NON-LINEAR CONTACT FINITE ELEMENT ANALYSIS OF SPLIT HOPKINSON INCIDENT BARS UNDER IMPACT LOAD

By:

NG YIE YEN

(Matrix no: 140108)

Supervisor:

Ir. Dr. Feizal Bin Yusof

July 2021

This dissertation is submitted to
Universiti Sains Malaysia
As partial fulfilment of the requirement to graduate with an honours degree in
BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

School of Mechanical Engineering

Engineering Campus

Universiti Sains Malaysia

DECLARATION

I hereby declare that this work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed: ________________________

(NG YIE YEN)

Date: 12/7/2021

Statement 1

The work reported in my thesis is the result of my investigation, except where otherwise stated. Other sources are acknowledged by giving explicit references. Bibliography/references are appended.

Signed: ________________________

(NG YIE YEN)

Date: 12/7/2021

Statement 2

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

Signed: ________________________

(NG YIE YEN)

Date: 12/7/2021

ACKNOWLEDGEMENT

First and foremost, I would like to express my utmost appreciation to my supervisor, Ir. Dr Feizal Bin Yusof, for his guidance, support, and patience throughout the project. He had contributed a lot of helpful advice and suggestions throughout the whole simulation process. He also helped me a lot by providing functional theory and patiently explaining the theory behind that will help me in the project. It was a great privilege and honour to work under his guidance.

Next, I would like to thank the School of Mechanical Engineering for giving me a platform to conduct my work and providing facilities for me to complete my work. Besides, I would like to thank Dr Muhammad Fauzinizam Bin Razali, the EMD452 Final Year Project coordinator, who helped arrange workshops for the students. Not forget to thank the CAD lab technician, Mr Jamari Sadli, who in charge of taking care of the computers in the CAD lab. He was endlessly providing support in terms of facilities. He is ensuring that the facilities are working well, which enables me to conduct my simulation smoothly.

Moreover, I would like to thank my friends who have given me a helping hand whenever I needed them. Finally, I must express my gratitude to my family for providing me with their support and encouragement throughout my years of study.

TABLE OF CONTENTS

LIST OF TABLES

Page

LIST OF FIGURES

LIST OF SYMBOLS/ ABBREVIATIONS

A_b	Cross-sectional area of bar
A_o	Initial cross-sectional area of the material
A_{st}	Cross-sectional area of striker
C_b	Wave speed through the bar
C_d	Drag coefficient
D_b	Diameter of input bar
D_p	Inner diameter of pipe
d_i	Inner diameter of the striker
d_o	Outer diameter of the striker
E	Young's modulus
f	Friction factor
F	Force applied to the material
F_D	Drag Force
F_k	Kinematic friction force
F_y	Load at yielding
g	Gravitational force
H_f	Flange's height
k	Constant for virgin material
L	Current length of the material
L_f	Flange's length
L_p	Length of pipe
L_o	Initial length of the material
$L_{striker}$	Length of striker
L/D	Length-to-diameter ratio of the bar
m	Mass of object
N	Normal force
Re	Reynold number
S	Principal deviatoric stress
T_s	Thickness of striker
t_{total}	Total time of one test
V	Velocity of fluid

ν	Kinematic viscosity
ε	Strain
$\acute{\varepsilon}$	Strain rate
ε_a	Absolute roughness
ε_i	Incident strain
ε_r	Reflected strain
ε_t	Transmission strain
ρ	Density
t_{bar}	Total time for the wave to travel through the striker bar one time
F_{st}	Maximum force applied to striker
P	Maximum pressure that can be exerted by the pressure tank
ΔP	Pressure loss
σ	Stress
σ_{op}	Operable stress
σ_v	von Mises stress
σ_w	Allowable working stress
σ_y	Yield strength/ yield stress
μ_k	Kinematic coefficient of friction
DOF	Degree of Freedom
L/D	Length to diameter ratio of the incident bar
SHPB	Split Hopkinson Pressure Bar

ABSTRAK

Sistem pengujian kadar tegangan tinggi adalah penting untuk menentukan sama ada reka bentuk komponen dapat menahan hentaman. Oleh itu, pembinaan Split Hopkinson Pressure Bar (SHPB) adalah penting untuk mengkaji sifat bahan pada kadar tegangan yang tinggi. Namun, setelah kajian litertur diadakan didapati bahawa tiada pernerbitan yang benar-benar memberikan garis panduan reka bentuk khusus untuk menentukan parameter sesuatu Tensile SHPB dan untuk mencirikan tekanan yang dialami oleh komponen SHPB.

Tujuan projek ini adalah untuk mencirikan tekanan pada bar insiden dan menggunakan hasil simulasi untuk mencadangkan garis panduan reka bentuk untuk pembinaan Tensile SHPB berskala kecil dengan menggunakan Ansys Mechanical APDL. Untuk menjalankan analisis, penciptaan geometri dalam simulasi mengikuti bahan dan dimensi Tensile SHPB sebenar yang sudah dibina di makmal Kejuruteraan Mekanikal. Pendekatan untuk analisis finite elemen didasarkan pada mekanik kontak untuk mensimulasikan prinsip kerja SHPB.

Penyediaan eksperimen dalam konfigurasi pengujian yang berkaitan digunakan sebagai parameter pemuatan dalam simulasi. Dengan menggunakan konfigurasi pemuatan ini, model finite elemen menunjukkan bahawa tekanan maksimum yang dialami oleh bar insiden dalam keadaan tanpa geseran ialah 677MPa. Setelah itu, kehilangan tekanan dan geseran diperkirakan membuat hasil simulasi menghasilkan hasil yang masuk akal, dan tekanan 403.50MPa diperoleh dari pengiraan. Lebih-lebih lagi, SHPB berskala kecil ditubuh agar boleh dipasang pada meja biasa berukuran 1.8m x 1.2m dengan menggunakan nisbah tekanan σ_{op}/σ_y dan nisbah L/D. Nisbah tekanan juga digunakan untuk menentukan had tekanan bahan SHPB berskala kecil. AISI 4340 quenched dan tempered steel dipilih kerana had tekanan yang tinggi dan harganya yang berpatutan.

Oleh yang demikian, dapat disimpulkan bahawa simulasi untuk memodelkan tekanan yang dialami oleh bar insiden dalam keadaan tanpa geseran dapat dibinakan, dan teknik nisbah tekanan dapat digunakan dalam reka bentuk SHPB bersamaan dengan nisbah L/D.

ABSTRACT

A high strain rate testing system is important to define whether the component's design can resist impact loading. Therefore, Split Hopkinson Pressure Bar (SHPB) development is important for studying material behaviour at high strain rates. However, from the literature review conducted in this project, it was found that no publication gave a specific guideline to determine the design parameters of a Tensile SHPB and to characterize the stresses experienced by the SHPB's components.

The purpose of this project was to characterize the stresses of an incident bar and use the simulation result to propose a design guideline for the development of a small-scale Tensile SHPB by using Ansys Mechanical APDL. To carry out the analysis, the geometry created in the simulation followed the actual material and dimension of Tensile SHPB developed in the Mechanical Engineering laboratory. The approach for the finite element analysis was based on contact mechanics to simulate the working principle of the SHPB.

An experimental setup in a related testing configuration was used as the loading parameter in the simulation. Using this loading configuration, the finite element model demonstrated that the maximum stress experienced by the incident bar under frictionless conditions was 677MPa. Subsequently, pressure and friction losses were estimated to make the simulation a sensible outcome; the stress of 403.50MPa was obtained from the calculation. Moreover, small-scale SHPB was set to fit onto a 1.8m x 1.2m regular table using the stress ratio σ_{op}/σ_{y} and the L/D ratio. The stress ratio was also used to decide the small-scale SHPB material's yield stress and AISI 4340 quenched and tempered steel was selected due to its high yield stress and affordable price.

