ISTANBUL TECHNICAL UNIVERSITY ★ GRADUATE SCHOOL

OPTIMIZATION OF ADHESIVELY BONDED JOINT UNDER OUT OF PLANE IMPACT LOADING

M.Sc. THESIS

Ezgi BAKIR

Department of Aeronautical and Astronautical Engineering

Aeronautical and Astronautical Engineering Programme

JULY 2025

ISTANBUL TECHNICAL UNIVERSITY ★ GRADUATE SCHOOL

OPTIMIZATION OF ADHESIVELY BONDED JOINT UNDER OUT OF PLANE IMPACT LOADING

M.Sc. THESIS

Ezgi BAKIR
(511211162)

Department of Aeronautical and Astronautical Engineering

Aeronautical and Astronautical Engineering Programme

Thesis Advisor : Prof. Dr. Vedat Ziya DOĞAN

JULY 2025

İSTANBUL TEKNİK ÜNİVERSİTESİ ★ LİSANSÜSTÜ EĞİTİM ENSTİTÜSÜ

DÜZLEM DIŞI DARBE YÜKLEMESİ ALTINDA YAPIŞTIRMA BAĞLANTILARININ OPTİMİZASYONU

YÜKSEK LİSANS TEZİ

Ezgi BAKIR
(511211162)

Uçak ve Uzay Mühendisliği Anabilim Dalı

Uçak ve Uzay Mühendisliği Programı

Tez Danışmanı: Prof. Dr. Vedat Ziya DOĞAN

TEMMUZ 2025

Ezgi BAKIR, a M.Sc. student of ITU Graduate School student ID 511211162, successfully defended the thesis entitled “OPTIMIZATION OF ADHESIVELY BONDED JOINT UNDER OUT OF PLANE IMPACT LOADING”, which she prepared after fulfilling the requirements specified in the associated legislations, before the jury whose signatures are below.

Thesis Advisor :	**Prof. Dr. Vedat Ziya DOĞAN** Istanbul Technical University	.....
Jury Members :	**Assist. Prof. Dr. Oğuzcan İNAL** Istanbul Technical University	.....
	Dr. Ceyhun TOLA ASELSAN – Military Electronics Industry	.....

Date of Submission : 30 May 2025
Date of Defense : 24 July 2025

To all my loved ones,

FOREWORD

This thesis was written during my Master's studies at Istanbul Technical University, Department of Aeronautics and Astronautics. I would like to thank my advisor, Prof. Dr. Vedat Ziya DOĞAN, for his help and support. I'm also thankful to Assist. Prof. Dr. Oğuzcan İNAL for answering my questions and supporting me along the way.

Thanks to my family and friends for always being there. Special thanks to my beloved Berke for his love, patience, and support. Lastly, thank you to my sweet cats, Loki and Lokum, for keeping me company during the late nights of writing.

July 2025

Ezgi BAKIR
(Aeronautical Engineer)

TABLE OF CONTENTS

Page

ABBREVIATION

ADCB	**:** Asymmetric Double Cantilever Beam
ANN	**:** Artificial Neural Network
ASME	**:** American Society of Mechanical Engineers
CDM	**:** Continuous Damage Model
CFRP	**:** Carbon Fiber Reinforcement
COPRAS	**:** Complex Proportional Assessment
DCB	**:** Double Cantilever Beam
FEA	**:** Finite Element Analysis
FEM	**:** Finite Element Method
FMTS	**:** Foam Filled Multi Cell Thin Walled Structures
GA	**:** Genetic Algorithm
GPR	**:** Gaussian Process Regression
MIF	**:** Maximum Impact Force
MOPSO	**:** Multiobjective Particle Swarm Optimization
PR	**:** Polynomial Regression
RBFN	**:** Radial Basis Function Networks
RMSE	**:** Root Mean Square Error
SEA	**:** Specific Energy Absorption
SLJ	**:** Single Lap Joint

SYMBOLS

kg	**:** Kilogram
g	**:** Gram
E_1	**:** Elasticity Modulus in Fiber Direction
E_2	**:** Elasticity Modulus in Transverse Fiber Direction
$E_{absorbed}$	**:** Absorbed Energy
$E_{kinetic,initial}$	**:** Inital Kinetic Energy
$E_{kinetic,final}$	**:** Final Kinetic Energy
F_{ft}	**:** Fiber Tension Failure Index
F_{fc}	**:** Fiber Compression Failure Index
F_{mt}	**:** Matrix Tension Failure Index
F_{mc}	**:** Matrix Compression Failure Index
J	**:** Joule
GPa	**:** Gigapascal
MPa	**:** Megapascal
mm	**:** milimeter
m	**:** meter
N	**:** Newton
t	**:** Time
s	**:** second
S_{12}	**:** In Plane Shear Strength
X_C, X_t	**:** Compressive And Tensile Strength In Fiber Direction
Y_C, Y_t	**:** Compressive And Tensile Strength In Transver Fiber Direction
ρ	**:** Density
σ_1	**:** Stress In Fiber Direction
σ_2	**:** Stress In Fiber Direction
τ_{12}	**:** shear stress in the plane

LIST OF TABLES

Page

LIST OF FIGURES

Page

OPTIMIZATION OF ADHESIVELY BONDED JOINT UNDER OUT OF PLANE IMPACT LOADING

SUMMARY

This thesis investigates the optimization of adhesively bonded metal and composite plates subjected to out of plane impact loading, with the aim of enhancing structural performance and reducing weight. Adhesively bonded joints are increasingly preferred in aerospace applications due to their advantageous mechanical performance and ease of manufacturing compared to conventional fasteners. However, they remain sensitive to impact loads, which can reduce the structural integrity of the assembly.

The study is divided into three main parts. In the first part, the thickness of an aluminum alloy plate, AL7050, is optimized under impact loading. Finite element simulations are created using LS-DYNA to generate training data. Kriging surrogate models are built to estimate two objective functions: von Mises stress and total mass. The optimization process is carried out using the multiobjective genetic algorithm, gamultiob, in MATLAB, resulting in a Pareto front of optimal thickness values. This approach enables an effective trade-off between minimizing stress and minimizing mass while maintaining a specified safety factor.

The second part of the study focuses on optimizing a woven composite plate, AS4 8552, subjected to impact loading. The primary goal is to maximize absorbed energy while minimizing mass. The stacking sequence and total number of plies are treated as design variables, selected from discrete values, the orientation angle is selected as (0^o and 45^o) and number of plies vary 8 from 16 for symmetric laminate. LS-DYNA simulations are used to impact model inputs and outputs, and Python scripts automate the generation of stacking sequence combinations. The Hashin failure index is used as a constraint to eliminate fail configurations.

Metamodels are created using the Gaussian Process Regression (GPR) method to predict absorbed energy and mass for different laminate designs. These models help estimate the results without running a full simulation each time. Then, a genetic algorithm is used to find the best designs by trying many different combinations. The goal is to find laminate configurations that can absorb a high amount of impact energy while keeping the weight as low as possible. In addition, only the designs that meet the required Hashin failure index limit is accepted.

In the third section, the optimization of the adhesively bonded joint is performed. Input and output data are obtained from LS-DYNA simulations. The design parameters include the aluminum plate thickness, the stacking sequence, and the number of plies. The objective functions are the total mass and the maximum von Mises stress on aluminum plate. The constraints are defined by the Hashin failure index for the composite plate.

This thesis presents an efficient design approach for adhesively bonded joints by combining finite element analysis with LS-DYNA, metamodels and multiobjective

optimization with genetic algorithms. The proposed method helps reduce simulation time while maintaining safety and performance, supporting the use of composite and metal bonding structures in aerospace and automotive applications.

DÜZLEM DIŞI DARBE YÜKLEMESİ ALTINDA YAPIŞTIRMA BAĞLANTILARININ OPTİMİZASYONU

ÖZET

Bu tez çalışması, düzlem dışı darbe yüklemesine maruz kalan, yapıştırıcı ile birleştirilmiş metal ve kompozit plakaların optimize edilmesini konu almaktadır. Havacılık ve uzay mühendisliğinde yaygın olarak tercih edilen yapıştırıcı bağlantılar, mekanik bağlantı elemanlarına kıyasla daha düşük bir toplam ağırlık sağlamalarının yanı sıra, yükün daha homojen bir şekilde dağıtılmasına olanak tanımaktadır. Ancak, bu tür bağlantılar darbe kaynaklı yüklemelere karşı oldukça hassastır ve mikroskobik düzeyde oluşabilecek görünmeyen hasarlar yapısal bütünlüğü ciddi şekilde tehdit edebilir. Bu nedenle, bu bağlantıların güvenli ve verimli çalışmasını sağlamak için dikkatli bir analiz ve tasarım süreci gereklidir.

Çalışma, üç ana bölümden oluşmaktadır. İlk bölümde, yüksek mukavemetli bir alüminyum alaşımı olan AL7050'nin düzlem dışı darbe yüklemesi altındaki davranışı incelenmiş ve bu bağlamda kalınlık parametresi dikkate alınarak bir optimizasyon süreci gerçekleştirilmiştir. Bu amaçla, LS-DYNA yazılımı kullanılarak çeşitli kalınlık değerlerine sahip plakaların sonlu eleman analizleri yapılmış, her bir analiz sonucunda elde edilen von Mises gerilmesi ve toplam kütle gibi çıktılar kullanılarak Kriging yöntemi ile metamodel (yaklaşım modeli) oluşturulmuştur. Oluşturulan bu modeller, yeni analiz yapmaya gerek kalmadan farklı tasarımların tepkilerini tahmin etme olanağı sağlamıştır. Optimizasyon aşamasında, MATLAB yazılımı içinde yer alan gamultiobj fonksiyonu kullanılmış ve genetik algoritma temelli çok amaçlı bir optimizasyon gerçekleştirilmiştir. Bu sürecin temel hedefi, von Mises gerilmesini ve toplam plakaya ait kütleyi en düşük seviyeye indirmektir. Gerçekleştirilen analiz ve optimizasyon sonucunda, belirli güvenlik katsayıları altında kalan ve yapısal dayanımı koruyarak malzeme verimliliğini artıran optimum kalınlık değerleri elde edilmiştir.

AL7050 plakalarına ait sonlu eleman analizleri, LS-DYNA yazılımı aracılığıyla gerçekleştirilmiştir ve bu analizlerde farklı darbe senaryoları dikkate alınmıştır. Plakanın dört kenarındaki tüm düğüm noktalarına x, y ve z eksenlerinde sıfır yer değiştirme atanarak yer değiştirme sınır şartı uygulanmıştır. Darbe uygulayıcısı (impactor) olarak düşey doğrultuda hareket eden, belirli bir hızda sabit çarpmaya sahip çelik bir bilye seçilmiş ve plakanın tam merkezine çarpması sağlanmıştır. Bu yapılandırma ile plaka kalınlığı tasarım değişkeni olarak tanımlanmış ve her kalınlık için farklı çıktı verileri elde edilmiştir. Oluşan bu veriler yardımıyla Kriging yaklaşım yöntemi kullanılarak sistemin davranışını yaklaşık olarak tahmin eden metamodel oluşturulmuştur. Böylece her tasarım değişikliği için yeniden simülasyon yapmaya gerek kalmadan, zamandan ve işlem gücünden önemli ölçüde tasarruf sağlanmıştır. Elde edilen maksimum gerilme ve kütle verilerine dayalı olarak, çok amaçlı optimizasyon gerçekleştirilmiştir. Bu tür çoklu hedef problemlerinde yaygın olarak kullanılan yöntemlerden biri, tüm hedef fonksiyonlara belirli ağırlıklar atayarak tek bir toplam amaç fonksiyonu oluşturmaktır. Bir diğer etkili yöntem ise, farklı hedefler

arasında en iyi dengeyi sağlayan çözümleri içeren Pareto cephesi yaklaşımıdır. Bu çalışmada, tasarımlar arasındaki en uygun denge noktalarını görmek amacıyla Pareto cephesi yöntemi tercih edilmiştir.

Çalışmanın ikinci bölümünde ise, dokuma yapıya sahip AS4 8552 karbon reçine kompozit plakanın düzlem dışı darbe altındaki tepkisi incelenmiş ve buna yönelik bir optimizasyon süreci gerçekleştirilmiştir. Bu bölümdeki temel hedef, darbe enerjisinin mümkün olduğunca fazla miktarda emilmesini sağlarken, aynı zamanda yapıda herhangi bir hasar oluşmasını önlemek ve toplam yapısal kütleyi minimize etmektir. Bu doğrultuda, tasarım parametreleri olarak toplam katman sayısı ve katman açısı belirlenmiş, her bir katman sadece 0° veya 45° yönelimlerinden birine sahip olacak şekilde sınırlandırılmıştır. Ayrıca, laminatın simetrik olacak şekilde serilmesi sağlanmıştır; yani laminat yapısı, orta düzleme göre simetrik olarak tanımlanmıştır. LS-DYNA ortamında oluşturulan sonlu eleman modellerinde, metal plaka analizinde olduğu gibi düşey hızla hareket eden bir çelik bilye kullanılmıştır. Kompozit plakaların kenarlarına, metal plakada olduğu gibi yer değiştirme sınır koşulu uygulanmıştır. Başlangıç modeli olarak [0]16 yönelimine sahip bir yapı tanımlanmış ve ardından Python betikleri yardımıyla toplamda 496 farklı laminat serilim kombinasyonu oluşturulmuştur. Bu kombinasyonlardan 155 tanesi analiz girdisi olarak kullanılmış ve LS-DYNA'da çözülmüştür. Elde edilen sonuç verileri, daha sonra metamodel oluşturmak amacıyla kullanılmıştır.

