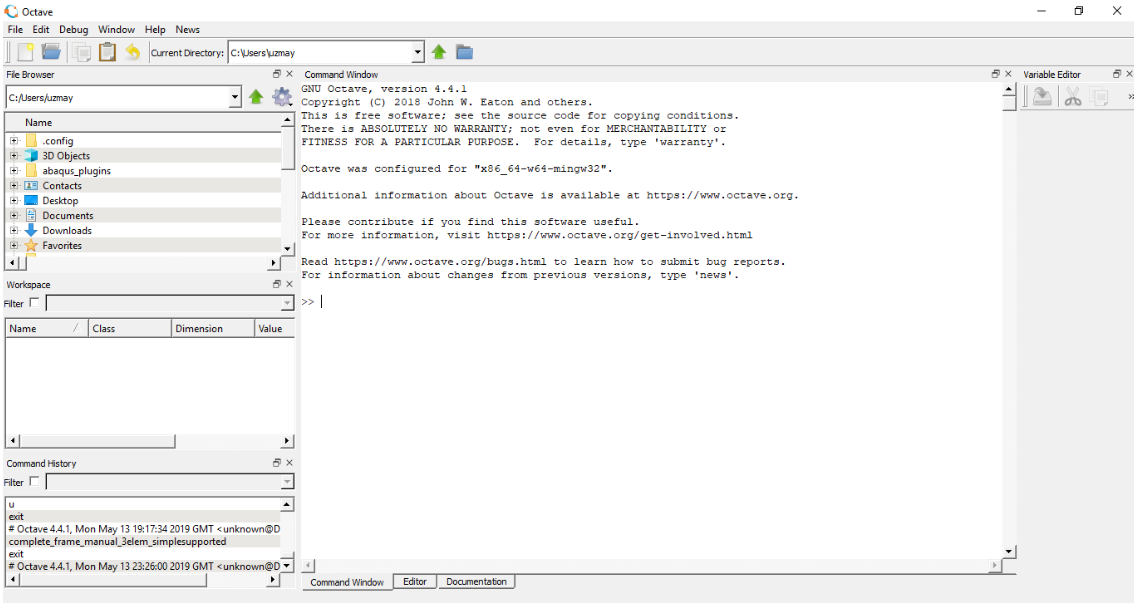


Figure 3.8 First Phase of Octave

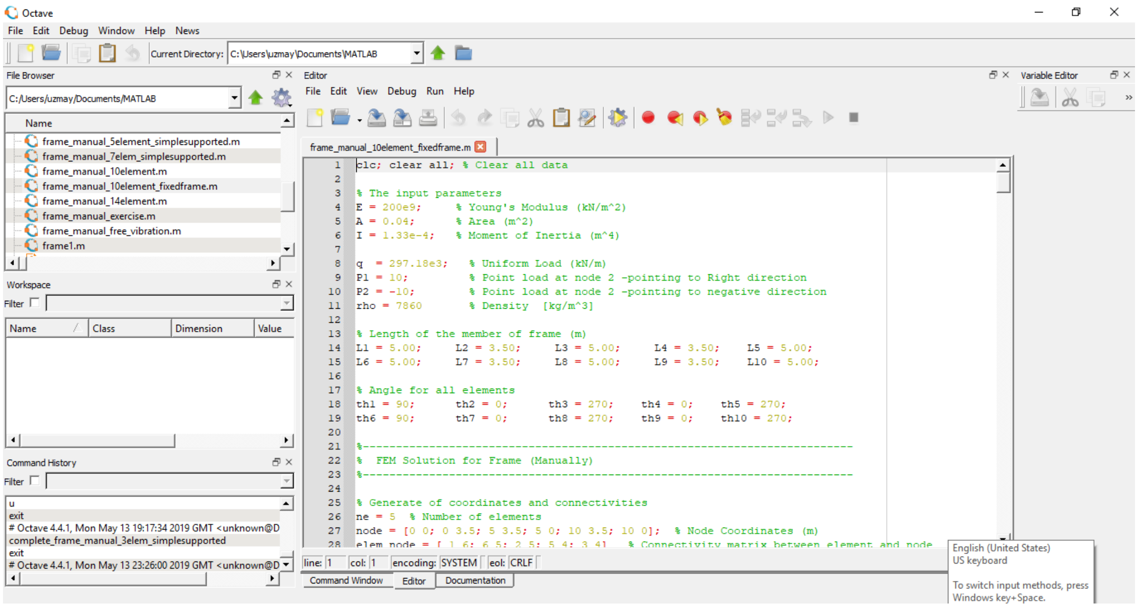


Figure 3.9 Editor Windows

Based on Figure 3.8 and Figure 3.9, first phase is first page when the open the Octave. Next, editor windows is where the coding is placed to run the analysis. All the command based on FE is included into analysis.

