ANALISIS KAEDAH UNSUR TIDAK TERHINGGA TERHADAP KESAN HENTAMAN PADA RASUK KESELAMATAN PADA SISI PINTU KERETA

(FINITE ELEMENT ANALYSIS ON SIDE DOOR IMPACT BEAM)

Oleh
ALWI SHARIL FAUZAN
62074

Penyelia
DR. ROSLAN BIN AHMAD

4 Mac 2003

Disertasi ini dikemukakan kepada
Universiti Sains Malaysia
Sebagai memenuhi sebahagian daripada syarat untuk pengijazahan dengan kepujian
SARJANA MUDA KEJURUTERAAN MEKANIK

UNIVERSITI SAINS MALAYSIA

Pusat Pengajian Kejuruteraan Mekanik
Kampus Kejuruteraan

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree

Signed……………………………………………………………………………(candidate)
Date ……………………………………………………………………………

STATEMENT 1

This thesis is the result of my own investigations, except where otherwise stated. Other sources are acknowledged by giving explicit references. Bibliography/references are appended.

Signed…………………………………………………………………………… (candidate)
Date ……………………………………………………………………………

STATEMENT 2

I hereby give consent for my thesis, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available to outside organizations.

Signed……………………………………………………………………………(candidate)
Date ……………………………………………………………………………

Acknowledgement

I want to extend my heartfelt thanks to Dr Roslan Bin Ahmad for providing guidance and ideas in completing this final year project on Finite Element Analysis on Side Door Impact Beam. I am equally indebted to my fellow colleagues for giving advised, physically and morally for their support during this research especially to Muhammad Najib in teaching me the basic of using ABAQUS 6.4. I owe a special debt of gratitude to my parents for their continued love and support ever since the very first day I was born into this world. Most of all I want to express my greatest *Syukur* to Ilahi, with HIS blessing I were able to complete this thesis as intended within the given time.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLE **Page**

LIST OF FLOW CHARTS

ABSTRACT

Side Door Impact Beam is a secondary safety element in a car. It is a crashworthiness feature that works by absorbing an enormous amount of impact energy with the slightest penetration in an event of impact. The objective of this thesis is to study the mode of collapse and characterizing the most suitable material for Side Door Impact Beam when subjected to frontal impact. Parametric studies such as the effect of material and D/t ratio is also investigated in this research. The variable that has been made to the impact simulation is the type of material diameter over thickness ratio of the beam. The analysis of impact analysis on Side Door Impact Beam is done by simulation using ABAQUS 6.41. ABAQUS 6.41 is a Finite Element Software for advanced finite element analysis especially in dynamics problems. The scope of this thesis is limited and discuss thoroughly in the following aspects:

i. Maximum displacement at the time of impact subjected.
ii. Maximum stress on the critical area of impact.
iii. The amount of energy absorbed.
iv. The magnitude on contact force created due to impact.

Based on the data gathered on the vehicle, a suitable material and design for Side Door Impact Beam can be determined.

ABSTRAK

Rasuk sisi keselamatan yang dipasang pada kereta berperanan sebagai elemen keselamatan yang menahan kesan hentaman apabila berlaku perlanggaran. Ia bertindak dengan menyerap sebahagian besar tenaga dari hentaman dengan kesan pemesongan yang sedikit untuk mengelak ia dari teus sampai kepada pemandu. Tesis ini bertujuan mengkaji kesan hentaman terhadap rasuk sisi keselamatan dan menentukan bahan yang paling sesuai digunakan untuk menyerap kesan hentaman ini. Beberapa pengubahsuaian telah dibuat terhadap rasuk sisi keselamatan ini antaranya ialah dengan mengubah jenis bahan panel tersebut dan mengubah nisbah diameter terhadap ketebalan rasuk (D/t). Pengubahsuain ini dibuat untuk melihat kesannya terhadap hentaman. Analisis dinamik kesan hentaman terhadap rasuk sisi keselamatan ini dilakukan secara simulasi dengan menggunakan perisisan ABAQUS 6.41. ABAQUS 6.41 ialah suatu perisian yang berkonsepkan kaedah unsur tidak terhinngga dan ia ideal bagi menyelesaikan masalah dinamik khususnya kes yang melibatkan hentaman. Skop kajian kesan hentaman ini dihadkan kepada beberapa parameter yang tertentu sahaja. Antaranya ialah :

v. Pemesongan maksimum pada masa hentaman
vi. Kesan tegasan maksimum pada kawasan kritikal
vii. Jumlah tenaga yang diserap.
viii. Kesan daya terhadap antaramuka semasa hentaman.

Berdasarkan parameter yang telah ditetapkan ini, rekabentuk dan jenis bahan yang paling sesuai menahan kesan hentaman dapat ditentukan.

1.0 INTRODUCTION

1.1 Crashworthiness

Every car is equipped with primary safety and secondary safety element, in case of emergency if the car is involved in accidents. The term ‘secondary safety’ refers to the protection that a vehicle provides its occupants when involved in an accident, whereas the primary safety element refer the features such as braking system that should help the driver to avoid becoming involve in accidents. Table 1.0 and Table 1.1 show the primary and secondary safety in a car.

Primary safety “‘*Crash Avoidance*”	
1	Fade-resistant brakes make a major contribution to driving safety.
2	Electronically-controlled anti-lock braking system (ABS) ensures that wheels do not 'lock up' and helps to maintain full steering under hard braking.
3	Traction Control offers stable acceleration on difficult road surfaces.
4	Independent rear suspension (IRS) allows traction when accelerating, cornering and braking.
5	Powerful headlamps provide excellent beam spread and penetration.

Table 1.0 Primary Safety Element

Secondary safety “*Occupants Protection*”	
9	Integrated safety cell is engineered to withstand high energy loads.
10	Strengthened upper B pillar is computer modelled to reduce side impact velocity to head, neck and chest.
11	Front seat belts feature webbing clamps and pyrotechnic buckle pre-tensioners to minimise forward movement in a frontal collision.
12	Anti-submarining ramps, front and rear, reduce the risk of sliding under seat belt in a collision.
13	Padded, adjustable head restraints help protect against neck and head injury.
14	Driver and front passenger airbags feature low pressure inflators and are effective in reducing head, neck and chest injuries.

Table 1.1 Secondary Safety Elements

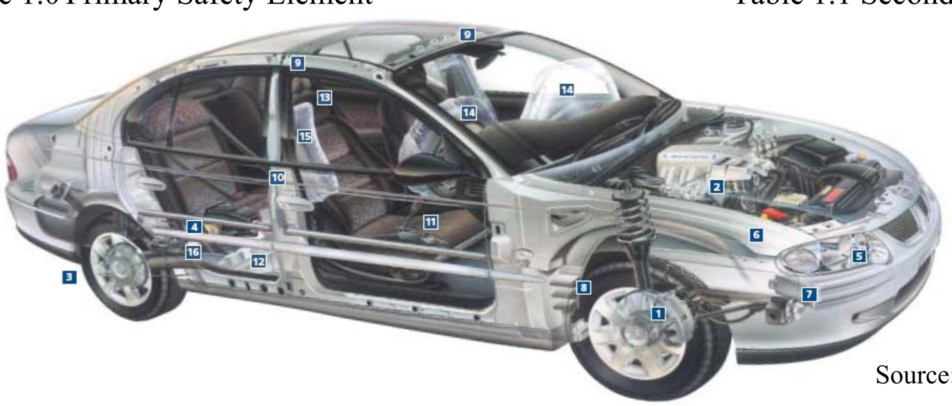


Figure 1.0 shows the location of primary and secondary safety in a car

Source: Hanenberger (2002)

The car secondary safety is also known as crashworthiness. Crashworthiness is a study of crash prediction of ductile metal structures that deformed plastically. It has become an indispensable tool for design of crash and passenger safety system. The study on crashworthiness has been carried out using various type of finite element software. ANSYS, PAM and DYNA are some of the CAE (Computer Aided Engineering) finite element software that is used in present studies of crashworthiness.