Simülasyon sonuçlarından elde edilen gerilme değerleri aracılığıyla Hashin hasar indeksi hesaplanmıştır. Hashin hasar kriteri, kompozit malzemelerde meydana gelen lif ve reçine hasarlarını ayrı ayrı değerlendiren ve çekme ile basma yüklemelerini dikkate alan bir yöntemdir. Bu kriter sayesinde, analiz edilen her konfigürasyonda hasar olup olmadığı belirlenmiştir. Hashin değeri 1'in üzerine çıktığında, o yapı başarısız olarak kabul edilmektedir. Pek çok farklı hasar kriteri bulunsa da, çalışmanın içeriği ve yükleme türü göz önünde bulundurularak Hashin kriteri bu analizler için uygun görülmüştür.

Özellikle havacılık sektöründe, yapısal bileşenlerin mümkün olduğunca hafif tasarlanması büyük bir öneme sahiptir. Çünkü daha hafif yapılar, yakıt verimliliğini artırmakta, taşıma kapasitesini iyileştirmekte ve genel performansı olumlu yönde etkilemektedir. Bu nedenle, yapının ağırlığını azaltmak, tasarım iyileştirme sürecinde ilk göz önüne alınan ve öncelikli olarak ele alınan hususlardan biridir. Bu çalışmada kullanılan kompozit yapılar farklı sayıda katmana sahip olacak şekilde modellenmiştir. Katman sayısının artması, doğal olarak yapının toplam kütlesini de artırmaktadır. Plakaların kütle bilgisi, her bir analiz sonrası elde edilen çıktı dosyalarından okunmakta ve bu şekilde doğrudan değerlendirme yapılabilmektedir. Bu yöntem, özellikle karmaşık katman yapısına sahip modellerde, ağırlık değerinin doğrudan ve pratik biçimde elde edilmesine olanak tanımaktadır.

Kompozit malzemelerin darbe yükleri altındaki davranışı incelenirken dikkat edilmesi gereken bir diğer önemli konu ise, çarpan cismin taşıdığı enerjinin ne kadarının plaka tarafından emilebildiğidir. Kompozit plakanın, herhangi bir kırılma, hasar ya da yapısal başarısızlık yaşamadan, üzerine gelen darbeyi sönümleyerek daha yüksek bir enerji miktarını emmesi, tercih edilen ve hedeflenen bir davranış biçimidir. Bu çalışmada, kompozit plakanın emdiği enerji miktarı, analiz çıktı dosyalarından elde edilen başlangıç enerjisi ve son enerji değerleri kullanılarak hesaplanmıştır. Bu hesaplama sayesinde, her analiz için yapının darbe enerjisini ne ölçüde sönümleyebildiği sayısal olarak belirlenmiştir.

Gerçekleştirilen analizlerin ardından elde edilen çıktı dosyalarından, üç temel büyüklük elde edilmiştir: Hashin hasar kriteri değeri, plaka tarafından emilen enerji miktarı ve toplam ağırlık değeri. Bu değerler, her bir analizde kullanılan farklı giriş (input) parametreleriyle eşleştirilerek, bu ilişkiyi modelleyen yaklaşım (metamodel) modelleri oluşturulmuştur. Hashin indeksi, ileride optimizasyon sırasında bir kısıtlayıcı (constraint) olarak kullanılacak olup, sadece hasarsız çözümlere izin verecektir. Diğer iki çıktı olan emilen enerji ve ağırlık ise, optimizasyon sürecinde birbirine zıt yönlü amaç fonksiyonları olarak tanımlanmış ve böylece tasarımlar arasında en iyi dengeyi bulmak için çok amaçlı bir optimizasyon gerçekleştirilmiştir.

Bu çalışmanın son aşamasında, önceki bölümlerde ayrı ayrı optimize edilen metal ve kompozit plakaların, bir yapıştırıcı aracılığıyla birleştirilerek oluşturduğu bütünsel bir bağlantı yapısının optimizasyonu gerçekleştirilmiştir. Bu aşamada, metal ve kompozit plakalar belirli bir bindirme alanı çerçevesinde yapıştırıcı ile birbirine bağlanmış olarak modellenmiştir. Böylece, iki farklı malzeme davranışına sahip olan yapı bileşenleri, birlikte çalışan bir hibrit sistem olarak ele alınmıştır. LS-DYNA ortamında yapılan bu modelleme sürecinde, daha önceki optimizasyonlarda da olduğu gibi, metal plakanın kalınlığı, kompozit plakanın toplam katman sayısı ve her bir katmanın açıları tasarım değişkenleri olarak tanımlanmıştır. Bu değişkenlerin her biri belirli aralıklarla tanımlanmış ve kombinasyonları oluşturularak geniş bir tasarım uzayı elde edilmiştir.

Ancak bu tasarım uzayında yer alan tüm kombinasyonların analiz edilmesi, yüksek sayıda sonlu eleman analizi gerektirdiği için oldukça zaman alıcı ve hesaplama maliyeti açısından verimsiz olabilmektedir. Bu nedenle, tüm olası tasarım kombinasyonları arasından rastgele bir örnekleme yöntemiyle sınırlı sayıda analiz için gerekli giriş verileri seçilmiştir. Seçilen bu örneklerin her biri için LS-DYNA kullanılarak analizler gerçekleştirilmiş ve bu analiz sonuçları, yaklaşım modellerinin eğitilmesi amacıyla kullanılmıştır. Böylece, farklı tasarımların sonuçları doğrudan simülasyon yapılmadan tahmin edilebilir hale getirilmiştir.

Yapıştırıcı ile bağlanmış bu iki plakanın oluşturduğu yapı sistemi için optimizasyon sürecinde iki ana amaç fonksiyonu belirlenmiştir: plaka tarafından emilen toplam darbe enerjisinin maksimum seviyeye çıkarılması ve tüm yapının toplam ağırlığının mümkün olan en düşük seviyeye indirilmesi. Bu hedefler, yapının darbe dayanımını artırırken, hafifliğini koruyarak performans ve verimlilik açısından dengeli bir tasarım sağlamayı amaçlamaktadır. Bu optimizasyon sürecinde kullanılan kısıtlayıcılar ise, kompozit plaka için Hashin hasar kriteri, metal plaka için ise von Mises gerilme değeri olarak belirlenmiştir. Yani, sadece bu sınır değerleri aşmayan ve güvenli olduğu kabul edilen çözümler geçerli sayılmıştır.

Bu kapsamda, analiz çıktılarından elde edilen dört temel büyüklük, emilen enerji miktarı, yapının toplam kütlesi, Hashin kriteri değeri ve metal plakanın maksimum von Mises gerilmesi için ayrı ayrı yaklaşım modelleri oluşturulmuştur. Bu modeller, genetik algoritma tabanlı çok amaçlı optimizasyon sürecinde hem hedef hem de kısıt fonksiyonu olarak görev yapmıştır. Sonuç olarak, bu aşamada oluşturulan sistematik yaklaşım ile hibrit bir bağlantı yapısının tasarımı, hem performans hem de ağırlık açısından optimize edilerek en uygun yapı konfigürasyonları elde edilmiştir.

Bu tez çalışmasında, yapıştırıcı ile birleştirilmiş metal ve kompozit plakaların düzlem dışı darbe yükleri altındaki davranışları incelenmiş ve bu yapıların çok amaçlı optimizasyon gerçekleştirilmiştir. Çalışma üç ana bölümden oluşmaktadır. İlk bölümde AL7050 alüminyum plakasının kalınlık optimizasyonu yapılmış, ikinci bölümde ise AS4 8552 dokuma kompozit plakanın katman sayısı ve açılarına göre

serilim optimizasyonu gerçekleştirilmiştir. Her iki bölümde de LS-DYNA kullanılarak darbe analizleri yapılmış, analiz çıktılarından elde edilen gerilme, kütle, emilen enerji ve Hashin kriteri değerleriyle Kriging metamodeli oluşturulmuştur. Optimizasyon süreci MATLAB ortamında genetik algoritma yardımıyla yürütülmüş, amaç fonksiyonları olarak yapısal ağırlık ve emilen enerji kullanılmıştır. Son bölümde ise bu iki plaka yapıştırıcıyla birleştirilmiş hibrit bir yapı olarak modellenmiş ve sistem düzeyinde optimizasyon gerçekleştirilmiştir. Bu kapsamda, hem kompozit hem de metal plaka için kısıtlayıcılar tanımlanmış ve yaklaşım modeli temelli çok amaçlı optimizasyonla en uygun yapı konfigürasyonları elde edilmiştir.

1. INTRODUCTION

Metals such as aluminum alloys and structural steel have been the primary materials used in aerospace structures since the beginning of the aerospace industry. Nevertheless, composite materials, which offer high strength to weight ratios, have gained increasing importance and wider application over time. As composite materials continue to replace metals in various structural applications, hybrid joints combining metals and composites have gained attention due to their potential to combine the best properties of both material types. As a result, the performance and analysis of metal-composite contact behavior have become critical topics in aerospace engineering.

There are a lot of contact types such as fastener contact, tight contact and adhesively bonded contact etc. Each contact type is used in different applications and has its own advantages and disadvantages. For this research, the focus is on adhesively bonded contact types. Adhesively bonded joints have become increasingly significant in modern aerospace industry, where lightweight and high-strength materials are crucial. Compared to fastener contact type, adhesive bonding offers several advantages, including weight reduction, improved load distribution, and enhanced fatigue performance. Adhesively bonded contact allows for the creation of connections between metal-metal, composite-composite, and composite-metal surfaces. It indicates that the area of application is expected to expand over time. Hart, Smith is demonstrated that adhesively bonded joint applications in the SAAB 340 airplane in Figure 1.1. Also, a close look to frame longeron connection is given in Figure 1.2 [1].

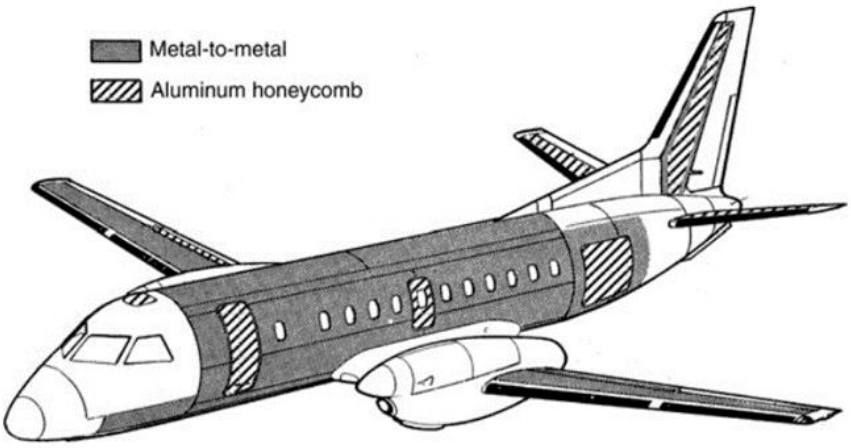

Figure 1.1 : Adhesively joint region is demonstrated for SAAB 340 [1].

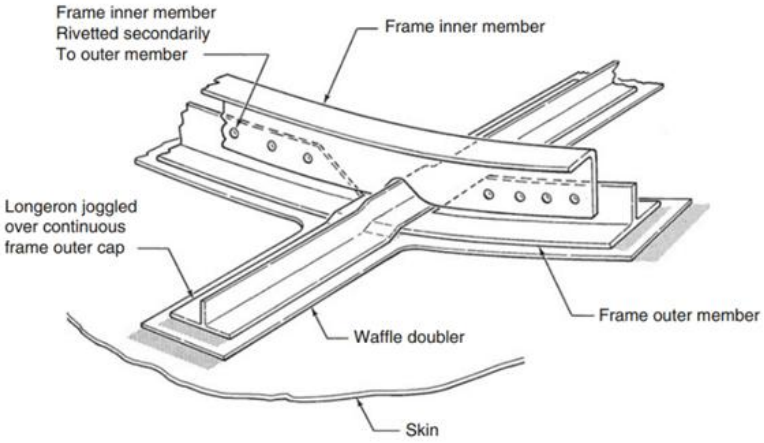

Figure 1.2 : A close look to adhesively bonded frame longeron [1].

However, these adhesively bonded structures are often subjected to impact loads in service, such as tool drops, bird strikes, or runway debris in aircraft applications. These events can lead to severe internal damage, often invisible on the surface, affecting the structural integrity and safety of the component. Therefore, understanding and optimizing the performance of adhesively bonded metal-composite plates under impact loading has become a critical area of research.

In recent years, the increasing complexity of structural design and loading scenarios has led researchers to adopt advanced optimization methods to improve the performance of bonded composite-metal structures under dynamic conditions. Traditional design approaches may fall short when dealing with multiple conflicting objectives, such as reducing mass while maintaining structural integrity under impact. Therefore, multiobjective optimization techniques have gained significant attention in aerospace applications. However, direct optimization using finite element simulations can be computationally expensive, especially when a large number of design variables

or simulations are involved. To overcome this limitation, metamodel based optimization methods have been developed as efficient alternatives.