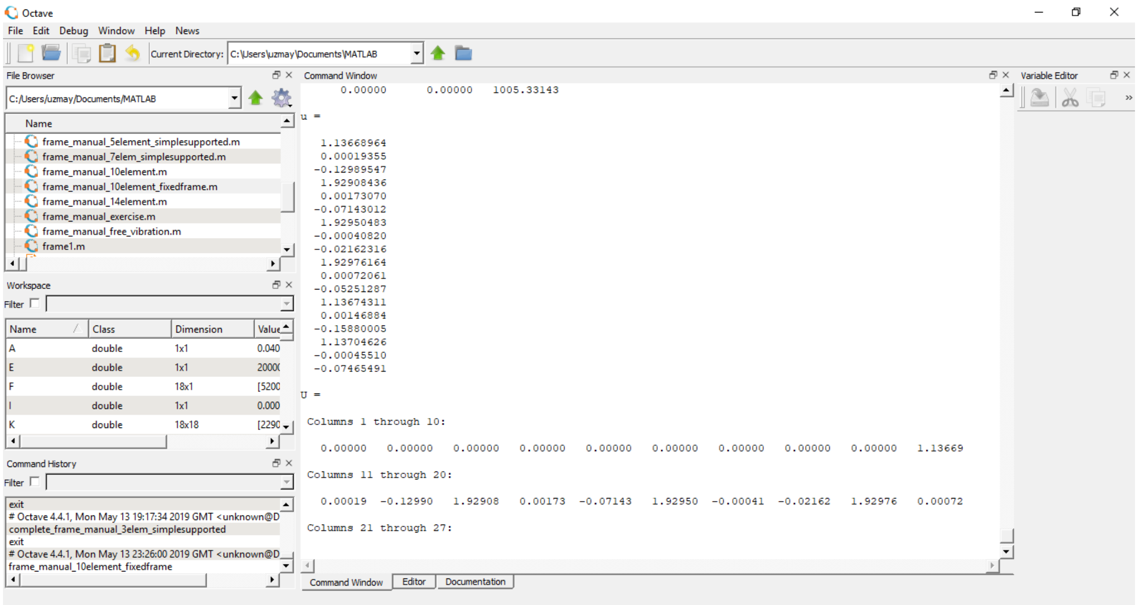


Figure 3.10 Command Windows

Command Windows is a panel where the result of the analysis is stored.

3.5 Summary

Overall, this chapter presents the method and approach used in developing Finite Element (FE) source code. The time-integration used for this study are Runge-Kutta (RK4) and Finite Central Difference (FD). The force shape function used in this study is rectangular and triangular shape. In addition, this chapter explained about the OCTAVE, open source code that have been used as the platform in developing FE source code.