1.2 Side Door Impact Beam

Inside the vehicle side panel there is a *side door impact beam* structure. This structure is used to absorb impact in crashes. Figure 1.2 shows the location of side impact beam that are placed in the vehicle side panel

Figure 1.2 Side Impact Beam

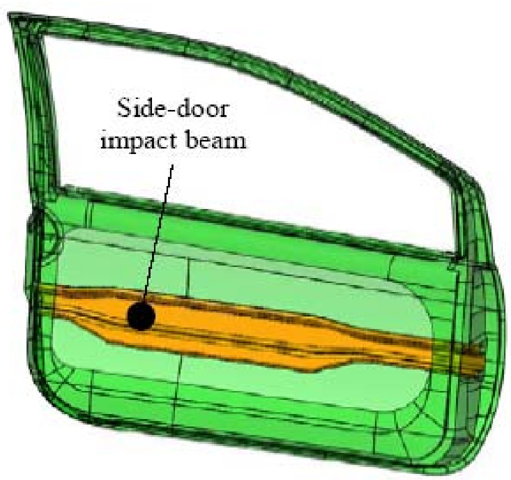
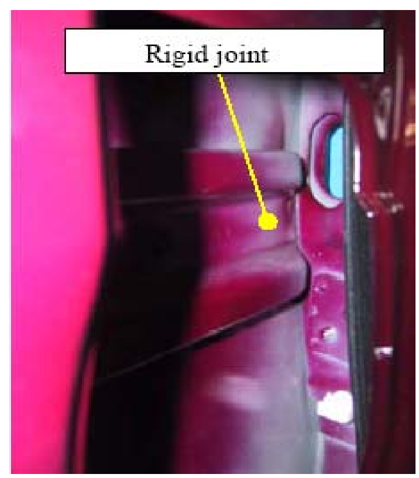


Figure 1.3 Side Impact Beam Joint



Source: S. Erzen et al (2004)

The exact position of the side door impact beam depends on the structural joints of the door and the position of the car seat. Proper placement considerably contributes to the passenger safety. The function of the safety beam in automotive applications is to provide a high level of safety for the passenger in the case of side impact by another vehicle. The beam should have the ability to absorb as much deformational energy as possible without breaking.

Figure 1.4 shows an example of a side impact crash. The side impact beam inside the door panel will absorb the impact energy and displacement from reaching the passenger. Source: Hanenberger (2002)

1.3 Objective Of Thesis

The main objective of this thesis is to conduct a finite element analysis on side door impact beam using ABAQUS 6.41. The other main objectives of this thesis

a. To learn, understand and manipulate ABAQUS to solve finite element problems.
b. To study the response and mode of collapse side panel when subjected frontal impact.
c. To find out the maximum displacement, stress, specific energy, contact force on the side impact beam during crash.
d. To characterize suitable type of materials for better absorbing capabilities
e. To perform computer simulation of impact using ABAQUS finite element software.

1.4 Scope of Thesis

There are a lot of aspect that we study in a impact analysis, however in this thesis the scope of research are limited to 2 parametric studies.

Types of material

The design of side impact beam is not changed however the type of material use for the structure is changed. The materials that are use in this study are aluminum, titanium and mild steel.

Diameter over Thickness Ratios

The D/t ratio is taken into consideration in this thesis. There are all together 3 types D/t ratios used in this simulation. The D/t ratio is also applied to the first parametric study. However in this parametric study, only the displacement effects are observed.

Based on the 2 parametric studies only the effect on displacement, stress concentration, contact force and specific energy of the material are taken into account for this thesis.

2.0 LITERATURE REVIEW

Research concerning crashworthiness analysis covers the response of the automobile structure during full frontal, offset frontal, and side impacts crash. However in this thesis, the study of crashworthiness analysis is only limited to side impact structure specifically on the side impact beam which is located in the vehicle side panel.

2.1 Equation Related to Impact Analysis

2.1.1 Failure prediction in impact equation based on failure models.

According to *K. Picketta et al (2004) IMPACT Project* , the failure prediction in impact can be solved by using material failure modeling.

Micro-mechanical approach

In their paper constitutive models with failure criteria based upon micro-mechanical void growth using the *Gurson model* and the *Gologanu model* were developed, together with the *Lemaitre mezzo-scale damage mechanics model* and the *Wilkins failure criteria*.

Comparison between models of failure

The *Gurson model* can realistically represent failure provided the loading state in the coupon used to determine the *Gurson* parameters is similar to that in the rupture zone of the structure and is predominantly hydrostatic. Improvements are possible using the *Gologanu model* to take into account an isotropic damage; however, this model introduces several additional parameters that are not easily identified. The Lemaitre model appears to give encouraging results and uses a well defined, if somewhat laborious cyclic testing procedure, to determine the model damage parameters. Finally, the *Wilkins failure model* is easily understood and relatively straightforward to calibrate against appropriate test coupons; consequently, it has been well received by industrial partners in the IMPACT project.

2.1.2 Structural intensity study of plates under low-velocity impact.

Another study regarding impact equation is brought up by Z.S. Liua et al (2004) . The structural intensity approach is used to study the transient dynamic characteristics of plate structures under low-velocity impact. In the dynamic impact response analysis, nine-node degenerated shell elements with assumed shear and membrane strain fields are adopted to model the target and impactor. The dynamic contact-impact algorithm and the governing equations for both the target and impactor are derived based on the updated Lagrangian approach.

2.2 Present Research on Side Door Impact Beam.

2.2.1 Analysis of FRP side-door impact beam

There is a large number of studies relating to use of composite material (*fiber reinforced plastic*) for side impact beam and one of them is S Eržen, Z Ren and Anžel (2003). In their research different stacking sequences of composite beam were analyzed with intention to find the most suitable solution in terms of strength, stiffness, absorbed energy and weight reduction.

2.2.2 Mechanically fastened composite side-door impact beams

Another research which are similar with S Eržen, Z Ren and I Anžel (2003) is carried out by Tae Seong Lim, Dai Gil Lee (2002). In this study, a composite side-door impact beam for passenger cars was designed to reduce the weight of steel impact beam using *glass fiber* reinforced composite. The static bending tests of the beams were performed for the optimum fiber stacking sequence, followed by the static tensile tests of the joint between the composite beam and the brackets on the car body. In order to increase the energy absorption characteristics of the composite impact beam, the mechanical joint was designed to fail with fiber shear-out mode, from which the impact energy might be dissipated during the side-door collision of passenger cars.