In conclusion, the simulation to model the stress experiencing by the incident bar under frictionless conditions can be developed, and the stress ratio technique can be used in SHPB design in conjunction with the L/D ratio.

CHAPTER 1

INTRODUCTION

1.1 Overview of Project

Impact is a high force or shock applied over a short period when two or more bodies collide [1]. If a material encounters an impact, the loading time is shortened, and the material inertia effects become important, and the loading becomes dynamic. Thus, the material needs to be tested using a high strain rate. On the other hand, suppose the material is tested by a low strain rate. In that case, the design somewhere is limited because quasi-static stress-strain data may not produce accurate and reliable predictions of material and product performance at a high strain rate [2]. According to Yu et al. [3], the difference between the material behaviour under high strain rate and low strain rate testing is that the dynamic strength or yield strength increases as the strain rate increases.

A high strain rate testing system is important for many structural engineering applications that undergo the impact of high strain rate deformation, such as automotive and aerospace crashworthiness [2], but high strain rate testing is not limited to these. However, it can also be found in everyday things that need to function after experiencing some form of impact event such as handphone dropped to floor or concrete, or any other mobile devices. Therefore, developing a high strain rate testing system is very important to characterize materials subjected to dynamic loading conditions. The Split Hopkinson Pressure Bar (SHPB) test is a commonly used method for determining the dynamic mechanical properties at high strain rates [4]. The SHPB works on the principle of one-dimensional wave propagation to ensure the load from impact can be transferred to the specimen.

The design guideline of the SHPB machine is important to ensure the stress wave of SHPB to be a one-dimensional wave. The SHPB should be designed to be rigid enough to prevent machine deformation under applied load. Thus, the SHPB machine’s material and dimension are important in designing an operable machine. However, from the literature review conducted in this project, it was found that no publication gave a specific discussion on the design criteria for the SHPB components.

There was no specific dimension stated to design an SHPB. It was found that the general guide is that an SHPB system should be based on a ratio of the incident bar length to the incident bar diameter, $L/D \geq 20$. It was arguing that this ratio will allow a one-dimensional stress wave to develop in the bars to calculate the stress and strain of the characterized material.

Traditionally, the prediction of how a structure behaved is described by governing equations that can be solved via closed-form solutions. However, this can only be done for simple part shapes. Finite Element Analysis (FEA) enables engineers to model complex structures by making realistic assumptions that can predict the behaviours of the structure [5]. Thus, numerical methods through the finite element method can be used to simulate the Split Hopkinson Pressure Bar (SHPB). According to Kim [6], when two or more bodies collide, there is contact between the surfaces of the bodies so that they cannot overlap in space. Contact analysis can help to obtain the stress that occurs in the contact interface of the structure. Stress state is important for a structure because it can help define its ability to hold the load being applied to it. Thus, a finite element contact analysis will be developed based on a tensile SHPB machine to characterize the stresses experienced by the bar.

1.2 Problem Statement

To design a Split Hopkinson Pressure Bar (SHPB) that is operable, there are proper design guidelines that were needed to comply with deciding on the parameters of the machine, such as the machine's material and the dimension of the machine. However, based on the literature review conducted in this project, there was no specific design guideline suggestion to design the Tensile SHPB. The available suggestions were merely based on a qualitative design approach instead of a quantitative design approach. Besides, there was no method available to characterize the stresses experienced by the components as a design guideline in a typical Split Hopkinson Pressure Bar. Thus, Finite Element Analysis for contact problems needs to be done to characterize stresses of the components of a Tensile SHPB as a design guideline for the development of small-scale Tensile SHPB.

1.3 Project Objectives

- To characterize stresses experienced by the components of a Tensile Split Hopkinson Pressure Bar (SHPB).
- To propose a non-linear contact finite element analysis of Tensile SHPB by using ANSYS Mechanical APDL.
- To propose a small-scale Tensile SHPB system design by modifying the dimension of standard size Tensile SHPB to develop small-scale Tensile SHPB.

1.4 Scope of Work

In this project, the scope is on simulation, which needs to be done by using ANSYS Mechanical APDL software based on the specifications of the designed Tensile SHPB [7]. Contact analysis was done by developing the finite element model first. The striker, the incident bar and the anvil were created in ANSYS based on the material (AISI 4140 Alloy Steel) and the actual dimension. After that, contact pairs were created with different contact behaviours (no separation, bonded and standard) between the surface of the components to make sure that the striker can slide along the bar, the anvil cannot move from its original position and no overlapping of surfaces during impact. Moreover, the load needed to be applied to the striker in the simulation. Constraints were applied to the components to make sure the components move in only one direction, and the momentum stopper acts as fixed support. Lastly, the simulation result, which was the stresses of the incident bar, is used as a design guideline for developing a small-scale Tensile SHPB.

CHAPTER 2

LITERATURE REVIEW

2.1 Overview of Tensile Split Hopkinson Pressure Bar (SHPB)

Split Hopkinson Pressure Bar is one of the methods to characterize the mechanical response of materials that deform at high strain rates. The setup of tensile SHPB, as shown in Figure 2.1, was discussed in Goh's thesis [7]. The components of the tensile SHPB consist of a gas launcher, incident bar, transmission bar, anvil, striker, momentum stopper, support frame, and bar support. The striker sits in the gas launcher, and the specimen is placed between the transmission bar and the incident bar. The working principle of the developed tensile SHPB started with the pressure tank. During testing, the pressurized air from the pressure line is directed into the gas launcher and then launch the hollow striker. The striker is slide along the incident bar and hit the anvil. At the same time, the specimen is pulled, and the transmission bar will move. The specimen is subjected to dynamic tensile loading. The whole process stopped after the incident bar collides with the stopper.

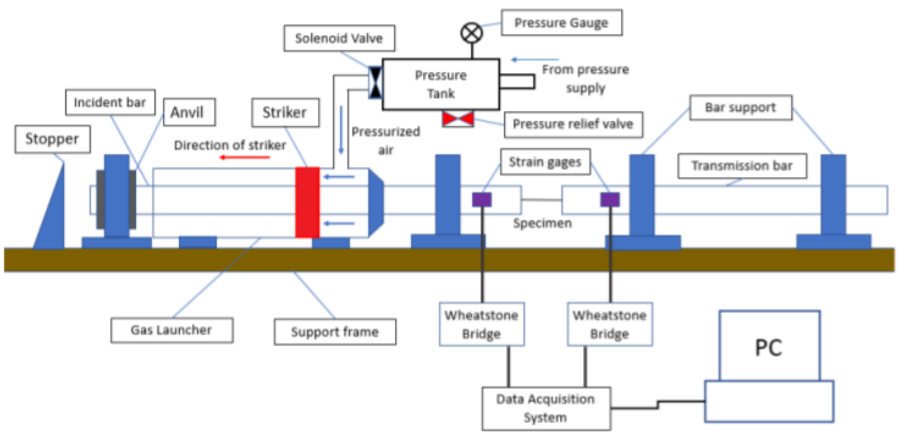


Figure 2.1 Tensile SHPB [7]

2.2 Design of SHPB Components

The design of SHPB components is very important for the SHPB machine to produce an accurate result. The initial version of the SHPB is for dynamic compression. Still, it has been developed into some other methods such as torsion, tension, triaxial and axial/shear combination, and is developed based on the same principles and identical mechanisms. Moreover, the differences between different types of SHPB are the loading and specimen gripping methods.