CHAPTER 4

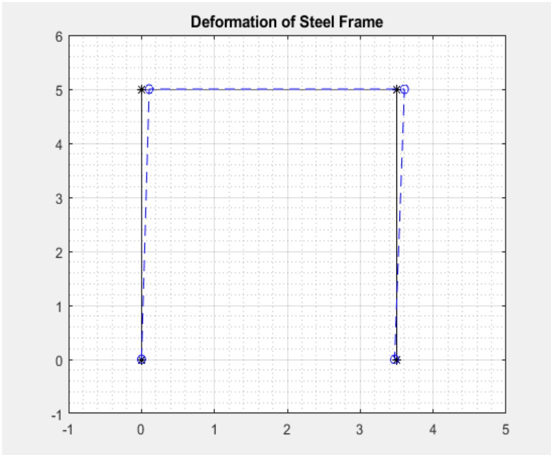
RESULTS AND DISCUSSION

4.1 Introduction

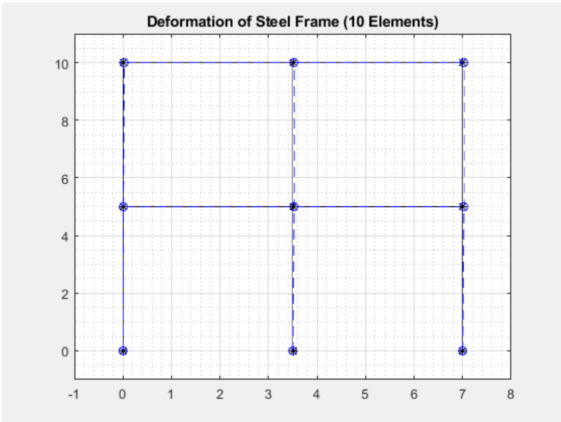
This chapter discusses on the results that has been obtained from the FE source codes that has been developed. Upon of that, the results obtained are verified with ABAQUS to assess the accuracy with the source codes that has been developed. After the verification, the results from different time-integration methods which is RK4 and FD is compared to know the reliability between them. Upon of the validation process, the behaviour of the steel frames was discussed based on the different boundary conditions and impulse loads. The results were also compared based on the guidelines of the damage levels that has been stated in American Society of Civil Engineers (ASCE) - Energy Division

4.2 Configuration of Steel Frames: Deformation

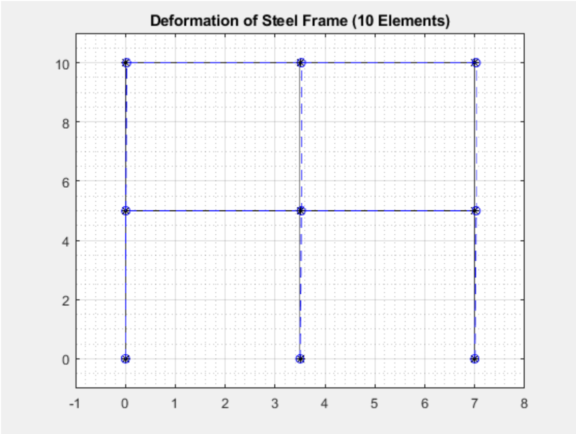
Configuration is performed to find the optimum displacements of the frames. From Figure 4.1 shown, the deformation of steel frames for 3 elements, 10 elements and 14 elements from the developed FE source code. It shown that when the number of elements of frames is increases, it will reduced the deformation of the frames. Thus, it can lessen the damage to structures.



(a) Deformation of steel frames for 3 elements



(b) Deformation of steel frames for 10 elements



(c) Deformation of steel frames for 14 elements

Figure 4.1 Deformation of the steel frames

Then, by choosing a point as shown in Figure 4.2 for 3, 10 and 14 elements, the displacement is obtained from the developed FE source codes. The results is shown in Table 4.1 and

Table 4.2 where the analysis are for simple supported steel frames and fixed steel frames respectively.

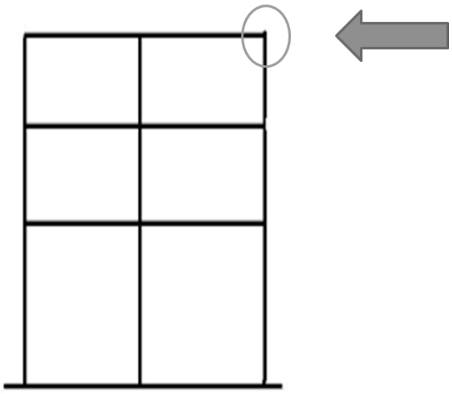


Figure 4.2 Point chosen for configuration of steel frames

Table 4.1 Simple Supported Steel Frames

	3 Elements (m)	**10 Elements (m)**	**14 Elements (m)**
Case 1	2.31	0.65	0.48
Case 2	0.61	0.17	0.13
Case 3	18.23	5.15	3.82
Case 4	4.84	1.37	1.01