2.2.3 Reverse-engineering of a finite element automobile crash model

The study related to side impact was also conduct in a reverse engineering method. This research was carried out by Z.Q. Cheng et al (2001). In this paper a single model is developed and successfully used in computational simulations of full frontal, offset frontal, side, and oblique car-to-car impacts.

3.0 METHODOLOGY

Introduction

This chapter will mainly discuss on the approach that is used in analysing the side impact simulation and the failure prediction of side-door impact beam. In order to understand such analysis it is compulsory to understand the nature of the analysis which could be categorized as the Failure Mode Effect Analysis (FMEA). Basically this chapter will be discussing 2 main topics.

a) The Introduction of ABAQUS 6.43, Finite Element Software.
b) Methodology in side impact analysis.

3.1 Software Introduction

ABAQUSE 6.4 is a powerful tool for Computer Aided Engineering. To be more precise ABAQUSE 6.4 Finite Element Analysis software that could be used in various purpose of analysis. Compared to other FE software that is available in our School such as ANSYS 6.0, ABAQUSE has more extended conditions in analysis especially in terms of Impact Analysis.

ABAQUS is a suite of powerful engineering simulation programs, based on the finite element method that can solve problems ranging from relatively simple linear analyses to the most challenging non-linear simulations. ABAQUS contains an extensive library of elements that can model virtually any geometry.

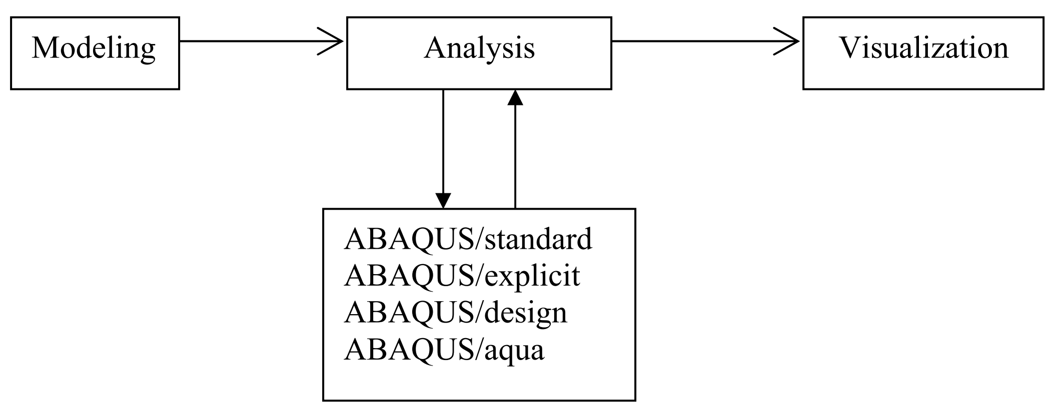
i. It has an equally extensive list of material models that can simulate the behaviour of most typical engineering materials such as metals, rubber, polymers, composite, reinforced concrete, crushable and resilient foam, and geo-technical materials such as soils and rock.
ii. It can simulate problems in such diverse areas as *heat transfer, mass diffusion*, thermal management of electrical components *(coupled thermal-electrical analyses), acoustics*, soil mechanics *(coupled pore fluid-stress analyses)*, and *piezoelectric* analysis.

iii. In terms of user friendly we could say that it is advantage in this specific aspect. Even the most complicated analyses can be modelled easily. In most simulations, including highly non-linear ones, the user need only provide the engineering data such as the geometry of the structure, its material behaviour, its boundary conditions, and the loads applied to it.

iv. In a non-linear analysis ABAQUS automatically chooses appropriate load increments and convergence tolerances.

v. It chooses the values for these parameters and continually adjusts them during the analysis to ensure that an accurate solution is obtained efficiently. The user rarely has to define parameters for controlling the numerical solution of the problem.

ABAQUS CAE Flow Chart



Flow Chart 3.0 ABAQUS CAE Flow Chart

3.1.1 ABAQUS/Standard

ABAQUS/Standard is a general-purpose analysis product that can solve a wide range of linear and non-linear problems involving the static, dynamic, thermal, and electrical response of components.

3.1.2 ABAQUS/Explicit

ABAQUS/Explicit is a special-purpose analysis product that uses an explicit dynamic finite element formulation. It is suitable for modelling brief, transient dynamic events, such as impact and blast problems, and is also very efficient for highly non-linear problems involving changing contact conditions, such as forming simulations.

3.1.3 ABAQUS/CAE

ABAQUS/*CAE* (Complete ABAQUS Environment) is an interactive, graphical environment for ABAQUS. It allows models to be created quickly and easily by producing or importing the geometry of the structure to be analysed and decomposing the geometry into mesh-able regions. Physical and material properties can be assigned to the geometry, together with loads and boundary conditions. ABAQUS/CAE contains very powerful options to mesh the geometry and to verify the resulting analysis model. Once the model is complete, ABAQUS/CAE can submit, monitor, and control the analysis jobs. The Visualization module can then be used to interpret the results

3.1.4 ABAQUS/Design

ABAQUS/Design is a set of optional capabilities that can be added to ABAQUS/Standard to perform design sensitivity calculations.

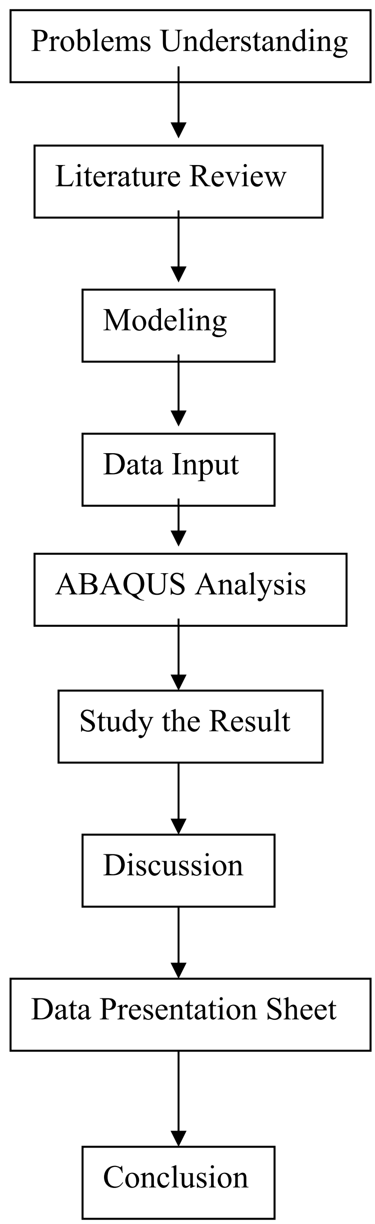
3.1.5 ABAQUS/Aqua

ABAQUS/Aqua is a set of optional capabilities that can be added to ABAQUS/Standard. It is intended for the simulation of offshore structures, such as oil platforms. Some of the optional capabilities include the effects of wave and wind loading and buoyancy

3.2 Method in Solving Problems

A systematic and analytical approach is essential in order to solve the problems that we are facing. A well and organized method in solving a difficulty determines how fast and effective the outcome of the research. Flow chart 3.1 shows the flow process of the thesis

Method in Solving Problems



Flow Chart 3.1 Method in Solving Problems

3.3 Problem Understanding

Impact analysis on side-door impact beams purpose is to study the Failure Mode Effect Analysis (FMEA). This procedure is to examine the probability of a component failed and its effect towards the performance of a system. The main objective of this thesis is to identify the critical area of impact way to overcome it by characterizing suitable material for better absorbing capabilities.