A classic Compression SHPB consists of an incident bar, transmission bar, a momentum trap device, and an extension bar (optional). Generally, the bars are developed of the same diameter, and the same material is used. This is to ensure uniform wave speed during material testing. The bar material is designed to have linear-elastic behaviour with a high yield strength since the stress waves of the bars are measured by surface strains. This indicates that the bar material should be rigid enough to have no machine or machine components deformation under applied forces [8].

Besides, the bars must remain straight and can move freely on their supports with less friction to ensure one-dimensional wave propagation in the bars. Moreover, overlapping between the incident and reflected pulses needs to be avoided by designing the length of the incident bar to be at least two times the length of the striker [8][9]. Besides, the length to diameter ratio of the bar, L/D, needs to be equal to or more than 20 to ensure one-dimensional stress wave propagation [8] [10]. The stress wave of SHPB needs to be a one-dimensional wave. This is because with a one-dimensional stress wave, the longitudinal stress and displacement are uniform over the cross-section of the bar and the radial stress is everywhere zero [8].

2.2.1 Design of Miniature SHPB

According to Clark [11], the length of the striker, incident bar, and transmission bar of SHPB depended on the material used on the bar, the desired strain, and the desired strain rate. Therefore, before the design process of a miniature compression SHPB began, the strain rate, strain, and stress level were set for further calculation. To determine the striker bar's length, the wave speed (C_b) through the bar was calculated by using Equation 1. Then density (ρ) and Young's modulus (E) were based on the bar's material.

$$C_b = \sqrt{\frac{E}{\rho}} \tag{1}$$

After that, the total time of one test (t_{total}) was calculated dependent on the strain rate ($\acute{\varepsilon}$) and strain (ε) that was set before the designing process. Then the value was used to calculate the total time taken for the wave to travel through the striker bar one time (t_{bar}).

$$t_{total} = \frac{\varepsilon}{\acute{\varepsilon}} \tag{2}$$

$$t_{bar} = \frac{t_{total}}{2} \tag{3}$$

The calculated value of the time taken for the wave to travel through the striker bar once was then used to calculate the length of the striker ($L_{striker}$) by using the equation below.

$$L_{striker} = C_b \ x \ t_{bar} \tag{4}$$

The Time-vs-Position graph, as shown in Figure 2.2, was used to figure out the incident bar and transmission bar length. In compression SHPB, the strain gages position needs to be located at the centre of the incident and transmission bars to read the appropriate wave at the proper time.

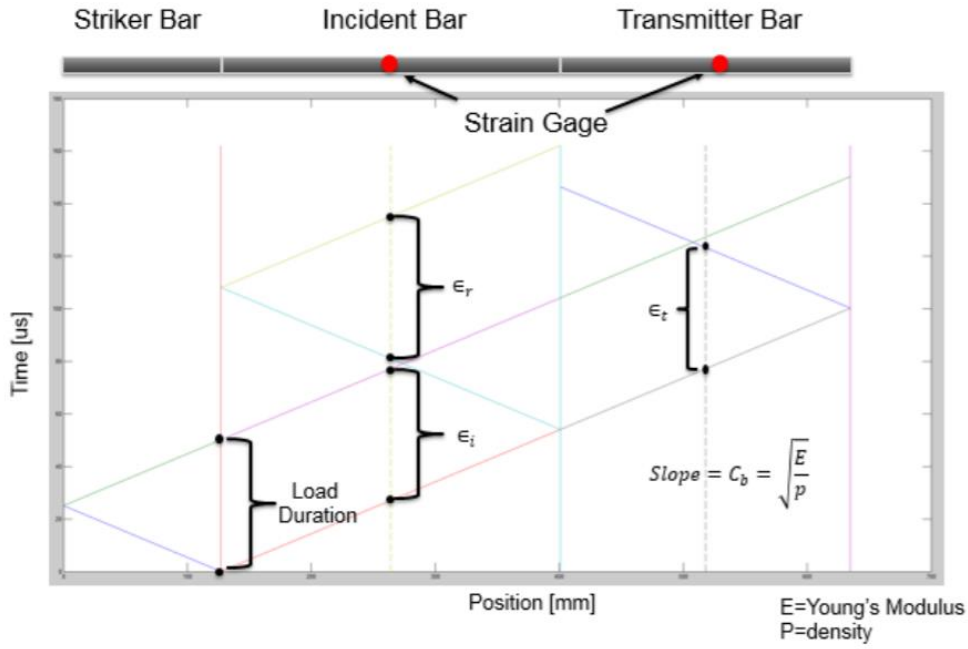


Figure 2.2 Time vs Position Graph [11]

The blue, purple, and red lines would represent the collision between the bars if the wave were travelling through them. The blue and yellow dotted lines represented the strain gages. The diagonal lines were the wave propagation through the bars. The slope of this wave propagation represented the wave velocity which was calculated based on the wave speed equation (Equation 1). The wave propagation started when a collision happened between the striker bar and incident bar and sent out a wave through the incident and transmission bars. A new wave was then formed whenever one of the waves reached another bar. The waves that were travelling then picked up by two strain gages to find the incident strain (ε_i), the reflected strain (ε_r), and the transmission strain (ε_t). Then the length of the incident bar and transmission bar can be obtained based on the requirement set by strain gauge readings [11].

2.3 Numerical Modelling on SHPB

Based on a survey of literature available open access, most of the numerical modelling involved SHPB was to characterize the specimen's mechanical behaviour [12]–[17]. In the article written by Xie et al. [15], the numerical simulation was carried on to study the dynamic characteristics of different concentrations of a mixture of **coal and rock samples** under the impact load by using LS-Dyna finite element analysis software. The specimen was placed between incident and transmission bars using the striker's impact velocities from 4.590 to 8.791 m/s. Besides, Gupta [16] studied the numerical behaviour of **AA7075 specimens** under dynamic conditions using the different types of shapes (circular and square) in striker and specimen using ABAQUS software. The simulation was run by using different velocities from 20m/s to 50m/s. In addition, material properties of the bar, such as density, elastic modulus, and Poisson’s ratio, were defined.

Another numerical simulation was done by Taşdemirci, Ergönenç, and Güden [17], to determine the dynamic deformation behaviour and stress state of the **316L stainless steel metallic hollow sphere** structure. The simulation was done by using LS-DYNA, the finite element software. The simulation model consisted of the incident bar, transmission bar, striker, and the specimen, as shown in Figure 2.3. The velocity of the striker bar, 12.5m/s, was applied as the boundary condition, and the components were modelled with eight-node solid elements.

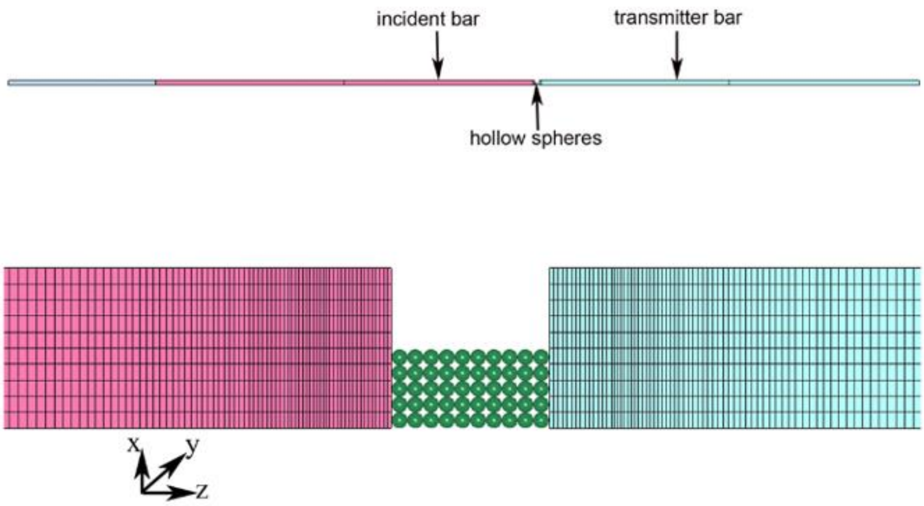


Figure 2.3 Incident bar, transmission bar, striker, and the specimen [17]

However, a few numerical modelling was conducted to characterize the SHPB component's behaviour under impact to propose a design guideline on the component to eliminate spurious waves, as discussed below.