Table 4.2 Fixed-Fixed Steel Frames

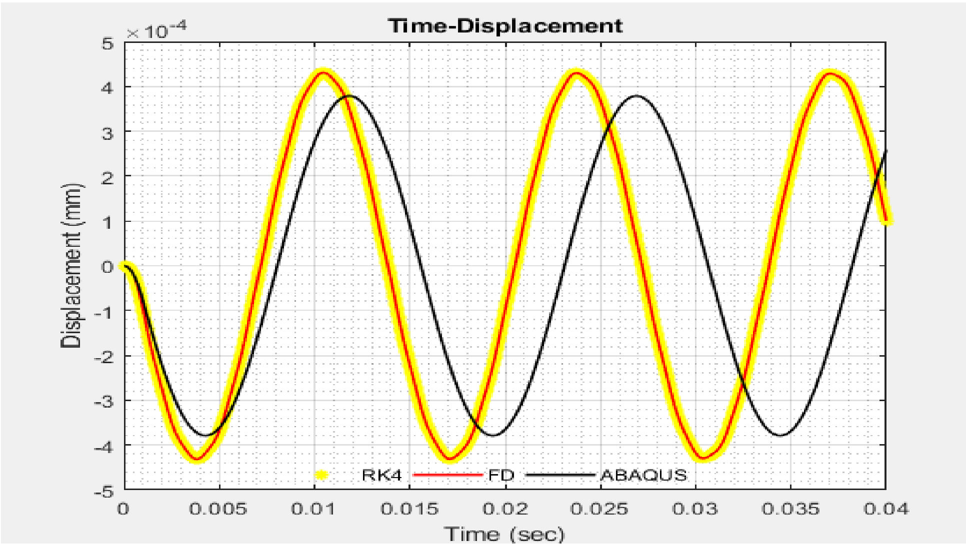
	3 Elements (m)	**10 Elements (m)**	**14 Elements (m)**
Case 1	0.40	0.16	0.13
Case 2	0.11	0.04	0.03
Case 3	3.19	1.29	0.99
Case 4	0.85	0.34	0.26

From the results obtained, when the number of elements is increased, the displacements will be reduced. In addition, different boundary condition is influenced the displacements of frames when subjected different impulse loads. It can be concluded that, when the frames is fixed frames, it produced lower displacement than when the structures is simple supported and when the higher impulse loads is applied on structures, the displacements will be increased.

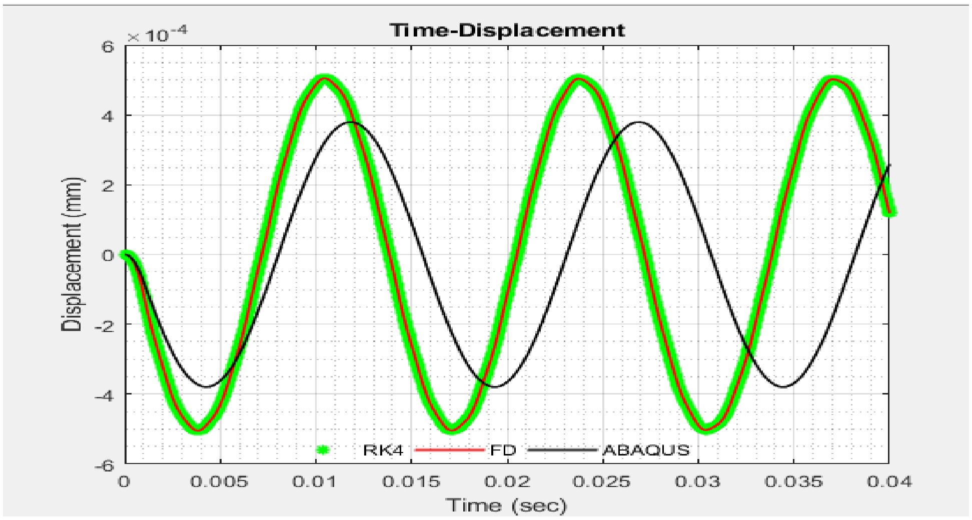
4.3 Validation of Finite Element Model

The equation of motion of the steel frames developed in Octave was solved using time-integration methods which are 4th Order Runge-Kutta (RK4) method and finite difference (FD) method according to central difference scheme as a verification process for the time-stepping procedures. The dynamic response of the frames obtain from the

RK4 and FD schemes were validated against the result from ABAQUS. The results are illustrated in Figure 4.3.



(a)

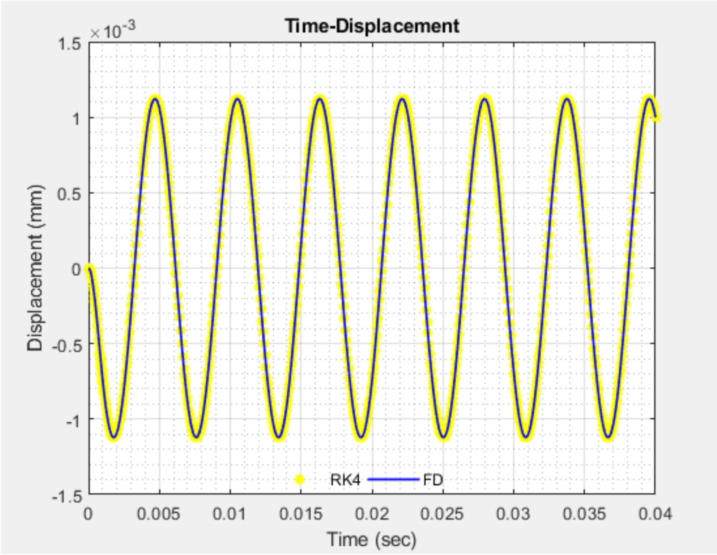


(b)