3.4 Literature Review

In this chapter previous research related to impact analysis is use as reference in conducting this thesis. Even though most of the journal does not match exactly this thesis, in terms of the how to develop models and the methodology of the research are as guidelines for this thesis.

3.5 Modelling

A car side panel includes at least 2 uprights and a lower member of the substructure of the vehicle which form a door frame that includes at least one door mounted upon the door frame and a solid lower part formed of an outer panel and an inner panel. For the purpose of modelling, the side-door impact beam in this thesis was taken from a C-Class Saloon Mercedes Benz model. There are 2 sub structures which look exactly look like a shaft are placed horizontally on the upright and lower part of the door. Figure 3.0, figure 3.1 and figure 3.2 show the part of the side-door impact beams used in this thesis.

Figure 3.0 the location of side-door impact beams in a door panel

Figure 3.1 lower part of side-door impact beams

Figure 3.2 the fix end of beam

3.5.1 Computer Aided Drawing (CAD)

Measurement

Before the drawing process using CAD is carried out the dimension side-door impact beams is taken. Below are the equipments that are used in dimensioning the side panel.

Table 3.0 Measuring Tools

No	Name
1	Vernier Caliper
2	Measurement Tape
3	Steel ruler

3.5.2 Drawing

ABAQUS is capable of producing a detail 3-D model furthermore it could even import other CAD type of drawing which is then can be used for analysis. There are altogether 8 types of CAD format support in ABAQUS:

i. ACIS SAT (.sat)
ii. IGES (.igs)
iii. VDA (.vda)
iv. STEP (.stp)
v. CATIA V4 (.model, .catdata, .exp)
vi. Pro_E Elysium Neutral (.enf)
vii. Parasolid (.x_t, .x_b, .xmt)
viii. IDEAS Elysium Neutral (.enf)

Below are the dimensions of the side-door impact beams used in the analysis. Basically the component of the side impact beam consists of three main parts:

i. Right Clamp
ii. Left Clamp
iii. Shaft Bar

The dimensions of the side-door impact beams are given in figure 3.3, figure 3.4, figure 3.5, figure 3.6 and figure 3.7. All of these parts are drawn using ABAQUS part module.

Clamp Dimension

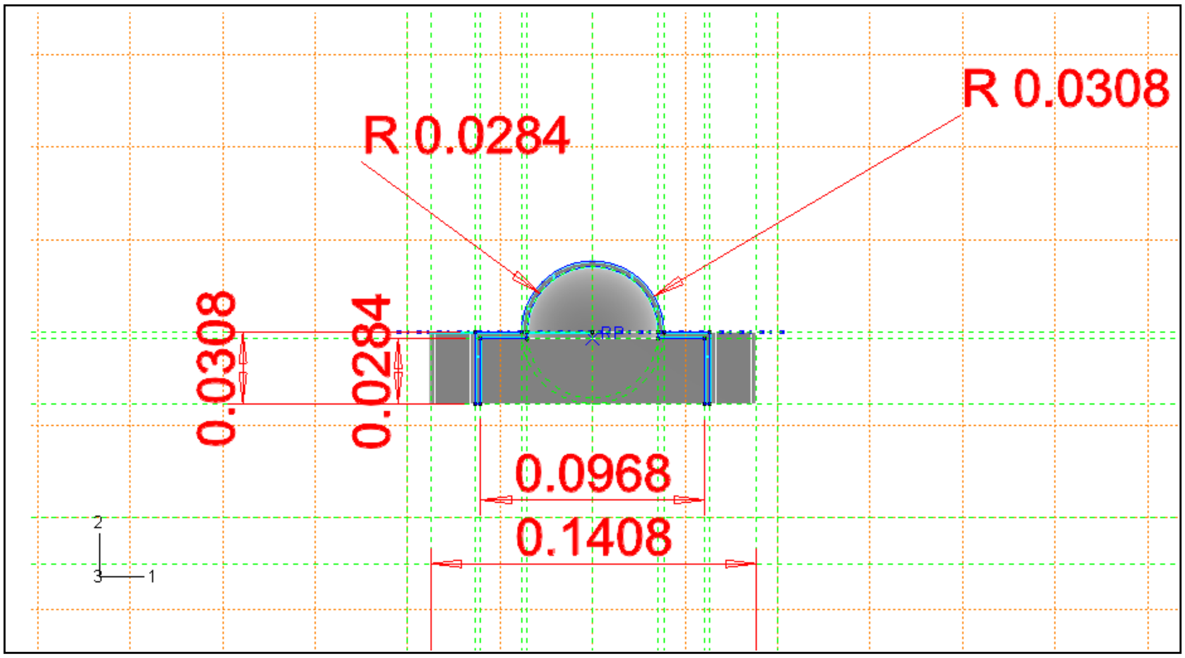


Figure 3.3 Front View of Clamp (in meters)

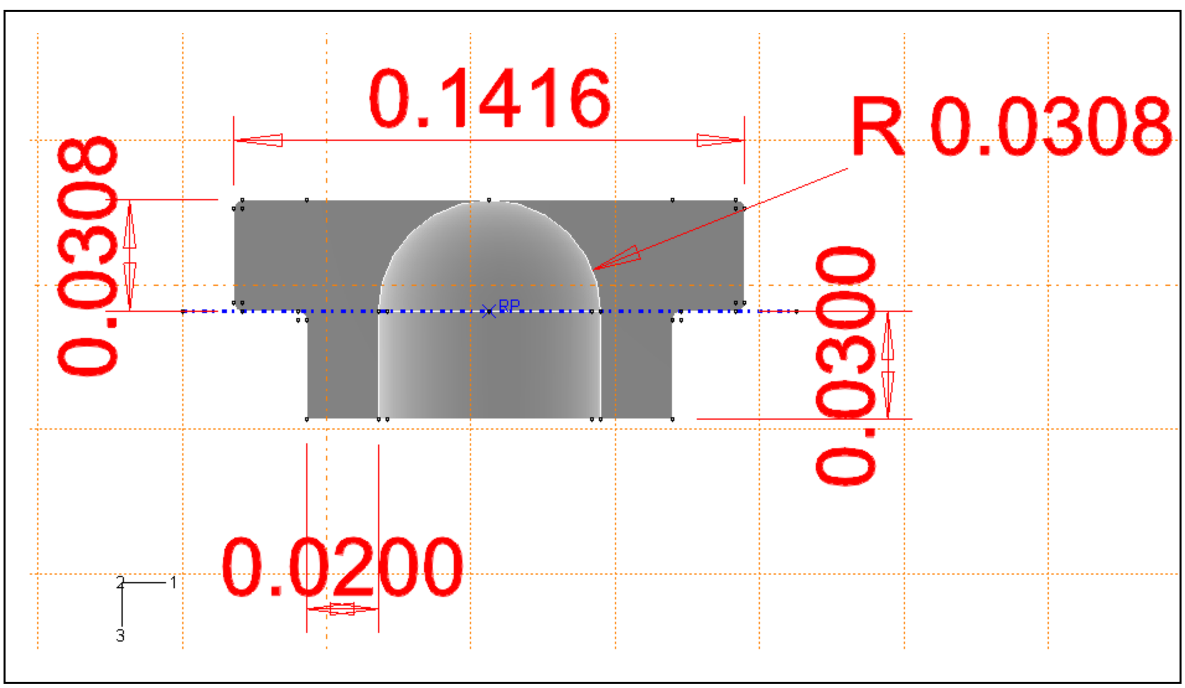


Figure 3.4 Top View of Clamp (in meters)