2.3.1 Numerical Modeling on Tensile SHPB

There was numerical modelling that had been done on Tensile SHPB by Acosta [9] using ANSYS LS-Dyna. The model was generated in two ways: in “dry” condition, only incident bar, striker, and transfer flange were included, and in full conditions, all components were included. The study of the mechanics of momentum transfer between the striker, incident bar, and the transfer flange can be done by modelling in “dry” conditions. In addition, the numerical model was allowed to determine stress concentrations at the flange-incident bar interface.

In Acosta’s simulations, explicit finite element models were created. First, a hollow cylindrical striker was fired against a flange with specific velocities (1 m/s to 20 m/s). Then, the striker's momentum was transferred by the flange to the incident bar, as shown in Figure 2.4. The working principle was similar to the set-up discussed by Goh [7]. Contact analysis was done to the SHPB, and Acosta specified automatic surface to surface contacts at the surfaces between striker and flange and between the flange and the incident bar.

Figure 2.4 Schematic numerical model in "dry" condition [9]

The dimension used for the contact analysis was the actual dimensions of the tensile SHPB developed for experimental testing. Before meshing and modelling, the linear elastic material properties such as density, elastic modulus, and Poisson's ratio were defined. Moreover, eight-node solid elements were used to mesh the solid geometry, as shown in Figure 2.5. The element size used was 3mm, but with a finer mesh of 1mm around the intersection areas.

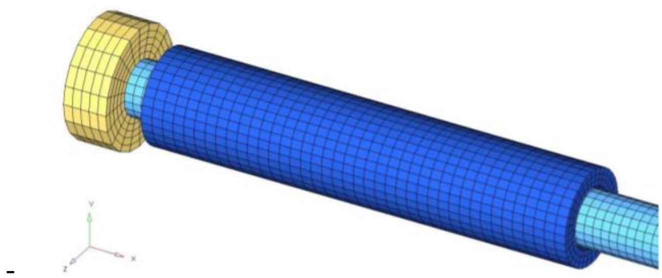


Figure 2.5 Mesh of component in 'dry' condition [9]

A momentum trap was then included in the model after the transfer flange as a part of the evaluation. The momentum trap was in contact at the end of the flange but was not attached to the flange. Figure 2.6 shows the momentum trap and its support.

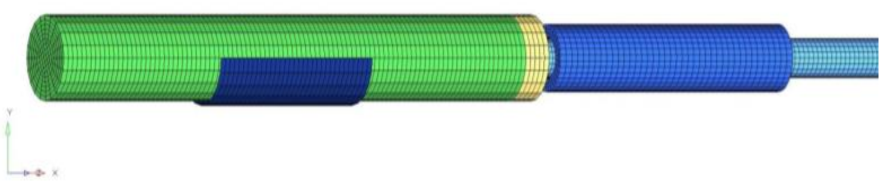


Figure 2.6 Mesh of momentum trap and support [9]

Stress concentration levels were then observed at the interface between the incident bar and the transfer flange representing a threaded, welded, or press-fitted connection as shown below.

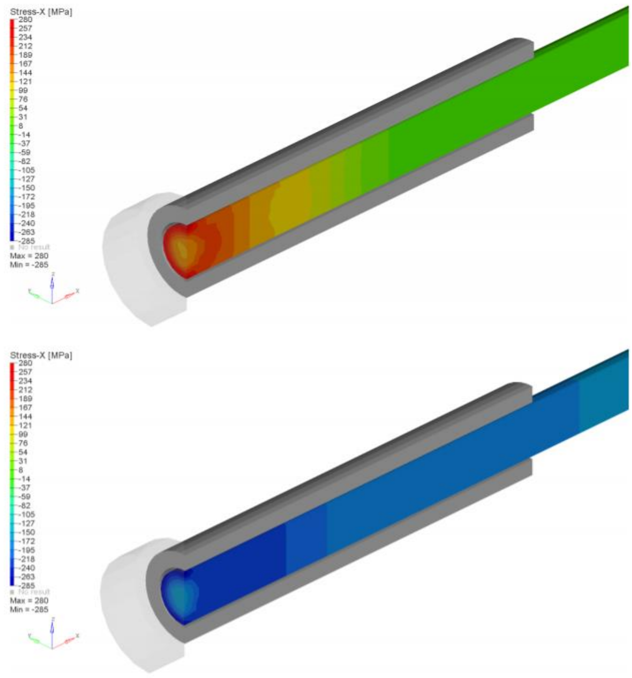


Figure 2.7 Stress levels at the incident bar and flange intersection [9]

Numerical modelling of tensile SHPB also done by Shin, Lee, Kim, and Sohn [15] to draw design guidelines for striker and flange in order to eliminate spurious waves. Spurious waves followed the main tensile pulse before the reflected pulse when the hollow striker generated a tensile pulse, and the transfer flange was recorded. Therefore, spurious waves should be prevented to get rid of their superposition with the reflected pulse.

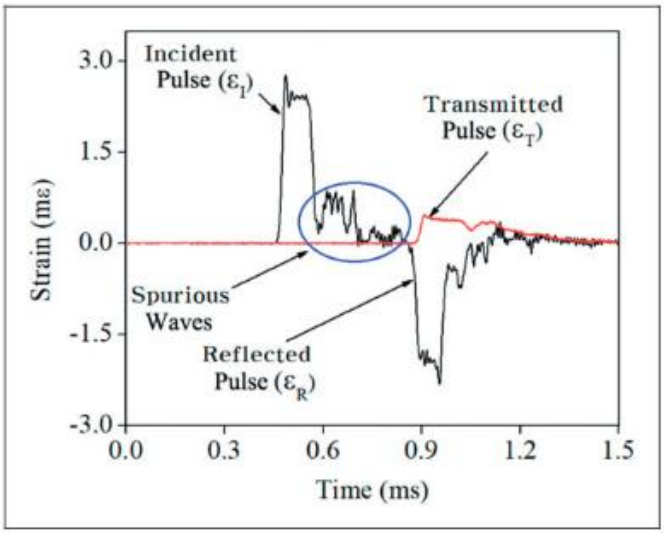


Figure 2.8 Spurious waves (circled) [18]

As shown in Figure 2.9 (b), a one-piece flange with the incident bar was chosen to undergo the simulation because a cleaner pulse can be obtained using the one-piece flange compared to other types of the flange. Several finite element models were generated with four design variables. The variables were the input bar's diameter (D_b) (10 to 25mm with a step size of 5mm), the striker's thickness (T_s) (0.1D_b to 0.4 D_b with a step size of 0.1 D_b), the height of the flange (H_f) (0.1D_b to 0.5 D_b with a step size of 0.1 D_b), and the length of the flange (L_f) (5 to 25mm with a step size of 5mm).