A metamodel, also known as a surrogate model, is an approximate mathematical model that captures the relationship between inputs and outputs based on a limited set of high-fidelity simulation data. In this study, Gaussian Process Regression (GPR), also referred to as the GPR method, is employed to construct surrogate models for predicting critical performance metrics such as von Mises stress, absorbed energy, total mass, and the Hashin failure index. Among various metamodeling techniques, GPR is selected due to its flexibility, ability to provide uncertainty estimation, and its proven accuracy with low RMSE (root mean square error), values in preliminary testing. These surrogate models are trained using data generated from LS-DYNA simulations, significantly reducing computational cost while maintaining sufficient predictive performance.

The optimization process is conducted using the "gamultiobj" genetic algorithm (GA) function in MATLAB, which is well suited for solving multi objective problems with nonlinear and discrete design spaces. Genetic algorithms simulate the process of natural selection through operators like crossover, mutation, and selection. The algorithm produces a set of Pareto optimal solutions, which represent the best trade-offs between competing objectives such as minimizing mass and maximizing impact resistance. This approach is applied to both aluminum and composite plates, where design variables such as thickness, number of plies, and stacking sequence are optimized. The result is a practical and efficient design methodology capable of guiding engineers in developing lightweight and damage tolerant bonded structures for aerospace applications.

1.1 Purpose of the Thesis

The purpose of this thesis is to improve the design of adhesively bonded metal and composite plates under out of plane impact loading. The goal is to reduce both the weight of the structure and the permanent damage caused by the impact. LS-DYNA is used to create FEM models then, based on these models, surrogate models are built using the Gaussian Regression Process (GRP) method in MATLAB to estimate absorbed energy, structural mass, Hashin failure index and stress. Then, the gamultiobj genetic algorithm in MATLAB is used to find the best stacking sequences. This

approach helps to suggest lightweight and damage-resistant designs that can be useful in aerospace and automotive applications.

1.2 Scope of the Thesis

This thesis focuses on the numerical optimization of adhesively bonded metal and composite plates subjected to out-of-plane impact loading. Predefined ply angle values are considered in the composite layups. Finite element simulations are performed using LS-DYNA, and surrogate models are developed using the Gaussian Regression Process (GRP) method in MATLAB. The optimization is carried out using the gamultiobj genetic algorithm. The study is limited to out-of-plane impacts and does not include thermal effects. Material properties and adhesive behavior are assumed to be known and remain constant throughout the analysis.

1.3 Outline of the Thesis

This thesis is organized into five main chapters. the Introduction, which presents the motivation, background, and objectives of the study, Chapter 2 provides a detailed Literature Review on adhesively bonded joints, their behavior under impact loading, and previous optimization approaches in composite and hybrid structures. Chapter 3 and 4 present metamodel based multiobjective optimization for metal and composite plate, respectively. In Chapter 5, adhesively bonded structure optimization is realized. Finally, Chapter 6 presents the conclusions of this thesis.

2. LITERATURE REVIEW

2.1 Adhesively Bonded Joints in Aerospace Structures

Adhesive bonding is a common alternative to mechanical fastening in aerospace industry. This method uses placing an adhesive between two adherent surfaces, which hardens to a strong bond form. Banea and da Silva stated that adhesively bonded joints offer better performance compared to mechanical fasteners such as rivets and bolts in composite materials. Because, adhesive bonding does not require drilling holes, which can damage fibers in composite materials and create stress concentrations. Instead, it provides a smoother load transfer and helps maintain structural integrity. It also offers advantages such as reduced weight, easier manufacturing, and more flexible design options [2]. Moreover, Silva, Öchsner, & Adams, claimed that unlike riveted or bolted joints, adhesive bonding provides a more uniform stress distribution along the bonded area, leading to increased joint stiffness and improved load transfer. This advantage is visually illustrated in Figure 2.1. which compares the stress distribution in bonded and riveted joints [3].

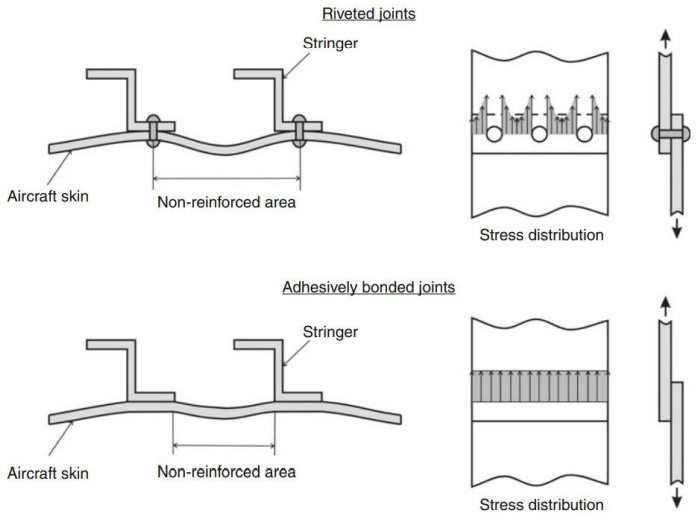

Figure 2.1 : Stress distribution comparison of riveted and adhesively bonded joints [3].

There are a lot of adhesively contact types which are shown in Figure 2.2, Abbot stated that single shear and single lap joints are widely used in aircraft structures, particularly

in connections such as wing ribs to wing skins and fuselage frames to fuselage skins. The main advantage of these joint configurations is their ease of manufacturing [4].

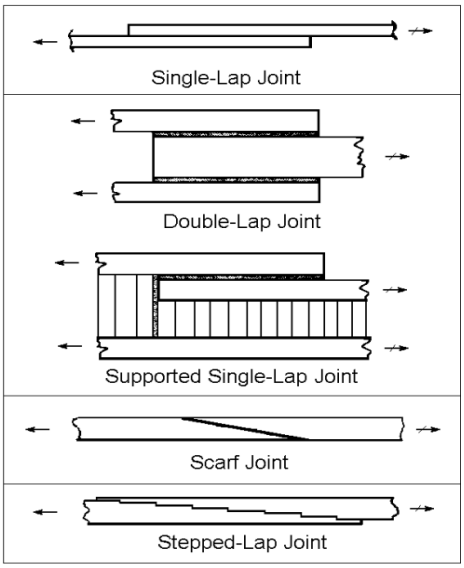

Figure 2.2 : Joint strength according to configuration and adherent thickness [4].

According to Banea and Silva, the strength of a joint depends on the stress distribution within it. The type of joint and the mechanical properties of the adhesive and adherends affect this stress distribution. Composite materials have low through-thickness strength; therefore, high through thickness stresses at the overlap ends can be critical. To mitigate this risk, peel and cleavage stresses should be minimized [2].

Adhesives are generally divided into brittle and ductile types, and they show very different mechanical behaviors. Hart-Smith showed that ductile adhesives can carry much higher loads than brittle ones, as seen in Figure 2.3 [1].

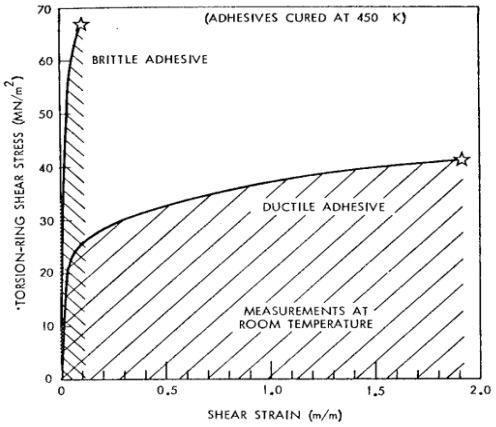

Figure 2.3 : Graphic of shear strain vs. torsion ring shear stress for ductile and brittle adhesives [1].

Ozel et al. investigated the mechanical behavior of various adhesively bonded single-lap joint (SLJ) configurations under tensile loading through both experimental and numerical methods. The joints were composed of Al 2024-T3 aluminum and carbon/epoxy composite adherends, bonded using a two-part structural adhesive, DP 460. The study examined several parameters, including composite stacking sequence, adherend thickness, and overlap length. Results showed that fiber orientation significantly influenced joint strength, especially, a $[0/45/45/90]_4$ layup improved load carrying capacity compared to more uniform layups like $[0]_{16}$ or $[45/45]_8$. Additionally, variations in the upper adherend thickness affected the failure load, especially in hybrid joints combining aluminum and composite materials. They found that increasing the overlap length also led to higher joint strength, with a more pronounced effect in composite joints. Finite element analysis revealed that peel and shear stresses were concentrated at the overlap ends, which aligned well with the experimentally observed failure locations. Overall, this study highlights the importance of joint geometry and composite layup configuration in the structural performance of bonded joints [5].

Performance of an adhesively joint depends on a lot of parameters. The behavior of the adhesive under load plays a key role. Marques and da Silva investigated the performance of adhesively bonded repair patches used in damaged aircraft structures, focusing particularly on aluminum components repaired with bonded rather than riveted patches. Traditional repair techniques such as riveting, while effective, can introduce additional stress concentrations and lead to further cracking over time. As an alternative, bonded patches offer a more uniform stress distribution and better long-term performance. However, these joints are prone to stress concentrations at the patch edges, which can initiate new cracks. To address this, the authors explored patch geometries including taper angles, spew fillets, and the use of dual adhesives, a combination of brittle and ductile materials. Types of taper angle examples are given in Figure 2.4. Through both experimental testing and finite element analysis, it was shown that a 45° taper significantly reduces edge stresses, especially when using stiff adhesives. Furthermore, taperless joints benefitted from combining brittle (AV138) and ductile adhesives, providing a balance between strength and flexibility. The study demonstrates that careful design of the adhesive interface and geometry can enhance both the strength and durability of bonded repairs, making them a promising solution

for aircraft damage restoration under dynamic loading conditions. [6] Fem modelling for simulate different taper types adhesive is complicated but it offers an important perspective.

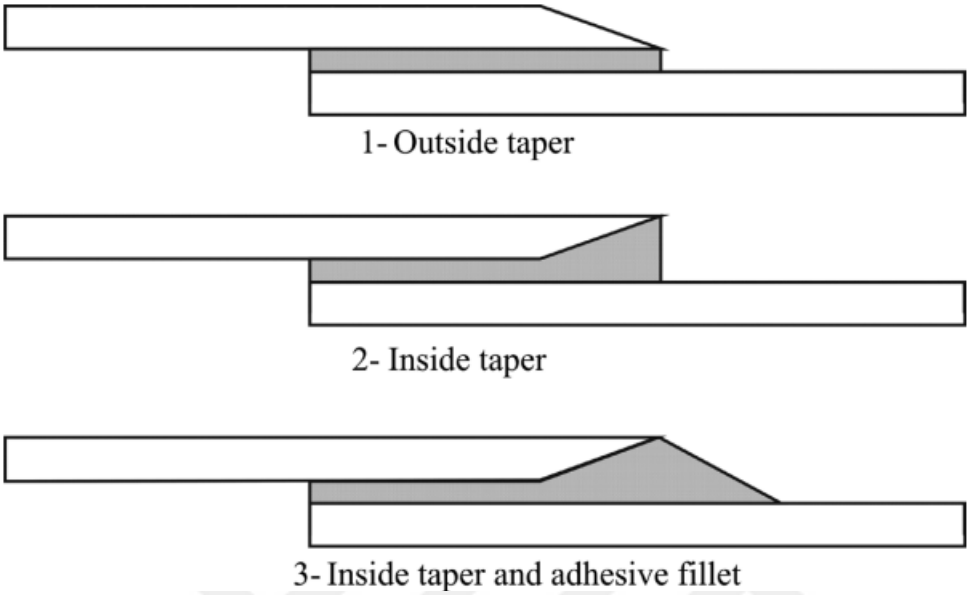

Figure 2.4 : Joint strength according to configuration and adherent thickness [6].

2.2 Impact Loading on Bonded Structures

Adhesively bonded structures, especially in aerospace applications, are frequently exposed to various impact scenarios during service life, such as bird strikes, hails, tool drops, or runway debris collisions. In real life conditions, bonded joints can be exposed to dynamic loads such as low-velocity impacts, dropped tools, or foreign object strikes. These events can cause high strain rates and localized stress, which may seriously affect the structural integrity of the joint. This section gives a general overview of typical impact situations, explains how bonded joints respond mechanically to impact, and describes the common failure types seen in bonded metal and composite structures.

Ozel et al. unlike static loads, impact loading creates more complex and rapidly changing stress patterns in both the adhesive and the materials being joined, adherends. This can lead to different types of damage, such as delamination, adhesive separation, matrix cracks, or fiber breakage in composite materials. To ensure the safe and reliable performance of bonded structures, it is important to understand how both the adhesive and the adherends behave under these high-rate loading conditions [5].