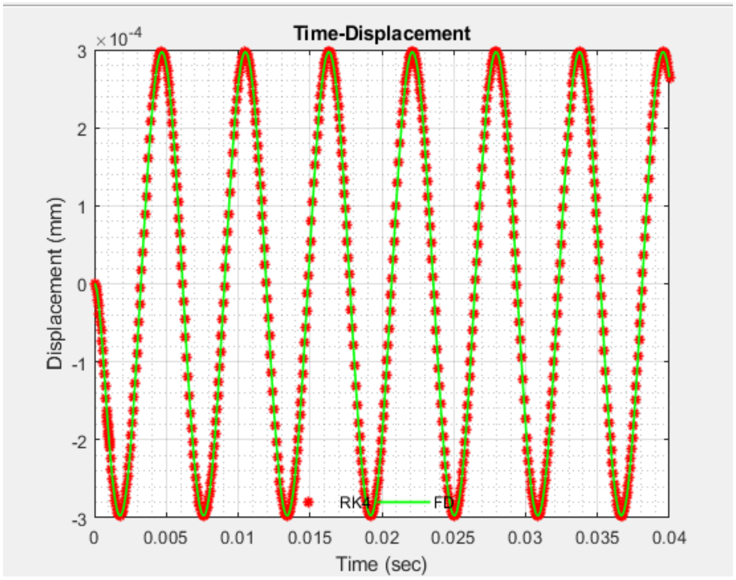
Figure 4.3 Analysis for (a) Simple Supported Steel Frames (b) Fixed Frames

According to the Figure 4.3, the result from Runge-Kutta (RK4) and Central Difference (FD) approaches have slightly difference than the result from the commercial software, ABAQUS. The error percentage between develop FE source codes with the ABAQUS is in range 2% to 10% for every cases. So, it can be concluded that the developed FE source code is satisfactory accurate.

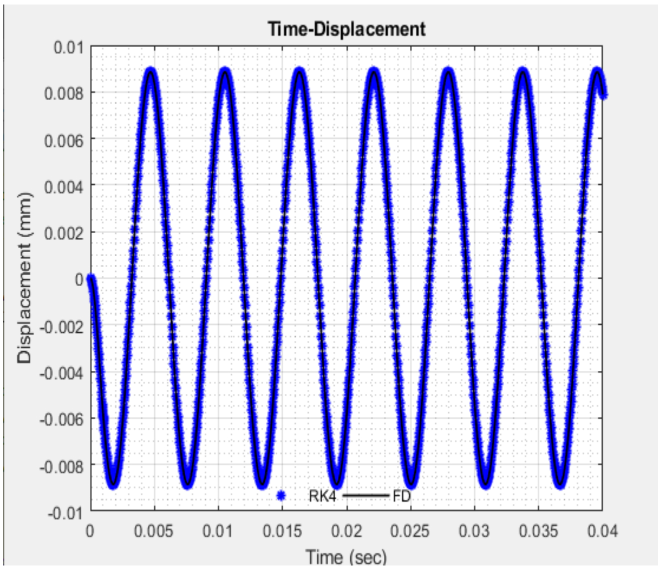
4.4 Verification of Finite Element Model (RK4 VS FD)