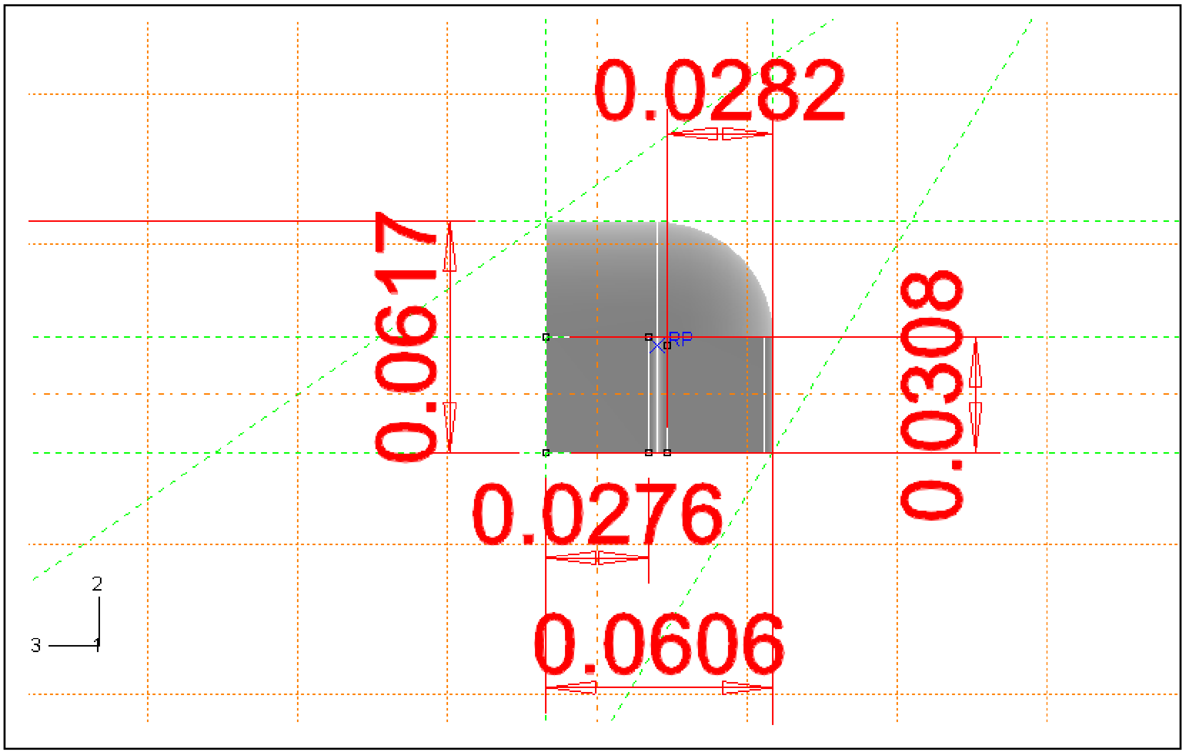


Figure 3.5 Side View of Clamp (in meters)

Beam dimension

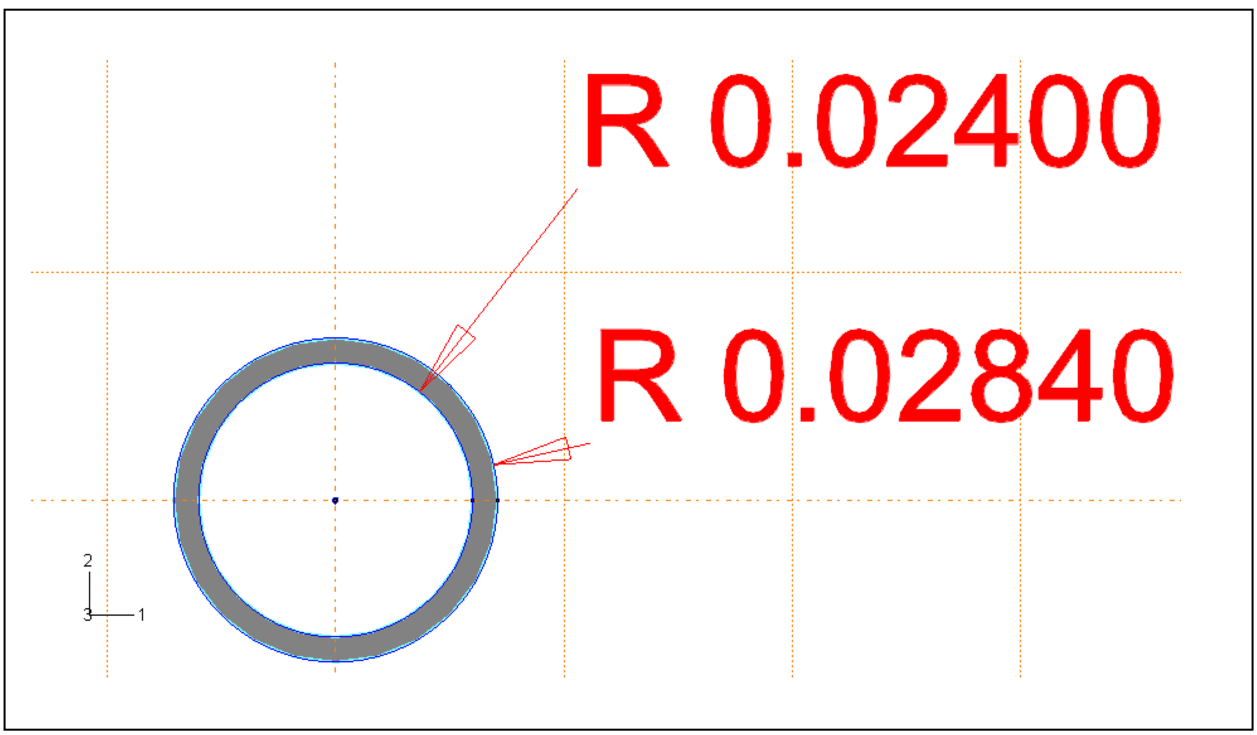


Figure 3.6 Front View of Beam (in meters)