Machado et al. provided a broad overview of how adhesive joints behave under impact loading, which is especially relevant for industries like automotive where lightweight and crash-resistant structures are essential. Their study shows that adhesives are quite sensitive to strain rate, tensile and shear strengths tend to increase with higher strain rates, while fracture toughness often decreases, making the adhesive more brittle. They also emphasized that the overall impact performance of a joint depends not only on the adhesive, but also on the materials being bonded. Environmental factors such as temperature, humidity, and aging also affect joint performance, but current research in this area is still limited. In addition, they reviewed modeling approaches and pointed out that improvements in adhesive formulations and simulation methods are making it easier to design joints that are both strong and reliable under dynamic loads [7].

He and Pan investigated the failure mechanisms of single-lap, adhesively bonded composite–titanium joints under solid projectile impact, combining experimental and advanced numerical methods. In their study, three-dimensional finite element analysis was performed using Abaqus/Explicit, where the cohesive zone model was used to simulate interfacial debonding and the Hashin damage criterion was employed to predict in-plane failure of the composite adherends. The results from the simulations were in good agreement with experimental data obtained from high-speed impact tests using a gas gun. The study revealed a distinct failure progression with increasing impact velocity, initial local debonding at the adhesive titanium interface, followed by matrix cracking in the composite adherend, and finally global debonding at the same interface. These findings underline the complex stress interactions caused by anti-clastic bending and demonstrate the importance of using detailed 3D damage models for accurately capturing the behavior of dissimilar material joints under high-speed impact conditions [8].

2.3 Finite Element Modelling of Impact on Bonded Joints

LS-DYNA is a widely used software for simulating impact analyses. It is especially preferred for problems involving high strain rates and complex contact interactions. The program can handle both explicit and implicit solutions depending on the type of analysis. Because of these capabilities, it is commonly used in automotive, aerospace, and defense industries.

Deka, Bartus, and Vaidya investigated the progressive damage of composites under ballistic loading using a continuous damage model (CDM) by developing an LS-DYNA model with the MAT 161 material card. Their study showed that energy absorption in composite plates leads to damage mechanisms such as fiber debonding, fiber pull-out, elastic deformation of secondary yarns, and matrix cracking. Additionally, it was found that the boundary conditions had a negligible effect on the ballistic impact response, as the damage was highly localized [9].

2.4 Optimization of Composite Materials

Composite materials have introduced significant advancements in the field of lightweight structural design, particularly within the aerospace industry. Thanks to their high strength-to-weight ratio and strong durability, they are used in many parts of aircraft, such as fuselage and wing skins. The aerospace sector commonly uses different types of composites, including woven plies, unidirectional plies, and short fiber composites.

In unidirectional composites, the stiffness along the fiber direction, E_1, is much higher than in the transverse direction, E_2, which is perpendicular to the fibers. Woven composites, by contrast, tend to have more balanced properties in both directions, E_1 and E_2, because of their symmetrical fiber arrangement. Still, in both types, the stiffness through the thickness of the lamina is different from the in-plane properties. Since the material behavior changes with direction, these composites are called anisotropic.

When optimizing composite laminate structures, key design variables include the stacking sequence, number of layers, and the material of composite used. These factors directly influence the laminate's stiffness, strength, impact resistance, and overall performance. The orientation of each ply controls how the laminate behaves under different loads, while the number of layers affects both thickness and total mass. Choosing between unidirectional, woven, or short fiber composites also impacts mechanical behavior.

By optimizing these variables, engineers can design composite structures to meet specific goals such as maximizing stiffness, reducing weight, or improving resistance to damage. Among these, the number of plies is a critical parameter since it influences

both strength and weight. Because these goals often conflict with each other, the problem becomes a multiobjective optimization task.

Another important parameter is the stacking sequence. In theory, ply orientations could vary continuously between 0° and 180°, but in practice, they are usually limited to discrete angles such as 0°, 30°, 45° and 90°. This is due to manufacturing limits and material certification processes. These restrictions help ensure consistency in production, structural safety, and compliance with aerospace industry standards especially in applications where safety is critical.

Talagani proposed an analytical approach to estimate the delamination onset in composite laminates under low-velocity impact and used this model as the basis for optimization. A genetic algorithm (GA) was applied to identify stacking sequences with improved damage resistance by minimizing the analytically predicted threshold load for delamination. Unlike time-consuming finite element-based methods, this approach allowed rapid evaluation of numerous layup configurations. The optimized laminates achieved significantly better performance compared to conventional quasi-isotropic designs. The study highlights the effectiveness of combining analytical damage models with evolutionary algorithms to support efficient and practical composite design in the early development stages [10].

2.5 Metamodel Based Multiobjective Optimization Techniques

A multiobjective optimization study is presented by Amouzgar, Rashid, Stromberg in ASME on a heavy truck disc brake system using evolutionary algorithms and radial basis function networks (RBFN). Three objectives were considered: minimizing disc temperature, maximizing brake energy, and minimizing the back plate mass. Latin hypercube sampling and thermo-mechanical finite element analysis were used to generate and evaluate the design space. The study showed that a mass reduction of 1.4 kg could be achieved with acceptable trade-offs in temperature and energy, and highlighted the effectiveness of using low-stiffness brake pad materials. This work demonstrates the value of surrogate based methods in solving complex engineering problems [11].

Marklund and Nilsson studied how to reduce the weight of the B-pillar in a car, which is the part between the front and rear doors. They focused on how the B-pillar behaves

during a side crash. To do this, they used LS-DYNA for crash simulations and response surface methods to find the best shape and thickness. Their results showed that it is possible to make the B-pillar up to 25% lighter without making it less safe. The study showed that both the thickness and shape of the B-pillar play an important role in protecting passengers during side impacts. This research shows how optimization methods can help make car parts both lighter and safe [12].

Yin et al. conducted a comprehensive study on the crashworthiness of foam-filled multi-cell thin-walled structures (FMTSs) under lateral impact loading, which are commonly used in vehicle body components due to their high energy absorption and low weight. Using nonlinear finite element analysis in LS-DYNA, they evaluated nine different FMTS configurations and ranked them using the COPRAS (complex proportional assessment) method. Among these, the FMTSs with 2, 3, and 9 internal cells showed the best crash performance. To further enhance their energy absorption capabilities, the top three designs were subjected to metamodel-based multi-objective optimization, combining polynomial regression (PR) and the multi-objective particle swarm optimization (MOPSO) algorithm. The objectives were to maximize specific energy absorption (SEA) and minimize the maximum impact force (MIF). The results indicated that the 9-cell structure (FMTS9) outperformed the others in most scenarios, making it a promising candidate for use in future lightweight vehicle designs requiring high crash resistance [13].

Saraswat and Sharma explored the use of genetic algorithms (GAs) for solving complex optimization problems using MATLAB. The paper emphasizes the efficiency of GAs in handling nonlinear, multi-variable problems especially, when the objective function is difficult to solve analytically. MATLAB was used as the primary development environment due to its built-in functions, visualization capabilities, and ease of handling mathematical operations. The authors focused on optimizing Rastrigin's function, a standard benchmark in optimization studies, and implemented GA using various parameters such as crossover rate, mutation, and elitism count. Results showed that MATLAB, combined with proper GA parameter tuning, can produce effective solutions quickly. Additionally, they demonstrated how different GA settings affect convergence and performance, concluding that MATLAB is a suitable and powerful platform for GA based optimization, especially in engineering applications [14].

Dong et al. addressed the challenge of accurately modeling CFRP composites in LS-DYNA by optimizing material and failure parameters using LS-OPT. One of the main difficulties in simulating CFRP behavior is the mismatch between coupon-level test results and the performance of full-scale structural components due to the complex interaction between fiber and matrix. In this study, the authors employed MAT_58 along with MAT_ADD_EROSION to represent failure mechanisms in shell elements. The optimization workflow shown in Figure 2.5 was designed to minimize the difference between simulated and experimental force–displacement curves. After several iterations, the results showed that the optimization significantly reduced error and improved convergence can be seen from Figure 2.6. This demonstrates the effectiveness of LS-OPT in calibrating material parameters for composite models, especially when transitioning from small-scale tests to full-structure simulations [15].

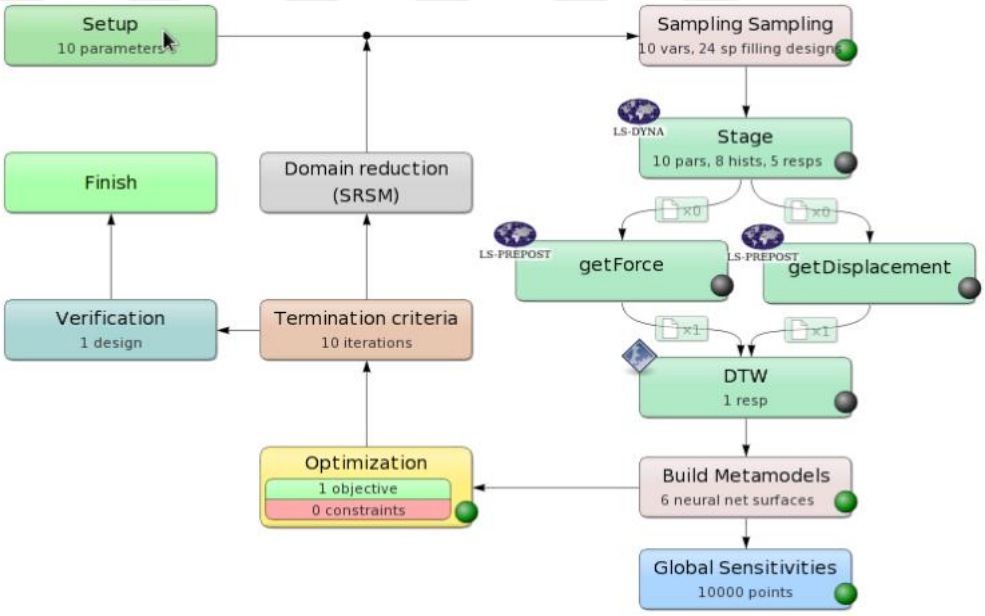

Figure 2.5 : Optimization flowchart in LS-Opt. [15].

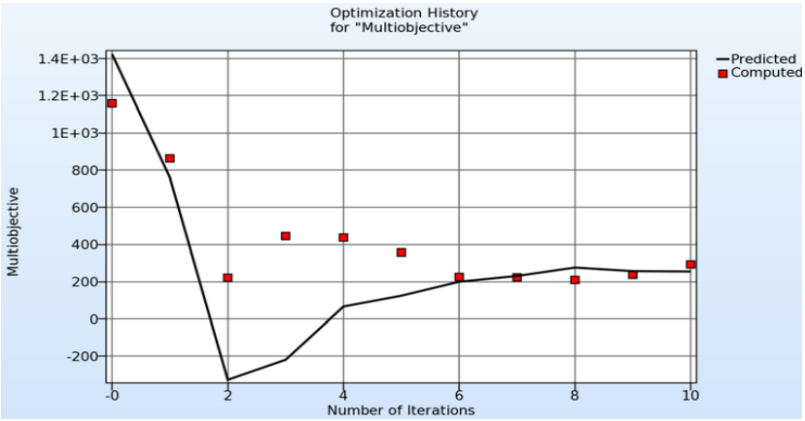

Figure 2.6 : Convergence of multiobjective optimization [15].

Albanesi et al. proposed a metamodel-based optimization framework to minimize the weight of composite laminated wind turbine blades without compromising structural integrity. Their method combines artificial neural networks, ANNs as surrogate models with a genetic algorithm (GA) for multi-variable optimization. The architecture of the ANN model is shown in Figure 2.7. To build the surrogate model, a dataset of input–output pairs was generated using Latin Hypercube Sampling and evaluated through finite element analysis (FEA). The ANN was then trained to replicate the FEA output with high accuracy, thus replacing the costly simulation model within the optimization loop [16].

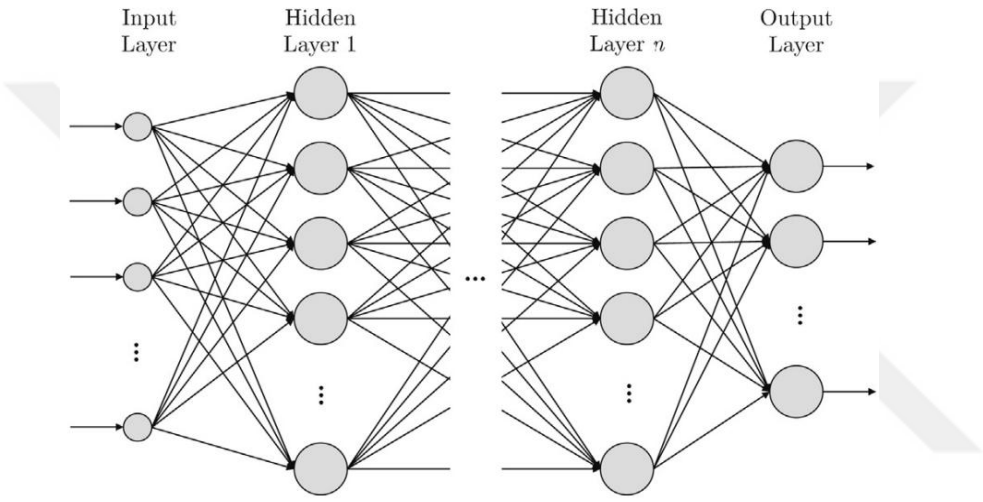

Figure 2.7 : ANNs model architecture [16].