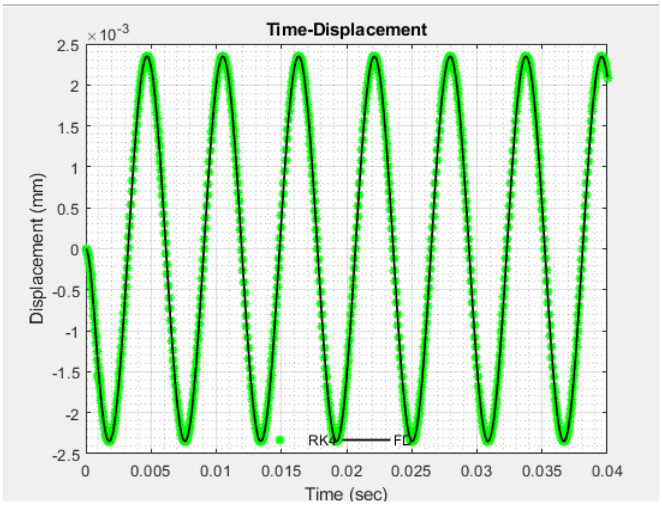
(a) Case 1



(b) Case 2



(c) Case 3



(d) Case 4

Figure 4.4 RK4 VS FD every cases

According to Figure 4.4, the Runge-Kutta (RK4) and Central Difference (FD) is compared. According to the results obtain from the develop FE source code shows, the percentage of error between RK4 and FD is about 0.05 – 0.6% for every cases. Then, the FD results from every cases is compared. It shows that, when the applied pressure in time is increase, the displacement will be increased.

4.5 Relationship between Impulse with Displacements

From the verification and validation FE model, found that impulse is linearly related to displacement for even different boundary condition and cases of impulse loading.

4.5.1 Different Boundary Conditions

For this study, the analysis is done for two different boundary condition as mentioned before.

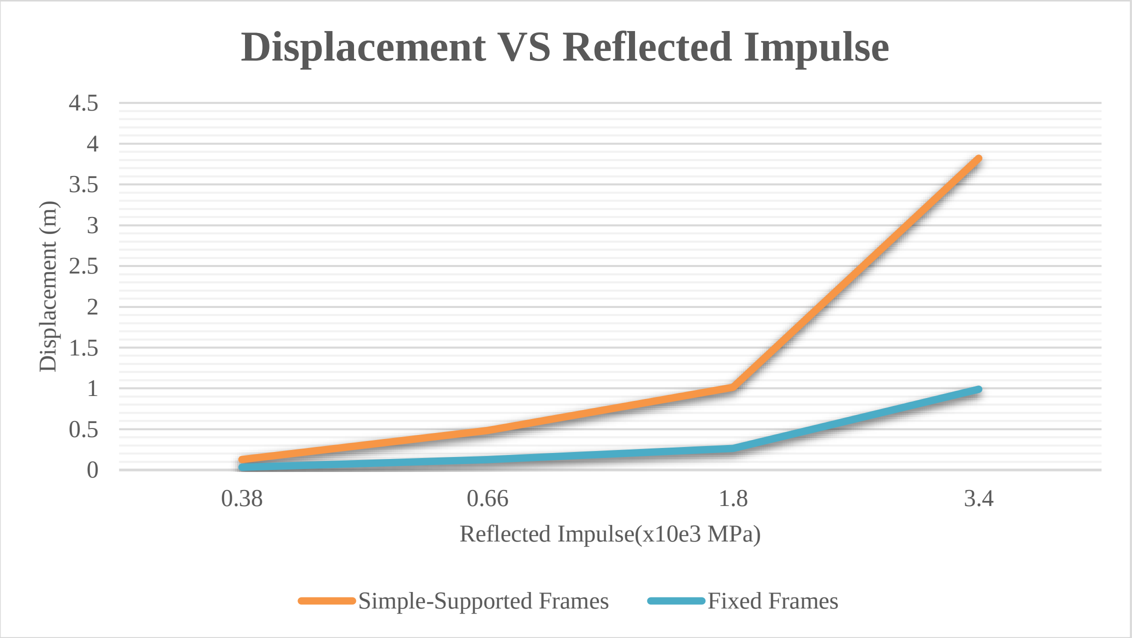


Figure 4.5 Impulse VS Displacement for fixed and simple supported steel frames

Based on the Figure 4.5, there are linear relation between impulse and displacement for every boundary conditions. However, simple-supported steel frames produced higher displacement than to the fixed steel frames.