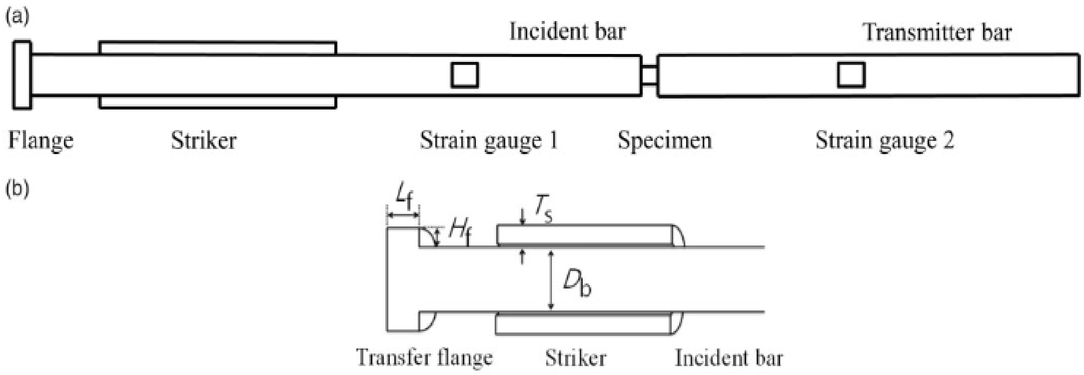


Figure 2.9 (a) Schematic diagram of tensile SHPB (b) One-piece flange [18]

The striker was set to 0.1mm far from the impact surface in the simulation with its initial velocity of 15m/s. The linear elastic material properties, which were the elastic modulus, density, and Poisson's ratio, were defined. However, the yield criterion was not set by assuming a purely elastic behaviour for the material. Moreover, tensile pulses were generated by using different values of design variables to observe the appearance of a spurious wave. Figure 2.10 shows the generated tensile pulse by using different bar diameters and the ratio of the cross-sectional areas of the striker and bar (A_{st}/A_b).

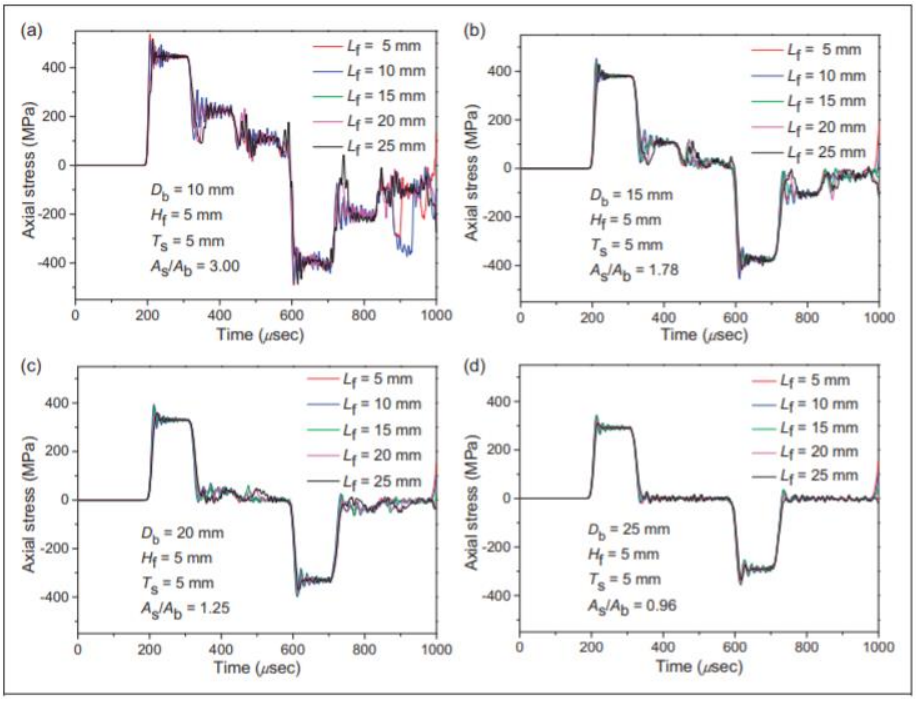


Figure 2.10 Generated pulses for flange lengths at H_f = 5mm and T_s = 5mm [18]

Based on the findings, it was suggested that the cross-sectional areas of the striker and flange needed to be the same. Moreover, the cross-sectional areas of the striker, flange, and bar should be the same. Further, the flange length identical to the bar diameter was suggested. As such, the signal fluctuation in the plateau region of the incident pulse can be reduced.

2.3.2 Boundary Condition and Load

Boundary conditions have a significant impact on the simulation's result. A boundary condition that was defined correctly can lead to a more accurate result. According to Bhatnagar [19], fixed support was set at the end of the transmission bar of his model of a compression SHPB as the boundary condition. Moreover, most of the articles [9], [10], [15]–[17] mentioned that velocity was applied as one of the boundary conditions in the simulation. In tensile SHPB, a force (from the pressurized air) was applied to the striker to slide along the incident bar and hit the anvil. Based on the thesis of Goh [7], there was some formula to be used to calculate the maximum force applied to the striker and is as shown below:

$$F_{st} = A_{st}P \tag{5}$$

$$Striker\ Area, A_{st} = \frac{\pi}{4}(d_o^2 - d_i^2) \tag{6}$$

where P is the maximum pressure that can be exerted by the pressure tank

d_o is the striker's outer diameter

d_i is the striker's inner diameter

The calculated force was considered the load applied to the striker's surface with the direction towards the anvil in the simulation.

2.4 Mechanical Properties of Materials

2.4.1 Stress and Strain

The relationship between the stress and strain in a material was determined by subjecting a material to loading configuration. In the test, a constantly increasing axial force was subjected to a material specimen. As the load increases, the deflection was measured. Material can deflect depending on its ability to extend under deflection based on its properties. The load values and deflection values can be converting to stress values strain values [20] as shown below:

$$Stress, \sigma = \frac{F}{A_o} \tag{7}$$

$$Strain, \varepsilon = \frac{L - L_o}{L_o} \tag{8}$$

Where F is the force applied to the material

A_o is the initial cross-sectional area of the material

L is the current length of the material

L_o is the initial length of the material

Stress is defined as a distributed force on an external or internal surface of a body. There were few types of force distribution which are compressive (pushing), tensile (pulling rather than pushing) and shear (rubbing or sliding) [21]. Stress is measured in newtons per meter squared (N/m^2) or pascal (Pa) [22]. Besides, strain is defined as the change in the dimension of material under an applied force [22]. There were two types of strain which are the normal and shear strains. Normal strain, ε is defined as the rate of change of the length of the stressed element in a particular direction. Shear strain is a measure of the distortion of the stressed element [21].

2.4.2 Stress-Strain Curve

A stress-strain curve can be plotted using the stress value and the strain value from the tensile test, as shown in Figure 2.11.

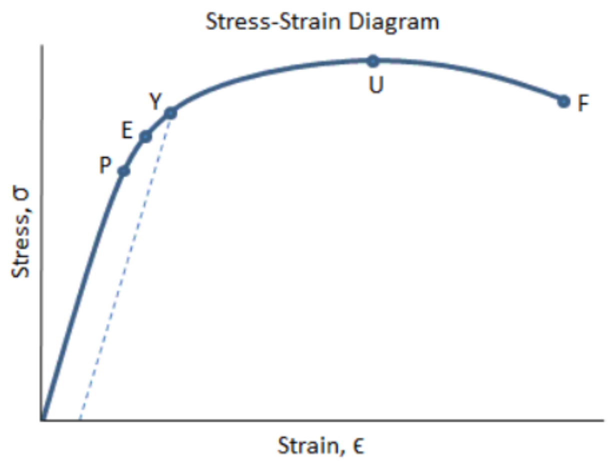


Figure 2.11 Stress-strain curve [20]

Stress-strain curves were commonly used when analyzing an engineered component. There are several points in Figure 2.11. Point P in the curve indicates the proportionality limit, which stands for the maximum stress at which the stress-strain curve is linear. Elastic limit, E, is the maximum stress that can be applied to a material without causing plastic deformation. When the material is stressed below its elastic limit, it returns to its original length once the load is removed. The linear-elastic relationship between the stress and the strain is represented by Hooke's Law. The slope of the linear line indicates Young's Elastic Modulus or modulus of elasticity, E.