The authors validated the ANN predictions against the FEA results, demonstrating high accuracy in displacement, mass, and strain energy estimations. Once validated, the ANN was coupled with the GA to perform optimization. According to Table 2.1 the optimized blade configuration achieved using ANN+GA resulted in a 20.2% reduction in mass compared to the reference model, while also decreasing maximum displacement by 3.5% and increasing strain energy by 1.5%, indicating enhanced stiffness. Additionally, this metamodeling approach led to an approximate 40% reduction in computational time compared to direct optimization using FEA alone.

Table 2.1 : Comparison of reference and optimum results [16].

Characteristic	Reference	IFEM + GA	ANN + GA
Number of Plies	11	9	7
Blade weight	28.82 kg	24.28 kg	22.85 kg
Tip deflection	0.3500 m	0.3425 m	0.3560 m

This study showcases the effectiveness of combining surrogate modeling with evolutionary algorithms in solving complex structural optimization problems, especially when dealing with computationally expensive simulations such as those involving laminated composite structures.

3. METAMODEL BASED OPTIMIZATION OF METAL PLATE UNDER IMPACT LOADING

This chapter aims to optimize the thickness of an AL7050 aluminum alloy plate subjected to impact loading. The main goal is to minimize both the maximum stress and the mass of the plate. The problem is formulated as a metamodel-based multiobjective optimization, which is suitable for problems with multiple objectives that lack mathematical formulations. In this case, the optimization can be done using mathematical expressions, but in the next part of the study, the optimization problem becomes more complex. Therefore, metamodel based multi objective optimization is appropriate for this study.

LS-DYNA is a powerful finite element software commonly used in dynamic simulations. It is one of the most common programs for simulating impact loading. The software provides a variety of material cards, allowing users to model different types of materials. It also offers many contact definitions that help simulate contact behavior more accurately. In addition, users can easily control time steps, output settings, initial conditions, and other key parameters.

Finite Element Method (FEM) data from LS-DYNA is used to analyze the impact behavior of the plate. Based on this simulation data, surrogate models are built using the Kriging method, and a genetic algorithm is applied to find the optimal solution. This approach helps reduce the computational cost by approximating the system's behavior, allowing efficient exploration of the design space. Objective functions are listed in equation 3.1 and 3.2

Design Parameters (x):

Aluminum plate thickness is the design parameters which is 1.5 mm to 3.5mm.

Objective Functions

1. Minimize the total mass of the aluminum plate:

$$f_1(x) = Mass\ (x) \tag{3.1}$$

2. Minimize the maximum Von Mises stress:

$$f_2(x) = vonMisesStress\ (x) \tag{3.2}$$

Constraint:

The safety factor is selected between 1.2 – 1.5 and corresponding stress range is 300 MPa- 380 MPa.

3.1 LS-Dyna FEM model

The material used in this study is AL7050, a high-strength aluminum alloy commonly used in aerospace and structural applications due to its excellent strength to weight ratio, processability, and availability. The mechanical properties of the material are shown in Table 3.1 taken from Smiths Metal Centres [17].

Table 3.1 : Properties of AL7050.

Properties	Properties Value
Density	2.70 g/cm^3
Modulus of Elasticity	72 GPa
Yield Strength	455 MPa

The plate dimensions are 100 mm × 100 mm, and the thickness is the design variable to be optimized. Out of plane impact loading is simulated using LS-DYNA, a software widely used for both explicit and implicit dynamic analysis. The plate is modeled using 2D quadrilateral elements to reduce analysis time, since many simulations are needed to generate the design space. All edges of the plate are fixed with constraints in x, y, and z directions. The impactor is a spherical rigid steel ball modeled using 3D hexahedral elements and hits the center of the plate at 10 m/s, as shown in Figure 3.1.

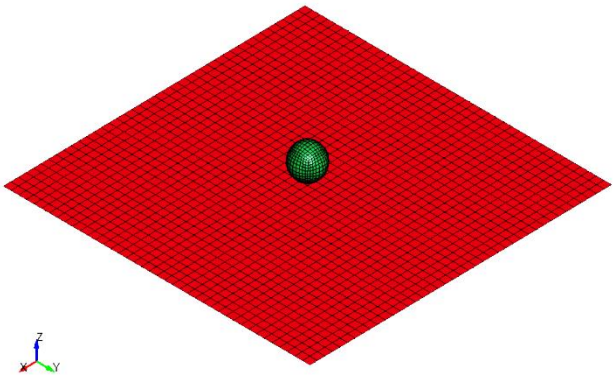

Figure 3.1 : AL7050 plate under out of plane impact loading FEM model.

3.2 LS-Dyna FEM model

To perform the optimization, surrogate modeling is used since deriving a mathematical function for impact behavior is highly complex. Kriging is chosen for this purpose due to its accuracy and flexibility. Kriging is a commonly applied surrogate modeling technique in computational engineering, originally introduced by the mining engineer Krige. According to Jack P.C. Kleijnen , Krige addressed the challenge of combining outcomes obtained from a limited number of sampling locations in gold mining. The method was later refined by the French mathematician Matheron, who established the field of geostatistics, also known as spatial statistics. Its foundation lies in the stationary Gaussian process (GP), characterized by a constant mean and variance, with covariances that depend solely on the distances between points in a three-dimensional space (length, width, and height). This framework defines a multivariate normal, or Gaussian, distribution [18]. The Kriging model is trained using simulation data from LS-DYNA, which includes various plate thickness values and their corresponding maximum stress and mass values. Figure 3.2 and Figure 3.3 show the Kriging models built in MATLAB, with the optimized solution marked with a star.

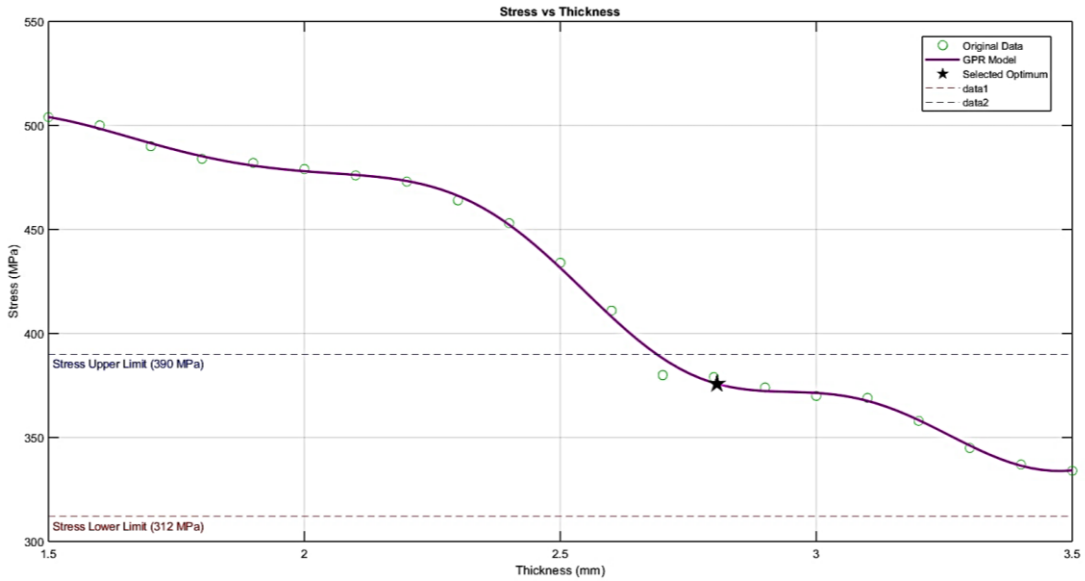

Figure 3.2 : Thickness vs. stress curve from MATLAB.

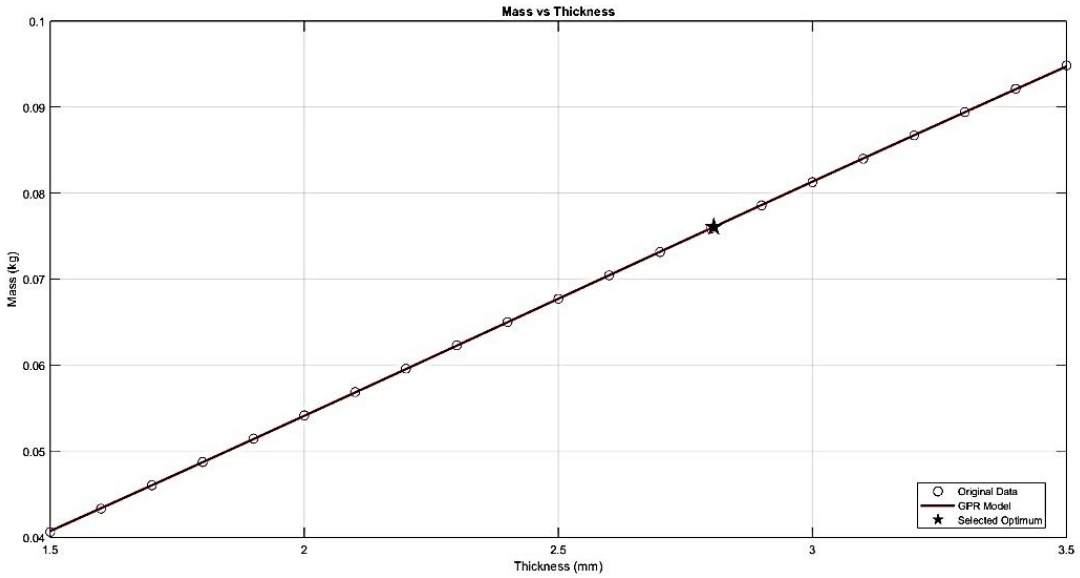

Figure 3.3 : Thickness vs. mass curve from MATLAB.

Since the problem involves two objective functions such as stress and mass a multiobjective optimization approach is needed.

The optimization is performed using a genetic algorithm (GA), which is well-suited for problems with complex design spaces. According to Saraswat and Sharma , MATLAB can be used to implement GA effectively, and increasing the population size can improve the performance of the optimization. GA helps find nearly optimal solutions in a short time and is robust against local minima [14]. Result of optimization is shown in Figure 3.2 and Figure 3.3 with a star and optimum parameters are given in Table 3.2.

Table 3.2 : Optimum plate thickness properties.

Optimum Thickness	2.8059 mm
Optimum Stress	375.78 MPa
Optimum Mass	0.076087 kg

The optimization process consists of four main steps. First of all is simulation data collection. LS-DYNA simulations are performed to identify the maximum von Mises stress and mass for different plate thicknesses. The time step is kept constant across all simulations. It was observed that the peak stress occurs when the impactor first contacts the plate. Figure 3.4 illustrates a typical stress distribution from LS-DYNA.

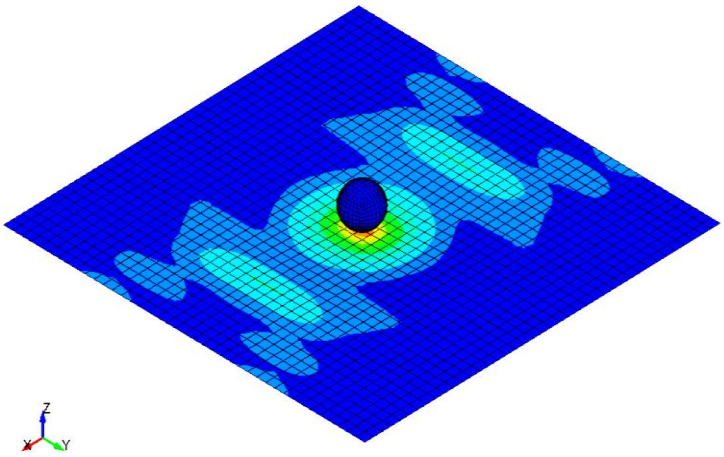

Figure 3.4 : Von mises stress distribution of AL7050 Plate.

Second step is metamodel creation, kriging models are developed to approximate the relationships between plate thickness and two objective function, stress and mass. The mass model is nearly linear, the stress model is nonlinear and more complex. These surrogate models provide fast approximations without the need for new simulations. Another important step is objective function combination: Stress and mass are combined into a weighted function. The weights can be adjusted depending on which performance metric is more critical. For example, assigning more weight to stress leads to a thicker optimal design. This flexibility makes weighted functions practical even for complex problems. Finally, the last step is genetic algorithm optimization. It involves using a genetic algorithm to search the design space efficiently. The algorithm identifies a thickness value that minimizes both stress and mass while meeting design requirements. This approach achieves a balance between structural integrity and material efficiency.

In summary, this study demonstrates the successful optimization of AL7050 aluminum plate thickness under out-of-plane impact loading using a metamodel-based multi-objective approach. The use of Kriging surrogate modeling allowed accurate approximations of stress and mass responses, significantly reducing computational cost. By applying a weighted objective function and solving the problem with a genetic algorithm, an optimal thickness was determined that satisfies both performance and efficiency criteria. This method proves to be effective and practical, especially for engineering problems in aerospace and structural design.