4.5.2 Force Shape Function

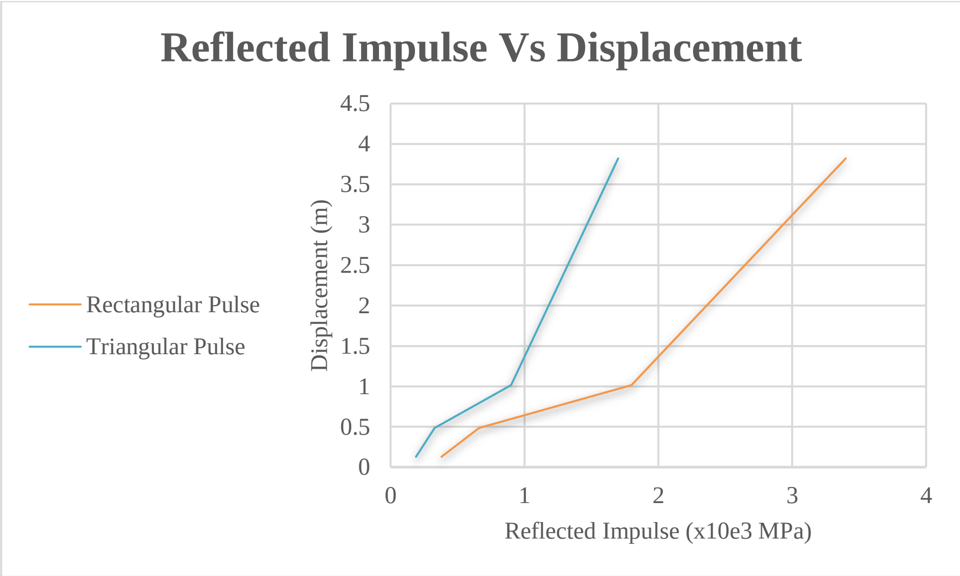


Figure 4.6 Impulse VS Displacement for rectangular and triangular force shape function

According to the Figure 4.6, the triangular force shape function has half of displacement from the rectangular force. Theoretically, the triangular force should half from the rectangular force shape function. Hence, the results also found that, impulse still linear to displacement in different force shape function.

4.6 Component Response Level

Component response for this steel frame is one of the viewpoints that can be identified. Based on the American Society of Civil Engineers (ASCE) - Energy Division and Unified Facilities Criteria (UFC 3-340-02), the component response can be identified based on the response limits in Table 4.3 using the results that has been analysed using Finite Element Analysis (FEA). Table below is extraction from the ASCE Energy Division on the component response level is categorized.

Table 4.3 Component Response Level, Source: (ASCE, 2010) (UFC, 2008)

Component Response Level	Description
Low	Component has none to slight visible permanent damage.
Medium	Component has some permanent deflection. It is generally repairable, if necessary, although replacement may be more economical and aesthetic.
High	Component has not failed, but it has significant permanent deflections causing it to be unrepairable.

	Low		**Medium**		**High**	
	μ_a	**Θ_a**	**μ_a**	**Θ_a**	**μ_a**	**Θ_a**
Steel Primary Frame Members (without significant compression)	1.5	1	3	2	6	4

From the developed FE source codes, the rotation of the frames is obtained and shown in Table 4.4, the component response level is stated.

Table 4.4 Results from developed Finite Element (FE) source code

Cases	**Simple Supported Frames**	**Fixed Frames**	**Component Response Level**
Case 1	$\theta = 3°$	$\theta = 2°$	Medium
Case 2	$\theta = 1\ °$	$\theta = 0.5°$	Low
Case 3	$\theta = 3°$	$\theta = 2°$	Medium
Case 4	$\theta = 2°$	$\theta = 1°$	Low (for Fixed frames)

According to the results obtained from developed FE source code, it shows that the steel frames fell into the low and medium response member for all the cases. Some cases will be in component has some permanent deflection and some cases will be component has none to slight visible permanent damage. Acknowledge that this study only cover linear analysis. Since there are some cases in medium class, please refer to extend study.

4.7 Summary

In conclusion, based on the results obtained, the developed FE source codes is satisfactory accurate and can be used to analyse the dynamic response to the steel frames. Other than that, the results shows that the increasing of the applied pressure can affect the increasing of the displacements towards the chosen steel frames. Besides, it also can be conclude that the boundary conditions that applied to the chosen steel frames can affect the displacements. In the next chapter will be discuss on the overall thesis conclusions and the recommendations that can be applied in the future studies.