Besides, Y represents the yield point. The stress at the yield point is the yield strength, σ_y. A line parallel to the linear portion of the curve is drawn that intersects the strain value of 0.002, and the point at which the line crosses the stress-strain curve is the yield point. This method is called the 0.2% offset method. Point U is the point of ultimate tensile strength, which is the maximum stress on the stress-strain curve. After reaching U, necking begins in ductile material where the cross-sectional area of the material reduces. Lastly, F is the fracture point in which the material fails and separates into two. However, not all stress-strain curve of the materials behaves the same. The stress-strain curve of different materials is as shown in Figure 2.12. [23]–[25]

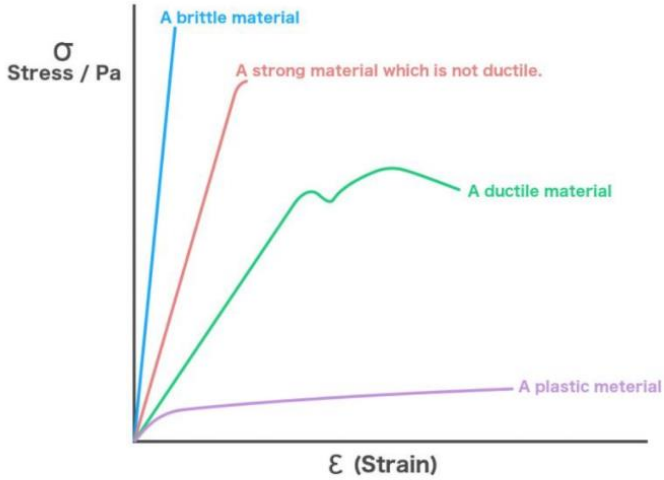


Figure 2.12 Stress-strain curve of different materials [25]

2.4.3 Yield Strength and Safety Factor

By looking into the stress-strain curve in the elastic region, if the tensile loading continues to increase, yielding will occurs at the start of plastic deformation. Yield strength, σ_y is the maximum stress that can be applied before the structure begins to change its shape permanently. Yield stress can be obtained by using the 0.2% offset method in the stress-strain curve. The yield stress indicated the start of plastic deformation, and it was necessary for engineering structural and component designs. Therefore, engineers use yield stress when designing products. Usually, the maximum load must below the yield stress limit [26]. Besides, yield stress also can be calculated with the use of a safety factor.

The safety factor was usually used to predict the required yield stress of a material or the allowable working stress that a material can withstand. The safety factor is defined as:

$$safety\ factor = \frac{\sigma_y}{\sigma_w} \tag{9}$$

Where σ_w is the allowable working stress

The safety factor value needs to be more than 1. This is due to a safety factor of 1 means that the structure or component fail precisely when it reaches the design load [26]. Moreover, some reasonable safety factor values were recommended for different types of material and loading conditions. Many specific industries have standards where specific safety factor values were recommended, and the value can be up to 12 when public safety was of concern. The selection of design safety factor was based on various considerations, the variations in material properties, the effect of size in material strength properties, class of materials, type of loading, manufacturing process, environmental effect, and specific requirement for reliability [21].

Table 2.1 Safety factor for different types of material and loading condition

Item	Material and Loading Conditions	Safety Factor
1	When exceptionally reliable known materials are used under controllable conditions and subjected to loads and stresses that can be determined with certainty, and where low weight is a particularly important consideration	$n = 1.25–1.5$
2	When well-known materials, under reasonably constant environmental conditions, subjected to loads and stresses that can be determined using qualified design procedures	$n = 1.5–2.0$
3	When average materials are used and are operated in ordinary environments and subjected to loads and stresses that can be determined	$n = 2–2.5$
4	When brittle materials under average conditions of environment, load, and stress are used. Also when less tried or uncommon materials are used	$n = 2.5–3$
5	When untested materials are used under average conditions of environment, load, and stress	$n = 3–4$
6	When better known materials are used in uncertain environments or subjected to uncertain stresses such as dynamic load	$n = 3–4$
7	When repeated or cyclic loads are used, the endurance limit (rather than the yield strength) should be used. Depending on the environment and materials' SF of 1 to 6 are acceptable	$n = 1–6$
8	When impact forces are used, depending on the severity of impact SF given in item 3 to 6 are acceptable	$n = 2–4$
9	When brittle materials are used, ultimate strength should be used as the theoretical maximum and depending on the probability of failure of the material the factors presented in items 1 to 6 should be approximately doubled	$n = 2–12$
10	When impact forces are used, the factors given in items 3 to 6 are acceptable, but an impact factor should be included	$n = 2–4$

CHAPTER 3

Methodology

3.1 Contact Analysis

Following ANSYS [27], there are three types of contact models: surface-to-surface, node-to-surface, and node-to-node. A different set of contact elements will be used on a different kind of contact model. In this project, surface-to-surface contact will be used. Some basic steps can be followed to perform a surface-to-surface contact analysis. The steps are mostly including model geometry creation and mesh, identification of the contact pairs, designate contact and target surfaces, define target and contact surface, set the element key options and real constants, define the motion of the target surface (only for rigid-to-flexible), apply boundary conditions and load, define solution options and load steps, solve the contact problem and review the results [28]. Figure 3.1 shows the flow chart of the contact analysis in this project.

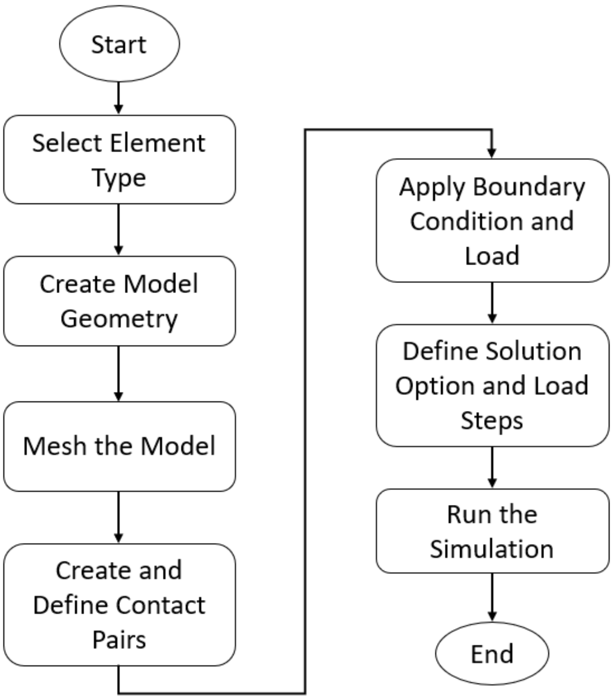


Figure 3.1 Flow Chart of Contact Analysis

3.1.1 Development of Finite Element Model

Before creating the solid model, element attributes such as element types and material properties needed to be assigned to the solid model geometry. There were different types of categories for the element type: beams, pipes, shells, and solid. Solid can be separated into two groups which are the 2-D solid and the 3-D solid. For the 3-D solid, the element name to be used was SOLID. In this project, the element type used was SOLID185. It was a 3-D 8-Node Structural Solid and having 3 degrees of freedom at each node [29]. After setting the element type used, linear-elastic material properties (modulus of elasticity, density, and the Poisson's ratio) for AISI 4140 steel and AISI 1020 steel were defined as shown in Table 3.1. As mentioned in the article written by Shin et al. [18], the material was assumed as a purely elastic behaviour material. AISI 4140 steel was applied to the incident bar, striker, and anvil, while AISI 1020 steel was applied to the momentum stopper, which acts as a momentum absorber.