4. METAMODEL BASED MULTIOBJECTIVE OPTIMIZATION OF COMPOSITE PLATE UNDER IMPACT LOADING

In this chapter, a metamodel-based multi-objective optimization is performed on a woven composite plate (AS4 8552) subjected to impact loading. The main goal is to maximize the absorbed energy, which is associated with delamination, and to minimize the mass of the plate. The Hashin index is used as a constraint. If the Hashin index exceeds 1, the corresponding case is excluded from the optimization. While the impact behavior of composite plates can be optimized using mathematical equations, as demonstrated in Talagani's study [21], this approach becomes insufficient for more complex problems, such as adhesively bonded joints. However, a metamodel-based multi-objective optimization method is more suitable for these type of problems. Therefore, this optimization method is chosen in the present study.

The optimization problem is formulated as follows:

Design Parameters (x):

The stacking sequence and number of plies of the composite laminate are selected as design parameters. The stacking sequence presented, where each ply orientation is selected from a discrete set of allowable angles (0^o, 45^o). The configuration is assumed to be symmetric with respect to the midplane. Number of plies are also varied as (8, 10, 12, 14, 16).

Objective functions are given in eq. 4.1 and 4.2.

1. Minimize the total mass of the composite laminate:

$$f_1(x) = Mass\ (x) \tag{4.1}$$

2. Maximize the energy absorbed during impact:

$$f_2(x) = -AbsorbedEnergy\ (x) \tag{4.2}$$

(Since optimization is performed by minimization, the minus sign is placed in front of the absorbed energy function.)

Constraint is given in eq. 4.3:

The maximum value of Hashin failure index must remain below the 1, the critical threshold.

$$g(x) = HashinIndex\ (x)\ \leq 1 \tag{4.3}$$

First, the LS-DYNA FEM model is introduced, including the geometrical properties, material properties, and boundary conditions. The initial input file is created using LS-DYNA. Second, the input file is updated with a new stacking sequence combination via a Python script. Third, the stacking sequence rules, stacking sequence selection, and the calculations of Hashin index and absorbed energy are presented. Finally, the metamodel development and the optimization process are described. The summary of this section is given in flow chart shown in Figure 4.1.

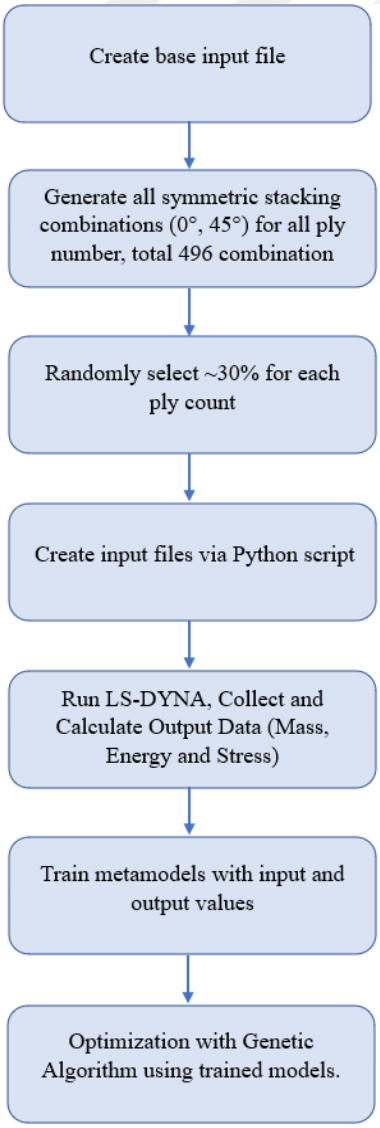

Figure 4.1 : Optimization flow chart.

4.1 LS-DYNA Based FEM Model Approach

LS-DYNA is used to create the input file. It is a suitable program for simulating composite plates under impact loading, as it offers a wide range of material models. The selection of a material card is critical and depends on the specific purpose of the simulation.

In this project, MAT058 (Composite Damage Model) is selected as the material card. This model allows for the prediction of progressive damage in composite materials and includes failure criteria such as the Hashin failure theory, which can distinguish between fiber and matrix damage modes. It also supports strain-rate effects, element deletion, and damage evolution, which makes it appropriate for impact simulations.

The input file defines the geometry, material properties, boundary conditions, initial conditions, contact properties and stacking sequence of the composite plate. Once the initial setup is complete, different stacking sequences are automatically applied and updated via Python scripting to enable parametric studies and optimization.

The composite plate has dimensions of 100 mm × 100 mm. A rigid impactor, modeled as a sphere with a radius of 4 mm, strikes the center of the plate at a velocity of 50 m/s in z direction. The FEM demonstration is given in Figure 4.2.

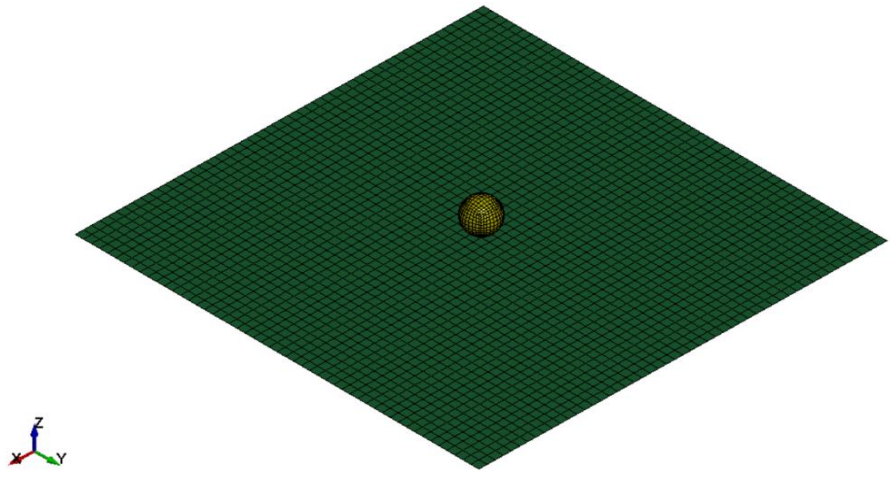

Figure 4.2 : Composite plate FEM model in LS-DYNA.

After defining the geometry and impact conditions, boundary conditions are applied to the model. In this simulation, all four edges of the composite plate are fully constrained in the x, y, and z directions, meaning that translation is restricted in all degrees of freedom. This configuration prevents any edge displacement and simulates a fully clamped plate, which is a typical boundary condition in impact analyses.

In LS-DYNA, these constraints are implemented using the *BOUNDARY_SPC_SET keyword. A node set is created that includes all nodes along the edges of the plate, and this set is assigned fixed constraints in the three translational directions (DOF 1, 2, and 3). This setup ensures that the plate cannot move along its edges during the impact.

This boundary condition setup is illustrated in Figure 4.3.

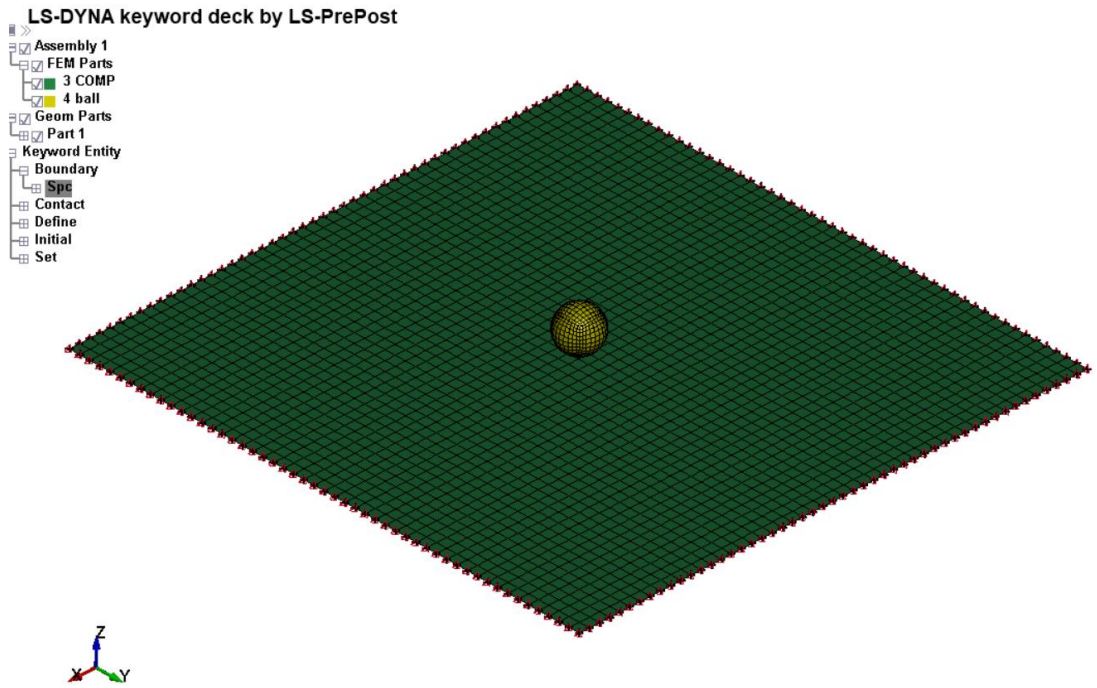

Figure 4.3 : Boundary conditions of FEM.

To model the interaction between the impactor and the composite plate, an *AUTOMATIC_SURFACE_TO_SURFACE contact is defined in LS-DYNA. In this setup, the rigid impactor ball is assigned as the master surface, while the composite plate is defined as the slave surface. This contact type automatically detects and handles contact between the two surfaces of parts during the simulation, allowing for accurate tracking of force transmission and potential separation or sliding at the contact interface.

The composite material used in this study is AS4 8552, a woven carbon epoxy system commonly used in aerospace applications. The material properties of AS4 8552 woven are given in Table 4.1 and properties are taken from Ata and Coker [19]. In the LS-DYNA model, the composite behavior is defined using the *MAT058 material card. The stacking sequence is assigned through the *COMPOSITE_ LONG keyword, which allows the definition of ply orientations, thicknesses, and material directions layer by layer.

Table 4.1 : Optimum plate thickness properties.

Physical Properties	V_f (%)	55.29
	ρ_c (g/cm^3)	1.79
	t (mm)	0,280
Elastic Properties	E_1 (MPa)	64000
	E_2 (MPa)	64000
	E_3 (MPa)	8500
	ν_{12}	0,05
	ν_{13}	0,3
	ν_{23}	0,3
	G_{12} (MPa)	4900
	G_{13} (MPa)	3700
	G_{23} (MPa)	3700
Plastic Properties	$(\sigma_{1T})_{ult}$ (MPa)	647
	$(\sigma_{1C})_{ult}$ (MPa)	657
	$(\sigma_{2T})_{ult}$ (MPa)	647
	$(\sigma_{2C})_{ult}$ (MPa)	657
	$(\tau_{12})_{ult}$ (MPa)	92
Fracture Properties	G_c Pure I (100% GI) DCB	0,375 N/mm
	G_c Mixed (68% GI, 32% GII) ADBC (J/m^2)	1,467 N/mm

The composite material is modeled as a symmetric laminate. In the initial input model, the stacking sequence was defined as $[0]_{16}$ which is shown in Figure 4.4.

```
*PART_COMPOSITE_LONG
$#                                                                      title
COMP
$#     pid    elform      shrf      nloc     marea      hgid    adpopt    thshel
         3         2       0.0       1.0       0.0         0         0         0
$#    mid1    thick1        b1    tmid1    plyid    shrfac    unused    unused
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
        32.80000E-4     0.0         0         0       0.0
```

Figure 4.4 : Boundary conditions of FEM.

After creating the initial input file, multiple input and output files are required to train the metamodel. To achieve this, a design space is generated that includes all possible

stacking sequence combinations. Two ply orientation angles are considered: 0° and 45°, and five different total ply numbers are evaluated: 8, 10, 12, 14, and 16. Since the laminate must be symmetric, the number of unique stacking combinations is reduced. For example, an 8 ply laminate has $2^4 = 16$ possible symmetric combinations, while a 10 ply laminate has $2^5 = 32$ combinations.

In total, 496 symmetric stacking combinations are generated. However, to reduce computational cost, the metamodel is trained using only a subset of these combinations. For each ply count, approximately 30% of the possible stacking sequences are randomly selected for training. Table 4.2 presents example input combinations. A value of "−1" indicates that the corresponding ply position is deleted in that specific stacking configuration.

Table 4.2 : Input configuration table.

	Input_0001	**Input_0002**	**Input_0003**	**Input_0004**	**Input_0005**
Ply1	0	-1	-1	-1	-1
Ply2	45	0	-1	-1	-1
Ply3	0	45	0	-1	-1
Ply4	45	0	45	45	-1
Ply5	0	0	45	0	0
Ply6	0	45	0	45	0
Ply7	45	45	0	0	0
Ply8	45	0	45	45	45
Ply9	45	0	45	45	45
Ply10	45	45	0	0	0
Ply11	0	45	0	45	0
Ply12	0	0	45	0	0
Ply13	45	0	45	45	-1
Ply14	0	45	0	-1	-1
Ply15	45	0	-1	-1	-1
Ply16	0	-1	-1	-1	-1
Number of Plies	16	14	12	10	8
Number of Combinations	256	128	64	32	16
Number of Randomly Selected Configurations	80	40	20	10	5

4.1.1 Mass

Mass is the first objective function, which changes with the number of plies. Therefore, there is no need to create a metamodel for mass. However, the validation results show that the trained mass model is very close to the actual values, so a surrogate model can still be used to predict mass. To obtain mass values for randomly selected stacking combinations, data is extracted from the "d3hsp" output file of LS-DYNA, which includes the mass of each part and the total mass. In this study, only the mass of the composite laminate is considered.