CHAPTER 5

CONCLUSION

5.1 Introduction

Finite Element (FE) source code was developed in order to give alternative and simplest way for a beginner or designer to analyse the frames subjected to impulse loads. Hence, the first objective of this study was achieved. The behaviour of the frames can be identified and it will helps the engineer to plan and built the structure that migrate any collapse and damage. From the validation from develop FE source code and ABAQUS, it shows that only 2% to 10% error and it can be concluded that FE sources code is satisfactory accurate. Then, the verification of RK4 and FD is compared to see the accuracy between this time-integration methods. However, the results obtained shown about 0.05% to 0.6% for every cases and it can be concluded that the two time-integration can be used in this analysis. Next, the result obtained from FE source code, it can be simplified and conclude that impulse load is linearly related to displacement where when the impulse loads applied is increases, the displacements will be increased. This theory is applied to different boundary conditions and even different shape functions. From this developed sources, the behaviour of the frames can be identified and it achieved the second and third objectives where to study response or behaviour of steel frames when subjected to blast loading. Moreover, this study will helps designer to perform the simple analysis but when source code developed is complex, more time required and it gives some disadvantages for this study. Overall, this study is satisfactory accurate and can be used in the future. However, this study only cover for elastic analysis for further study please refer to other studies

REFERENCES

Agrawal, P., 2015. *Review Paper on Structures of Blast Loading and Blast Effects on Structures,* s.l.: s.n.

ASCE, 2010. Chapter 5: Dynamic Material Strength And response Criteria. In: s.l.:American Society of Civil Engineers (ASCE.

British Standard, 2005. *BS 4-1:2005 British Standard.* s.l.:s.n.

Choi, M. D. H. & Choi, M. S., 2000. Vibration Analysis for Frame Structures Using transfer of Dynamic Stiffness Coefficient. *Journal of Sound and Vibration.*

FEMA 426, December 2003. *Risk Management Reference Manual to Mitigate Potential Terrorist Attacks Against Buildings.* s.l.:FEMA .

Ferreira, A. J. M., 2009. *MATLAB Codes for Finite Element Analysis (Solid and Structures).* Porto: Springer.

Hibbeler, R. C., 2017. *Structural Analysis.* Ninth Edition in SI Unit ed. Essex: Pearson.

Ibrahim, Y. E., Ismail, M. A. & Nabil, M., 2017. Response of reinforced concrete frame under blast loading. *Procedia Engineering,* Issue 171, pp. 890-898.

Karris, S. T., 2007. *Numerical Analysis.* 3rd ed. s.l.:Orchard Publication.

Liew, R. J. Y., 2008. Survivability of Steel Frame Structures subject to blast and fire. *Journal of Constructional Steel research 64.*

Logan, Daryl L., 2012. *A First Course in the Finite Element Method.* 5th ed. Stamford: Global Engineering: Christopher M. Shortt.

Michael, G. & Daniel, J. R., 2015. *Mechanical Vibrations Theory and Applications to Structural Dynamics.* 3 ed. s.l.:John WIley & Sons, Ltd.

Mohd Yassin, A. Y. et al., 2018. *Finite Element Formulations for Statics and Dynamics of Plane Sturctures (with Matlab).* Putrajaya: Malaysian Society for Numerical Methods.

Reddy, J. N., 2006. *An Introduction To Finite Element Method.* 3rd ed. Texas: The McGraw Hill Companies.

Ruwan, J., David, P. T., Nimal, J. P. & Vladis, K., 2011. Blast and residual capacity analysis of reinforced concrete framed buildings. *Engineering Structures,* Issue 33, pp. 3483 - 3495.

S. D. B., Y. R. S. & K. V. B., 2016. Dynamic Behaviour of Frame Structure Subjected to Blast Loadings. *International Advamced Journal in Science, Engineering and Technology,* 3(8), p. 4.

Smith, P. & Hetherington, J. G., 1994. *Blast and Ballistic Loading of Structures.* s.l.:Butterworth-Helnemann Ltd..

T. N., P. M., A. G. & J. R., 2007. Blast Loading and Blast Effects on Structures - An Overview. *EJSE Special Issue: Loading on Structures.*

Unified Facilities Criteria (UFC), 2008. *Structures to resist the effects of accidental explosions.* s.l.:s.n.

United States Bomb Data Center (USBDC), 2017. *The Annual Explosives Incident Report (EIR),* s.l.: United States Bomb Data Center (USBDC).

Zheng, C., Kong, X.-s. & Wu, W.-g., 2018. Experimental and numerical studies on the dynamic response of steel plates subjected to confined blast loading.. *International Journal of Impact Engineering 113.*

Zill, D. G., 2018. *Advanced Engineering Mathematics.* 6th ed. s.l.:Jones & Bartlett Learning.

APPENDIX A
SAMPLE APPENDIX 1

APPENDIX B
SAMPLE APPENDIX 2