Contact analysis only involved the striker, incident bar, and anvil without using the test specimen and transmission bar. The material and dimension used in the geometry created in the simulation correspond to the actual material and dimension of the tensile SHPB developed by Goh [7]. The dimensions of the striker, incident bar, and anvil are tabulated as shown in Table 3.2. A momentum stopper that acts as a momentum absorber was also created as part of the evaluation.

Table 3.1 Linear-elastic material properties for striker, incident bar, anvil and momentum stopper

Part Name	**Material**	**Modulus of Elasticity (GPa)**	**Density (kg/m^2)**	**Poisson's Ratio**
Striker	AISI 4140 Steel	205	7850	0.29
Incident Bar				
Anvil				
Momentum Stopper	AISI 1020 Steel	200	7900	0.29

Table 3.2 Dimension of striker, incident bar, and anvil

Part Name	Outer Diameter (m)	Inner Diameter (m)	Length (m)
Striker	0.03814	0.02	0.05
Incident Bar	0.02	-	2.00
Anvil	0.027	0.02	0.05

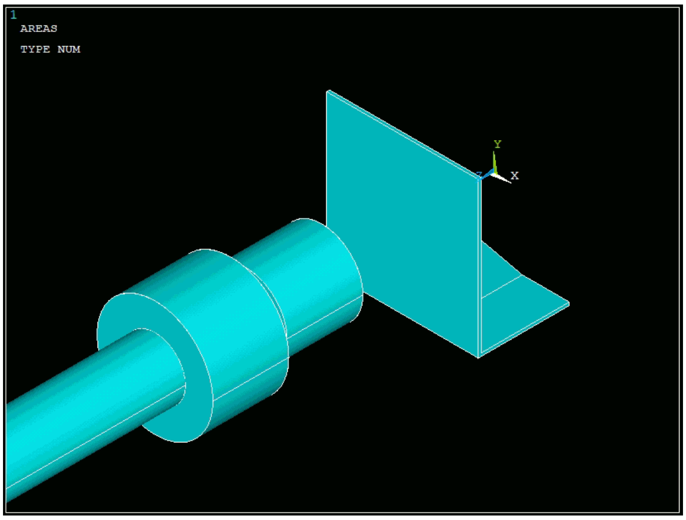


Figure 3.2 Tensile SHPB model

Next, the volume mesh was generated by the sweeping method. The average element size used was 0.01m for the incident bar, the default size remained for the striker, and anvil and element size of 0.002m was used for the momentum stopper. Hexahedral-shaped mesh was chosen. All the components had meshed to 66331 elements and 27516 nodes.

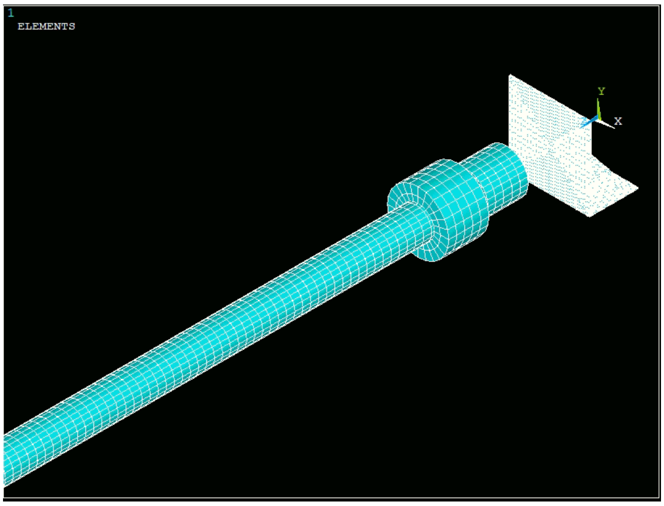
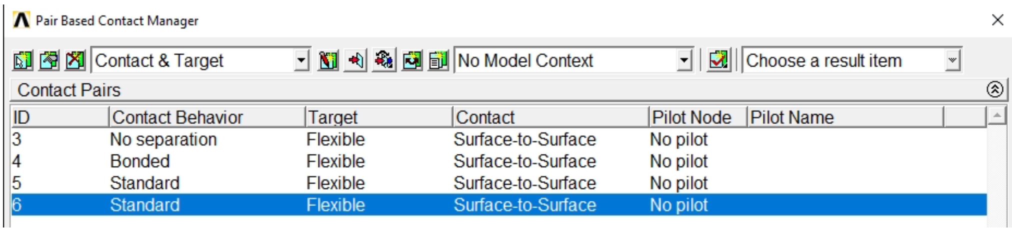


Figure 3.3 Mesh of the model

3.1.2 Define Contact Pairs

Surface-to-surface contact elements can be used to model rigid-flexible or flexible-flexible contact between surfaces. In the problems involving contact between two surfaces, one of the surfaces will be selected as the target surface while the other surface will be set as the contact surface, and this will form contact pair. For 3-D contact pairs, TARGE170 will be used with CONTA173 or CONTA174 [27]. In this project, TARGE170 was used as the target element type to represent different 3-D target surfaces. In contrast, CONTA174 was used as the contact element type to represent contact and sliding between 3-D target surfaces [29]. There were four contact pairs been created to run the simulation which was contact between (1) the inner surface of the striker and the outer surface of the incident bar, (2) the inner surface of the anvil and the outer surface of the incident bar, (3) contact surfaces of striker and anvil, and (4) contact surfaces of anvil and stopper. All the contact pairs were flexible-flexible contact.



ID	Contact Behavior	Target	Contact	Pilot Node	Pilot Name
3	No separation	Flexible	Surface-to-Surface	No pilot	
4	Bonded	Flexible	Surface-to-Surface	No pilot	
5	Standard	Flexible	Surface-to-Surface	No pilot	
6	Standard	Flexible	Surface-to-Surface	No pilot	

Figure 3.4 Created contact pairs

For the 1st contact pair, the outer surface of the incident bar was selected as the target surface, and the contact surface was the inner surface of the striker. The contact behaviour was chosen as the no separation contact. The no separation contact was where the contact surface bonded to the target surface in the normal direction. The contact surface cannot separate itself from the target surface, as shown in Figure 3.5. However, sliding is allowed between the contact and target surface [30]. With this contact behaviour, the striker was allowed to move along the incident bar.