4.1.2 Absorbed energy

Another objective function is the absorbed energy, which represents the amount of energy dissipated by the composite plate during impact loading. A higher absorbed energy value generally indicates improved impact resistance, as the structure can endure more energy through damage mechanisms such as matrix cracking, fiber breakage, and delamination. In this study, the Hashin index is defined as a constraint to ensure structural integrity. Accordingly, the optimization process aims to maximize the absorbed energy while preventing fiber or matrix failure, as indicated by a Hashin index value remaining below one. The absorbed energy is not included in any outputfile. Consequently, the energy values are extracted from the "glstat" file generated by LS-DYNA, which contains time-history data for global energies, then absorbed energy calculations are done according to eq 4.4.

$$E_{absorbed} = E_{kinetic,initial} + E_{kinetic,final} \tag{4.4}$$

where:

$E_{absorbed}$: is the energy absorbed by the structure during the impact in Joules,

$E_{kinetic,\ initial}$: is the kinetic energy at the initial of the impact in Joules,

$E_{kinetic,\ final}$: is the kinetic energy at the final of the impact in Joules,

4.1.3 Hashin criterion

Von Mises stress is commonly used to detect damage initiation in metal components. For composite materials, several failure indices are available, such as Tsai-Wu, Tsai-Hill, and Hashin criteria. The choice of failure criterion depends on the specific application. For example, according to Bartholomeusz and Rhead **,** in a pressurized

vessels made of composite materials, the Tsai-Wu criterion can be used to predict damage initiation [20]. On the other hand, for modeling impact damage in a composite plate using LS-Dyna MAT58, the Hashin criterion is more suitable and commonly preferred which is given in LS-DYNA Keyword User's Manual – MAT058 (MAT_COMPOSITE_DAMAGE) [21].

The Hashin criterion also provides information about the failure type and the location where damage initiation occurs. It includes four failure modes: fiber tension failure ($\sigma_1 \geq 0$), fiber compression failure ($\sigma_1 < 0$), matrix tension failure ($\sigma_2 \geq 0$), and matrix compression failure ($\sigma_2 < 0$) given in eq 4.5, 4.6, 4.7 and 4.8 [22].

Fiber Tension Failure Index (F_{ft}), ($\sigma_1 \geq 0$)

$$F_{ft} = \left(\frac{\sigma_1}{X_t}\right)^2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2 \tag{4.5}$$

Failure occurs if: $F_{ft} \geq 1$

Fiber Compression Failure Index (F_{fc}), ($\sigma_1 < 0$)

$$F_{fc} = \left(\frac{\sigma_1}{X_c}\right)^2 \tag{4.6}$$

Failure occurs if: $F_{fc} \geq 1$

Matrix Tension Failure Index (F_{mt}), ($\sigma_2 \geq 0$)

$$F_{mt} = \left(\frac{\sigma_2}{Y_t}\right)^2 + \left(\frac{\tau_{12}}{S_{12}}\right)^2 \tag{4.7}$$

Failure occurs if: $F_{mt} \geq 1$

Matrix Compression Failure Index (F_{mc}), ($\sigma_2 < 0$)

$$F_{mc} = \left(\frac{\sigma_2}{2S_{12}}\right)^2 + \left(\frac{S_{12}}{Y_c} - 1\right)\left(\frac{\sigma_2}{Y_c}\right) + \left(\frac{\tau_{12}}{S_{12}}\right)^2 \tag{4.8}$$

Failure occurs if: $F_{mc} \geq 1$

Where:

σ_1 = stress in fiber direction

σ_2 = stress in transverse direction

τ_{12} = shear stress in the plane

X_t, X_c = tensile and compressive strength in fiber direction

Y_t, Yc = tensile and compressive strength in matrix direction

S_{12} = in plane shear strength

4.2 Metamodel Based Multiobjective Optimization

Trained models are developed using input and output data. A critical step in this process is selecting an appropriate training method. The GPR (Gaussian Process Regression) method is chosen for all models. Other methods were also tested, but the GPR method produced the lowest RMSE (Root Mean Square Error) values. Therefore, GPR is selected for training the surrogate models of mass, absorbed energy, and Hashin index. Metamodel responses and MSE (Mean Square Error) graph of trained mass, absorbed energy and Hashin metamodels are given in Figure 4.5, Figure 4.6 and Figure 4.7.

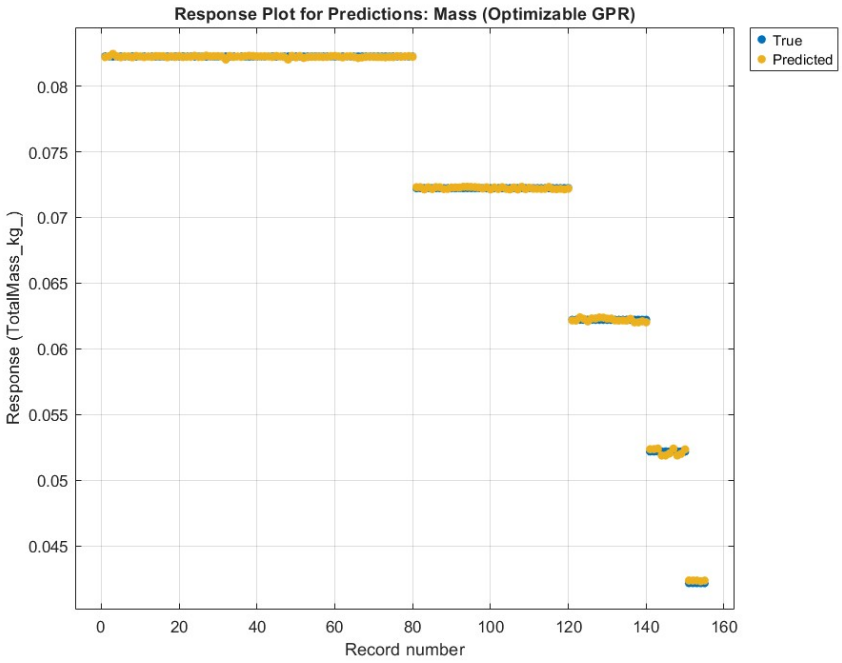

Figure 4.5 : Response of actual and metamodel prediction for Mass.

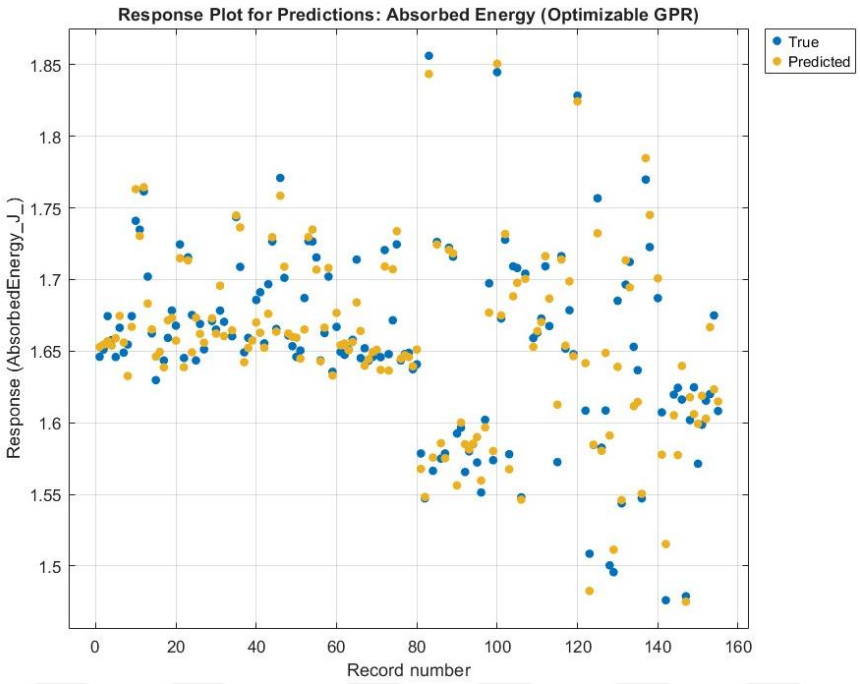

Figure 4.6 : Response of actual and metamodel prediction for von Mises stress.

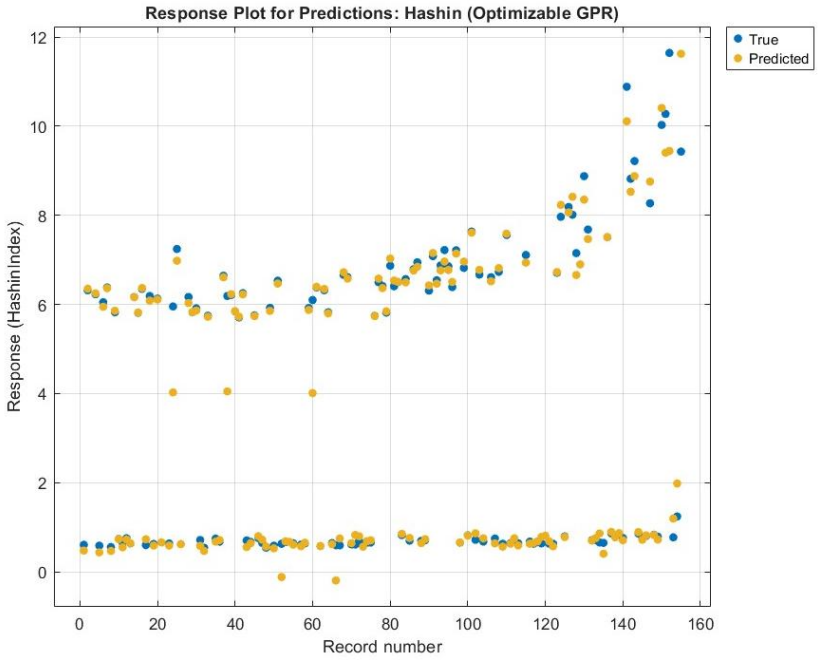

Figure 4.7 : Response of actual and metamodel prediction for Hashin index.

The optimization process was carried out in MATLAB using the “gamultiobj” function, which is a built in tool for solving multi-objective optimization problems using a genetic algorithm (GA). This function is particularly useful when there are two or more conflicting objectives, as in this study where the aim is to minimize the mass and maximize the absorbed energy of the composite plate. The gamultiobj algorithm searches the design space using evolutionary principles such as selection, crossover,

and mutation. Instead of providing a single solution, it generates a set of Pareto optimal solutions, which represent the best trade-offs between the objectives. The Pareto front shows the set of nondominated solutions were improving one objective would worsen the other. This allows designers to select the most suitable stacking sequence according to their specific design priorities. The optimal stacking sequence is shown in Figure 4.8.

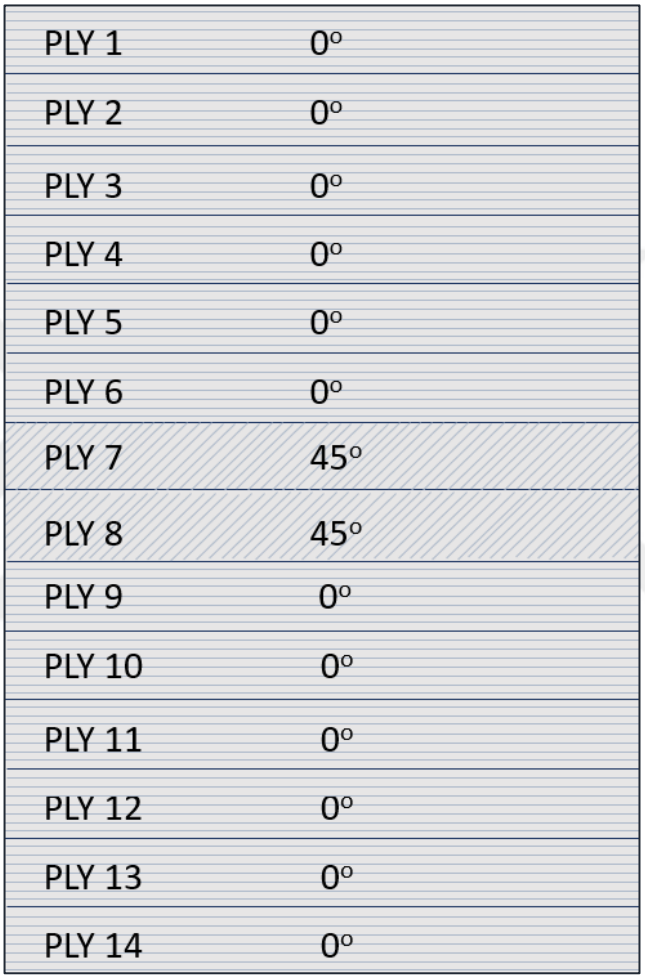

Figure 4.8 : Optimum stacking sequence.

In addition, optimal point mass, absorbed energy and hashin index are given in Table 4.3.

Table 4.3 : Optimum point properties.