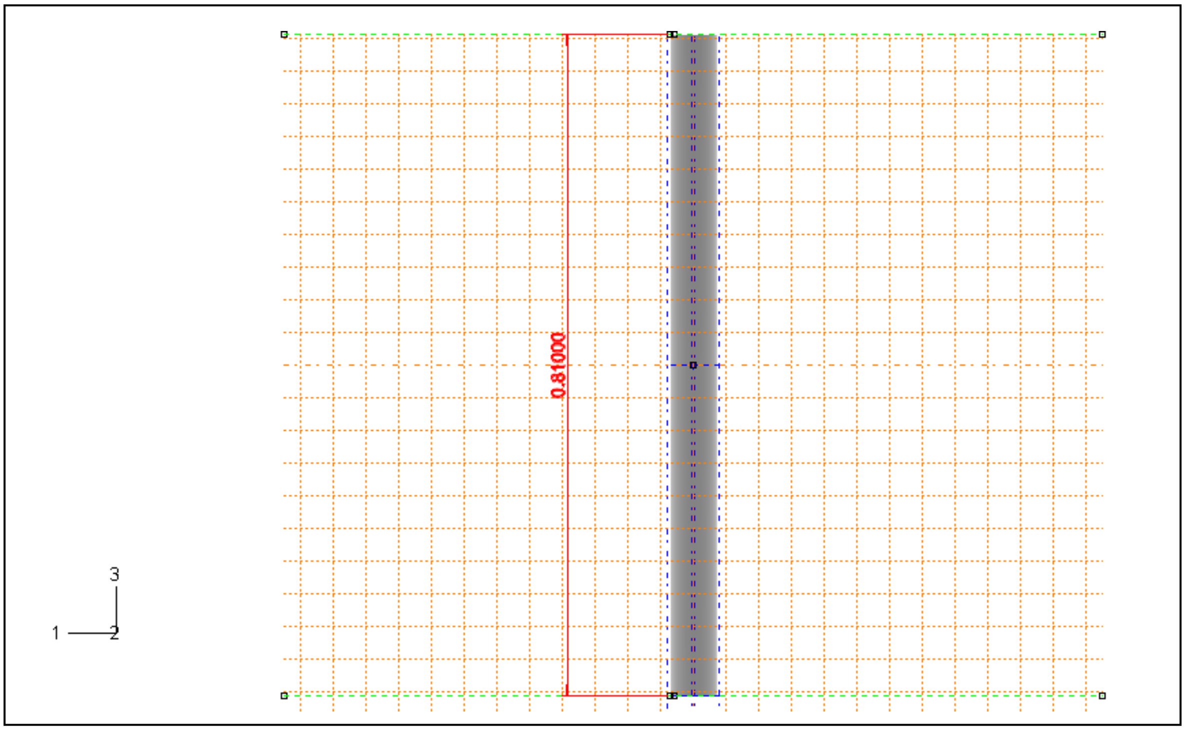


Figure 3.7 Top View of Beam (in meters)

3.6 Input Data

In this research, the experimental methodology conducted by U. N. GANDHI and S. JACK HU (2003) [5] was adapted in this simulation. Among the parameters that is used cover the following aspect:

i. Velocity of impact
ii. Mass of impacter

In ABAQUS 6.4, for conducting this impact simulation there are altogether 9 basic modules which are used in ABAQUS.

i. Parts
ii. Property
iii. Assembly
iv. Steps
v. Interaction
vi. Load
vii. Meshing
viii. Job
ix. Visualization

3.6.1 Parts

The Part module allows us to create individual parts by sketching their geometry directly in ABAQUS/CAE or by importing their geometry from other geometric modelling programs. The side-door impact beamsin this analysis came from a C-Class Saloon Mercedes Benz model. The entire dimensions in part module are in SI units. The parts of the side-door impact beams are drawn in this module even though the part can be actually drawn in other CAD software. The side-door impact beams consist of 3 main components which is the left and right clamp parts and the beam. Each and every dimension of this part is given in figure 3.3, figure 3.4, figure 3.5, figure 3.6 and figure 3.7.

3.6.2 Property

This module contains information about the properties of a part or a region of a part, such as a region's associated material definition and cross-sectional geometry. In the Property module the section that is drawn is define in terms of many behaviour and aspect. Table3.1 shows the property of material used in this analysis.

Table 3.1 Property of material

Mild Steel	Aluminum	Titanium
Modulus Young: 207GPa	Modulus Young: 70GPA	Modulus Young: 234GPa
Poisson Ratio : 0.34	Poisson Ratio : 0.33	Poisson Ratio : 0.33
Density : 7750 kg/m^3	Density : 2700kg/m^3	Density : 10000 kg/m^3

3.6.3 Assembly

When a part is created, it exists in its own coordinate system, independent of other parts in the model. Assembly module is use to create instances of parts and to position the instances relative to each other in a global coordinate system, thus creating the assembly. The part is instanced by sequentially applying position constraints that align selected faces, edges, or vertices or by applying simple translations and rotations. Figure 3.8 shows the assembly drawing of the side impact beam.

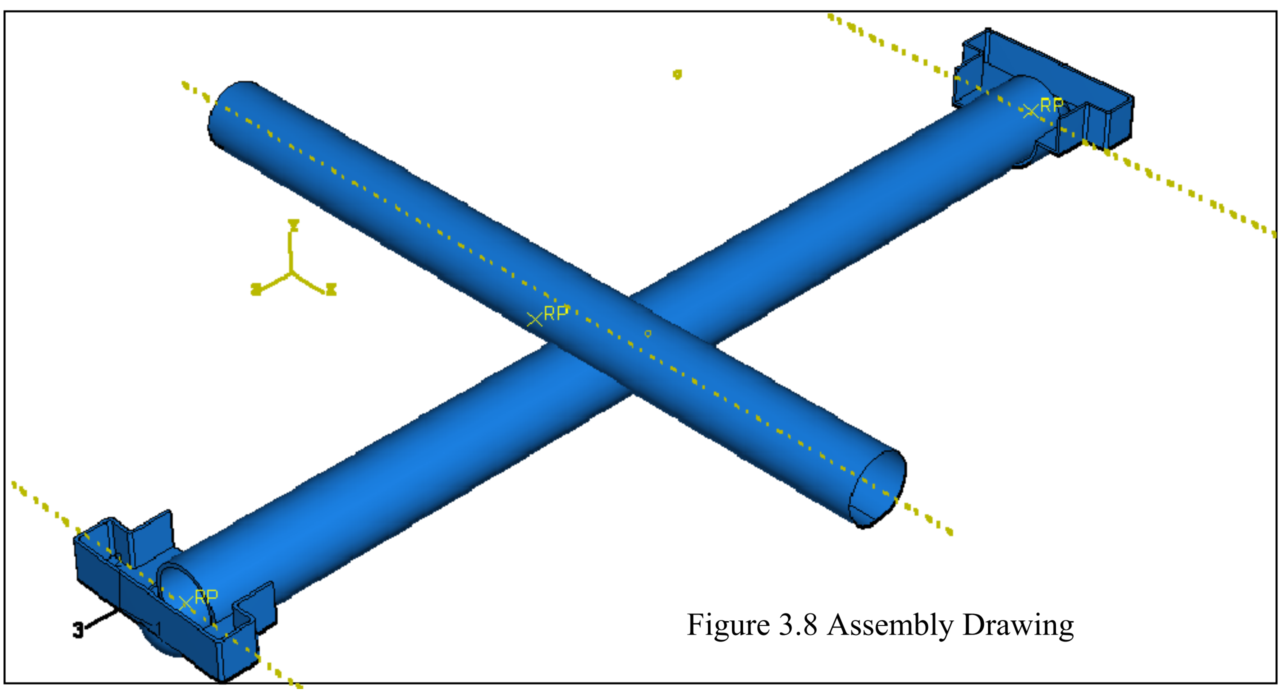


Figure 3.8 Assembly Drawing

All the parts created in the part module are instances in assembly module.