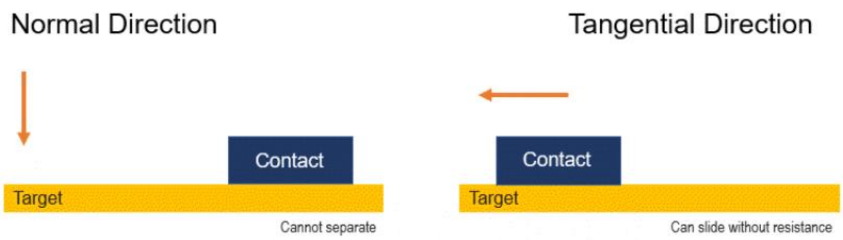


Figure 3.5 No separation contact behaviour [31]

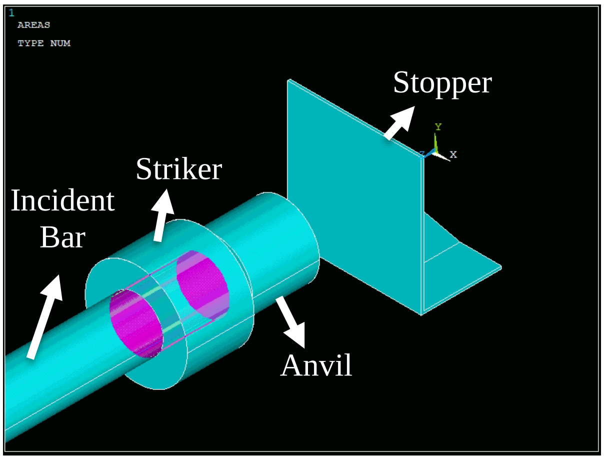


Figure 3.6 Inner surface of the striker (contact surface)

For the 2nd contact pair, the target surface was the outer surface of the incident bar, and the contact surface was the inner surface of the anvil. In actual conditions, the anvil and the incident bar were connected. The connection needs to be strong enough to withstand the impact of the striker. Thus, the contact behaviour selected for this contact pair was bonded contact. Bonded contact is where the target and the contact surfaces are tied together in all directions. The selected two surfaces cannot separate, and no sliding is allowed, shown in Figure 3.7.

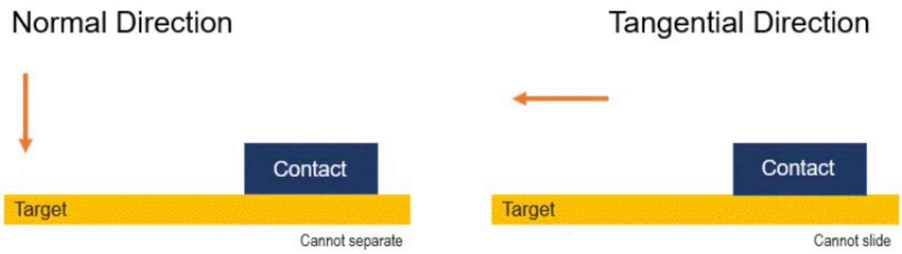


Figure 3.7 Bonded contact behaviour [31]

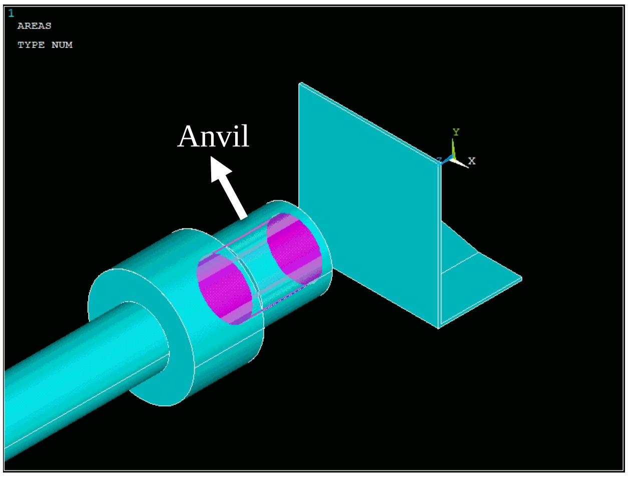


Figure 3.8 Inner surface of the anvil (contact surface)

The 3rd contact pair was a flexible-flexible contact where the target surface was the surface of the anvil that will in contact with the striker during impact, while the contact surface was the striker surface, as shown in Figure 3.9. The contact behaviour selected for this contact pair was the standard contact. With the selection of standard contact, the normal pressure will equal zero if there is a separation between the surfaces. Besides, impact constraint was used in this contact pair.

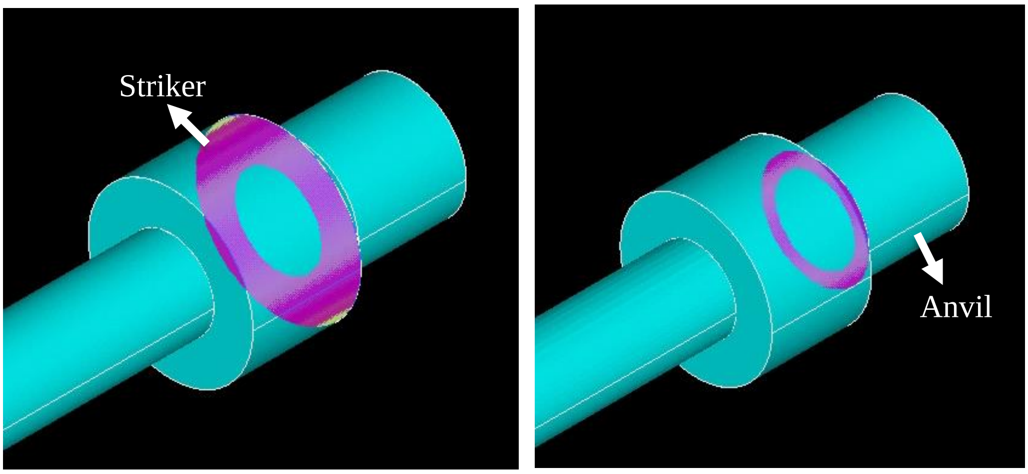


Figure 3.9 Contact surface on striker (left) and target surface on the anvil (right)

For the 4th contact pair, the target surface was the surface of the stopper that will hit by the anvil, and the contact surface was the anvil's surface that will be hitting the target surface, as shown in Figure 3.10. The contact behaviour of this contact pair was standard contact. Impact constraint was selected to adjust the time increment automatically, as mentioned in ANSYS Mechanical APDL [32]. As such, overlapping between impact surfaces can be avoided.

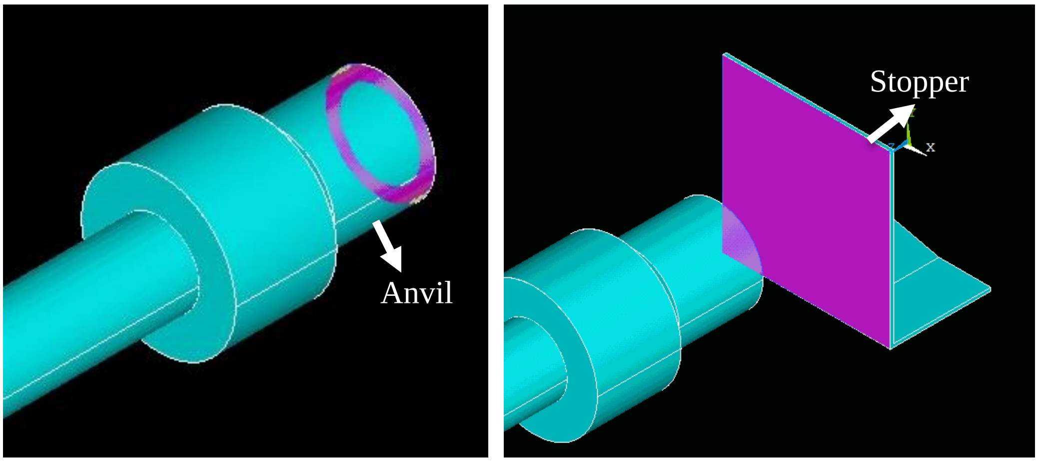


Figure 3.10 Contact surface on the anvil (left) and target surface on the solid block (right)