3.6.4 Step

The Step module is used to create and configure analysis steps and associated output requests. The step sequence provides a convenient way to capture changes in a model (such as loading and boundary condition changes); output requests can vary as necessary between steps. In this research the following changes below are monitored.

i. Stresses
ii. Displacement/Velocity/Acceleration
iii. Contact Force
iv. Energy

3.6.5 Interactions

In the *Interaction module* the mechanical and thermal interactions between regions of a model or between a region of a model and its surroundings is defined. An example of an interaction is contact between two surfaces. ABAQUS/CAE does not recognize mechanical contact between part instances or regions of an assembly unless that contact is specified in the *Interaction module*; the mere physical proximity of two surfaces in an assembly is not sufficient to indicate any type of interaction between the surfaces. Interactions are step-dependent objects, which mean that we must specify the analysis steps in which they are active. Figure 3.9 shows the interaction between the beam and the rigid body is defined.

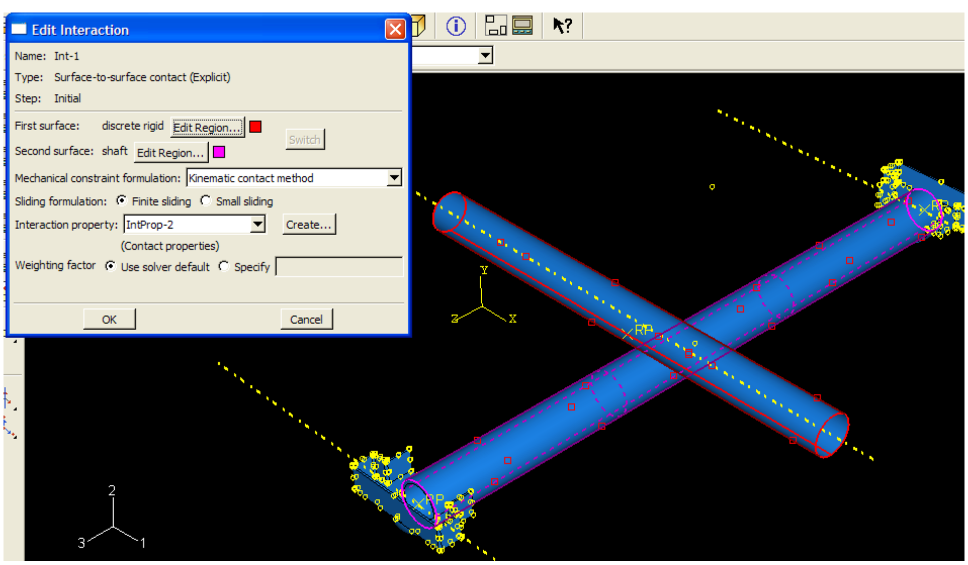


Figure 3.9 Interaction Property

Because of the side impact analysis only involve two main parts during impact; the interaction is only defined between the beam and the rigid body which will collide with each other. The discrete rigid part is defined as the *master surface* and the beam is defined as the *slave surface*.

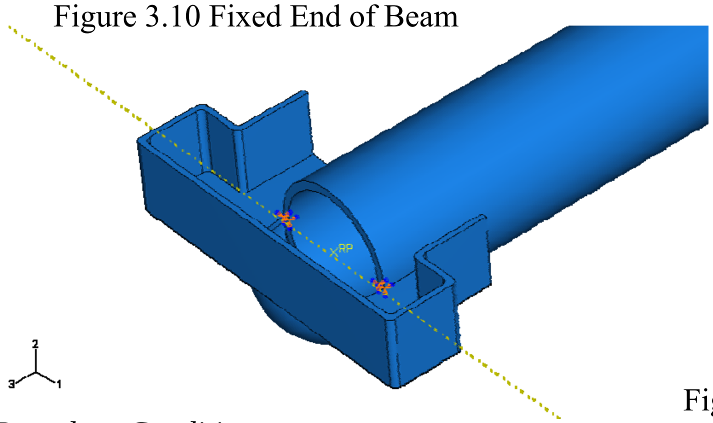
3.6.6 Load

The Load module is used to specify loads, boundary conditions, and fields. Loads and boundary conditions are step-dependent objects, which mean that the analysis steps must be specify in which they are active; some fields are step-dependent, while others are applied only at the beginning of the analysis. In this analysis only one step is used which is the impact velocity.

In this experiment the velocity field that is used is *v =10 m/s* which is equivalent to *36 km/h.* The momentum that is given to the side-door impact beams is automatically calculated. The information regarding the Mass, Density, Modulus Young, Poisson ratio, type of analysis are the key in the Property module.

In this module the *boundary condition* of the analysis is defined. Boundary conditions are the parameter that we used to define the nature of our analysis. Condition such as how a medium are fixed, the degree of freedom permitted for deformation.

Figure 3.10 Fixed End of Beam



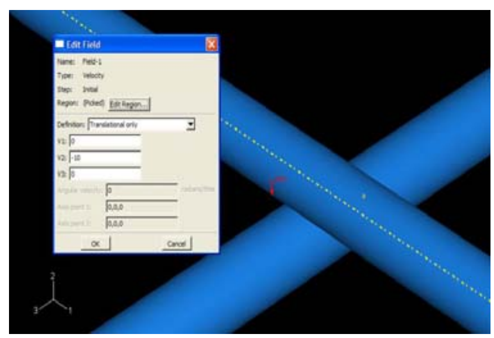


Figure 3.11 Impacter And Beam Interaction

Boundary Condition

i. Figure 3.10 shows the end of the shaft are constrain in ENCASTRE (U1=U2=U3=UR1=UR2=UR3=0). The reason is to constrain the shaft from further deflection in case of impact.

ii. The rigid body are constraint in all degree of freedom except in Vector 2 direction as shown in figure 3.11.

3.6.7 Meshing

The Mesh module contains tools that is use to generate a finite element mesh on an assembly created within ABAQUS/CAE. Various levels of automation and control are available so that you can create a mesh that meets the needs of your analysis. The purpose of meshing is to define the type of element that is use to run the analysis. There are together 3 types of element that is use in this analysis. Figure 3.12 shows the type of element used for meshing for side impact beam.

Table 3.2 Elements of Parts

Part	Element type	Description
Shaft	C3D8R	8-nodetrilinear displacement and temperature, reduced integration with hourglass control
Rigid Body	R3D4	4-node, bilinear quadrilateral
Clamp	C3D4	4-node linear tetrahedron

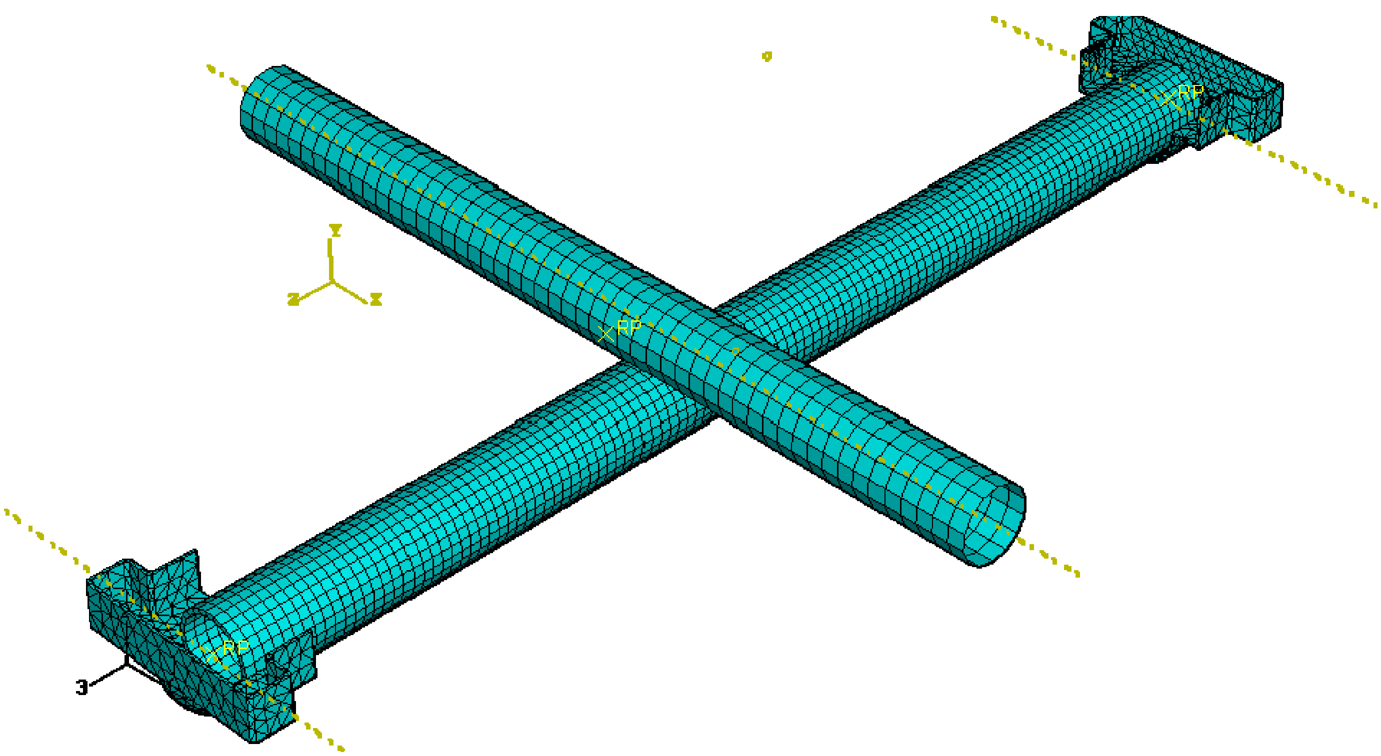


Figure 3.12 Types of Meshing Element Use in simulation

3.6.8 Job

The Job module is use to analyse your model based on the previous module task that has been completed. Basically the Job module interactively submits a job for analysis and monitors its progress. The job that is completed is stored as in .ODB file, which also known as the output files. The time that is use for analysis depends on the type of analysis that is being carried out, the output requested, no of element that involved in calculation and the type of element that we used in the analysis. Actually the Job module stored the output of our analysis, every time the value in the module is changed a new job is run task to get the new output The job that has been carried out is useful for comparison.

3.6.9 Visualization

The Visualization module provides graphical display of finite element models and results. It obtains model and result information from the output database; we can control what information is written to the output database by modifying output requests in the Step module.

This module is use to interpret output data in such statistically manner. However it depends on the step module history request that we set. Visualization allows us to create various type of view such as contour, deformed and un-deformed wire frame model. It also allows us to view the simulation of the impact frame by frame at the speed of our need.

Here in this module I present my result in various type of way

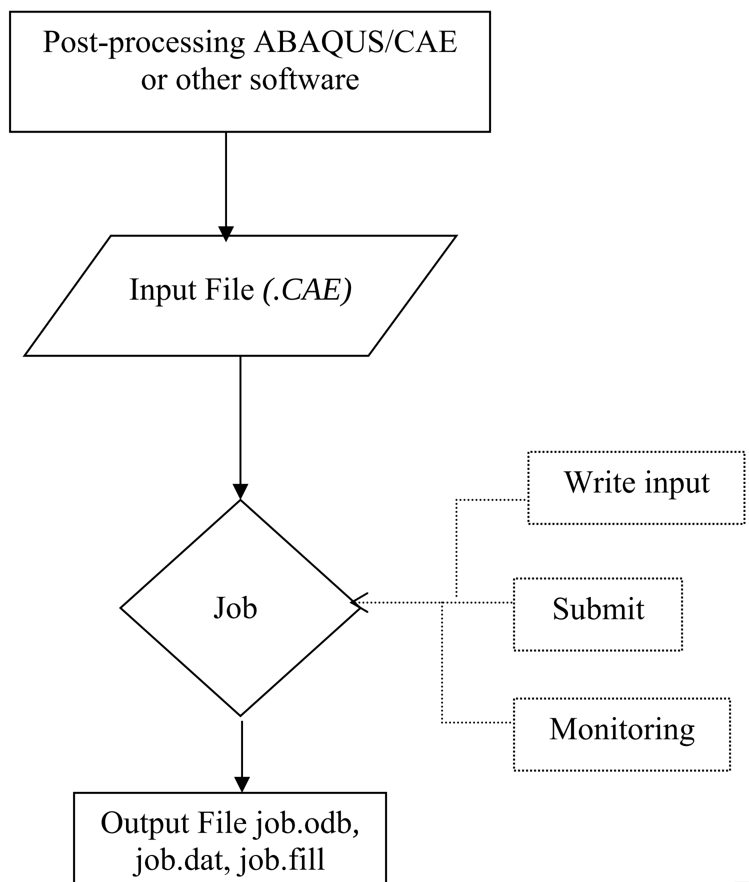
i. X-Y graphs
ii. Contour
iii. Sectioning

3.7 ABAQUS Analysis

As mentioned in previous chapter ABAQUS product range from various types of analysis, however specifically for impact the ABAQUS/Explicit. The module that has been completed is merely the input file which ABAQUS will store the input file as *.CAE* file. Several file will automatically be generated during analysis is being carried out. The output file is saved in ODB file. Table 3.3 summarize the type of file being used and produced during and after ABAQUS analysis..

Table 3.3 Types of Files

Before Analysis (Input File)	During Analysis (Output file)	After Analysis (Output file)
.CAE	.ABQ, rpy, .DAT, .INP, .IPM, .MDL, .PAC, .PRT, .SEL, .STA, .STT	.ODB



Flow Chart 3.2

Flow chart 3.2 shows the flow of analysis in ABAQUS