Optimum Stacking Sequence	$[0^o\ 0^o\ 0^o\ 0^o\ 0^o\ 0^o\ 45^o]_s$
Optimum Ply Number	14
Optimum Lamina Mass	0.072 kg
Optimum Lamina Absorbed Energy	1.845 J
Optimum Lamina Hashin Index	0.818

5. METAMODEL BASED MULTIOBJECTIVE OPTIMIZATION OF ADHESIVELY BONDED COMPOSITE PLATE AND ALUMINUM PLATE STRUCTURE UNDER IMPACT LOADING

In this chapter, a metamodel-based multi-objective optimization is applied to an adhesively bonded structure consisting of a woven composite plate (AS4 8552) and an aluminum plate (AL7050), both subjected to impact loading. The two main objectives are to minimize the von Mises stress in the aluminum plate and to reduce the overall mass of the structure. The Hashin failure index is used as a constraint for the composite laminate; any configuration with a Hashin index exceeding 1 is considered to have failed and is excluded from the optimization process.

Due to the complexity and nonlinear nature of the problem, it is not feasible to define the objective functions and constraints analytically. Therefore, metamodels are developed based on finite element simulation results and are used as surrogate models to represent both the objectives and the constraint.

A genetic algorithm is selected as the optimization method. This algorithm is inspired by the process of natural selection and is especially effective for solving problems with large, complex design spaces where gradients are not easily defined. It operates by generating a population of possible solutions and evolving them over successive generations through operations such as selection, crossover, and mutation. Its robustness and flexibility make it well suited for this type of optimization problem.

The formulation of the optimization problem is presented below:

Design Parameters (x):

the design parameters include the stacking sequence, the total number of plies in the composite laminate, and the thickness of the aluminum plate. Each ply orientation in the stacking sequence is selected from a predefined set of discrete angles: 0° and 45°. The laminate is assumed to be symmetric about its mid-plane, which reduces the number of independent design variables.

The number of plies is varied across five values: 8, 10, 12, 14, and 16. Similarly, five different aluminum plate thicknesses are considered: 0.0004, 0.0006, 0.0008, 0.001, and 0.0012 meters. These parameters are explored to evaluate their combined influence on the structural performance under impact loading.

Objective Functions are given in eq 5.1 and eq 5.2.

1. Minimize the total mass of the composite laminate and aluminum plate:

$$f_1(x) = Mass\ (x) \tag{5.1}$$

2. Minimize von-Mises stress on aluminum plate during impact:

$$f\_2\ (x) = vonMises(x) \tag{5.2}$$

Constraint is given in eq. 5.3:

The maximum value of Hashin failure index for composite plate must remain below the 1, the critical threshold.

$$g(x) = HashinIndex\ (x)\ \leq 1 \tag{5.3}$$

This section begins with the development of the LS-DYNA finite element model, where the geometry, material definitions, and boundary conditions are introduced. The initial input file is generated in LS-DYNA based on these definitions. In the next step, a Python script is used to modify the input file by assigning different stacking sequences and aluminum thickness values.

Following this, the selection of configurations is explained, along with the calculation procedures for the Hashin failure index, von Mises stress, and mass. Lastly, the metamodel construction and the optimization strategy are detailed. An overview of the entire process is illustrated in the flowchart provided in Figure 5.1.

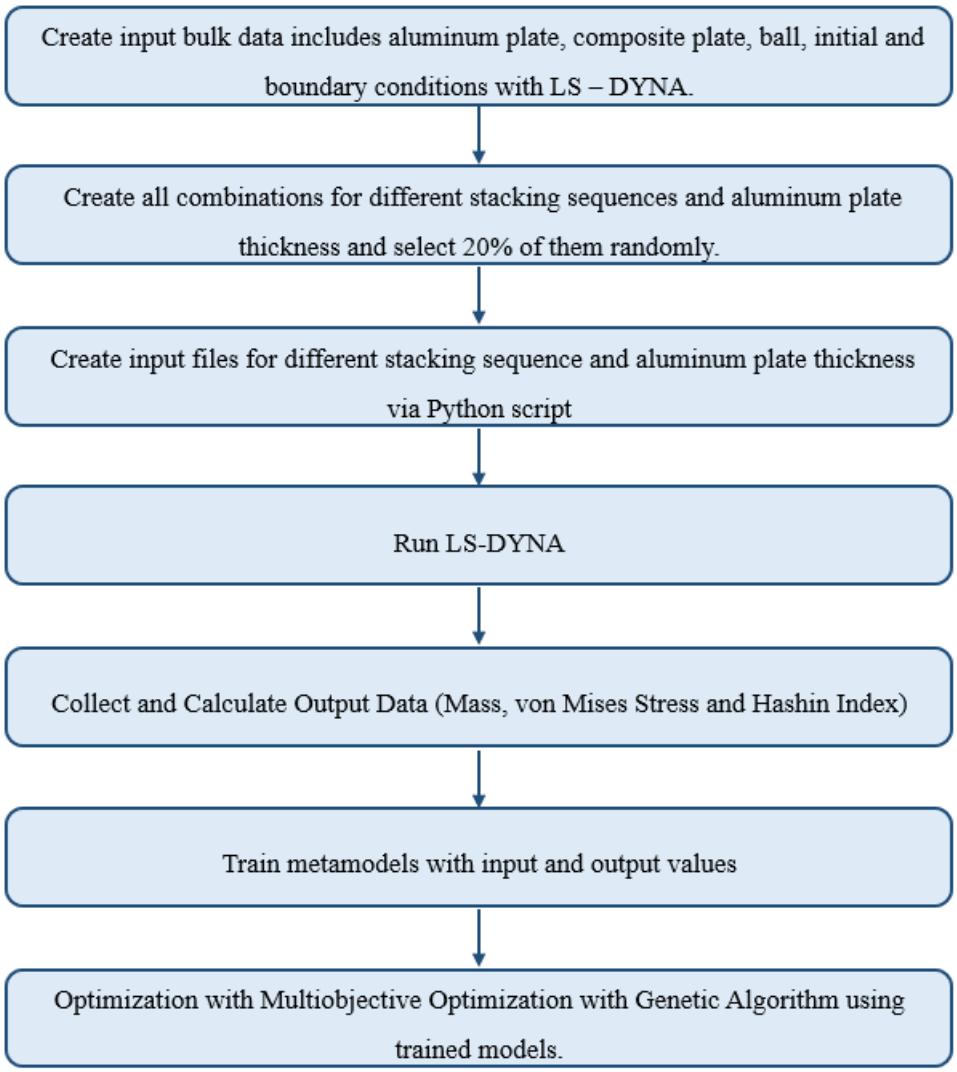

Figure 5.1 : Optimum stacking sequence.

5.1 LS-DYNA Based FEM Model Approach for Adhesively Bonded Model

The finite element input file is prepared using LS-DYNA, which offers robust capabilities for modeling impact behavior in composite-metal bonded structures. This includes cohesive zone modeling, which plays a crucial role in accurately representing the adhesive interface between dissimilar materials.

In this study, MAT_058 (Composite Damage Model) is used to define the mechanical behavior of the composite plies, accounting for progressive damage under impact. To model the adhesive layer, MAT_138 (Cohesive Mixed Mode) is selected for the cohesive zone elements. These elements are zero-thickness solid elements inserted between the composite and metal plates, enabling simulation of delamination and debonding under mixed-mode loading conditions (i.e., normal separation, shear sliding, or their combination). The cohesive law embedded in these elements describes

the traction–separation behavior and allows for the prediction of damage initiation and growth based on predefined fracture toughness parameters in Modes I and II.

The input file includes detailed definitions of the model geometry, material properties, boundary and initial conditions, contact interfaces, composite stacking sequence, and the thickness of the aluminum plate. Once the baseline configuration is established, a Python script is employed to automatically modify stacking sequences and aluminum thicknesses across different cases, enabling parametric simulations and optimization workflows.

A rigid sphere with a radius of 5 mm is used as the impactor, which strikes the center of the composite-metal plate assembly at a velocity of 50 m/s along the z-axis. Both the composite and metal plates have 250 mm in length and 50 mm in width, with 50 mm overlap length. The finite element representation of the model is shown in Figure 5.2.

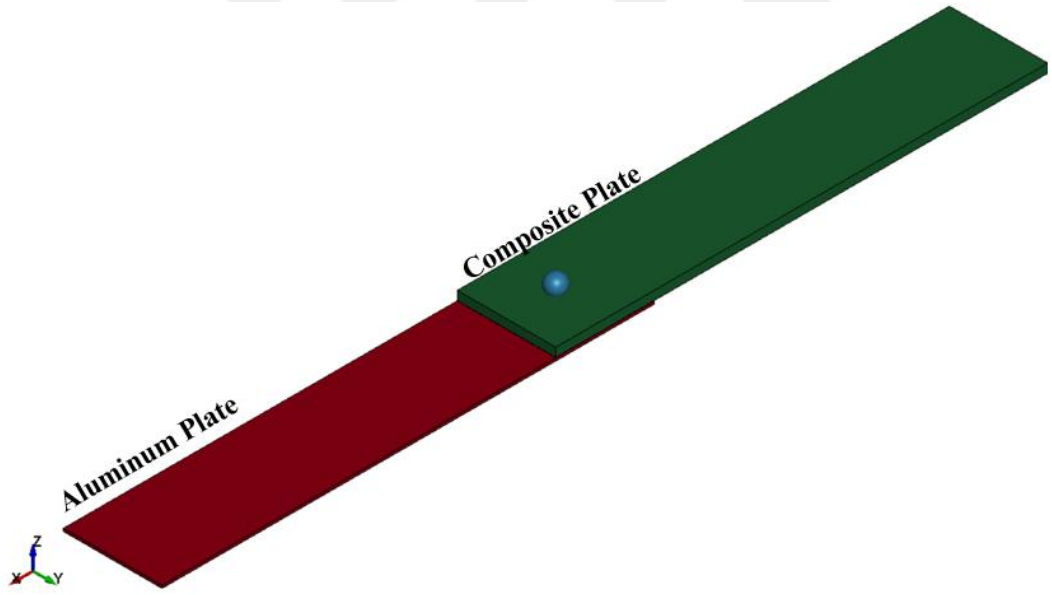

Figure 5.2 : Adhesively bonded FEM model in LS-DYNA.

In LS-DYNA, fixed-end boundary conditions are applied using the *BOUNDARY_SPC_SET keyword. A node set is defined that includes the nodes located along both ends of the composite and metal plates, representing a fixed-ended plate configuration. These nodes are constrained in all three translational degrees of freedom (DOF 1, 2, and 3), effectively restricting any movement at the plate ends during the impact simulation. This boundary condition setup is given in Figure 5.3.

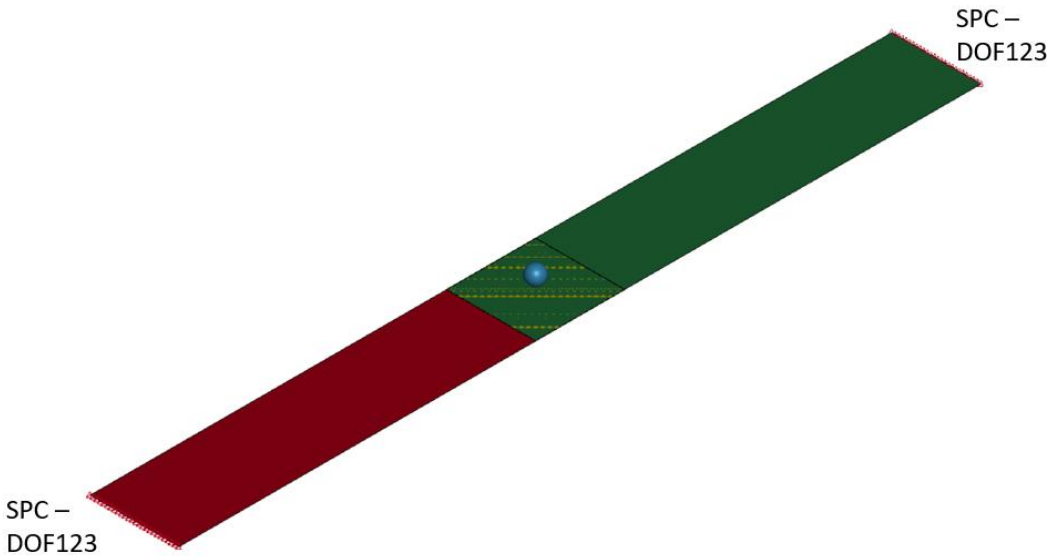

Figure 5.3 : Boundary conditions of FEM.

To simulate the interaction between the impactor and the composite plate, an *AUTOMATIC_SURFACE_TO_SURFACE contact definition is used in LS-DYNA. In this configuration, the rigid impactor is defined as the master surface, and the composite plate as the slave surface.

To ensure proper bonding between the composite, cohesive, and metal layers, *TIED_SURFACE_TO_SURFACE_OFFSET contact definitions are employed in the model. The first tied contact is defined between the top surface of the cohesive zone elements and the adjacent composite shell elements located directly above. In this setup, the composite shell is assigned as the master surface and the cohesive zone elements as the slave surface. Similarly, another tied contact is defined between the bottom surface of the cohesive elements and the underlying metal shell. In this case, the cohesive zone elements are set as the master surface, and the metal shell is defined as the slave surface. These tied contacts effectively constrain the corresponding surfaces, preventing separation and ensuring load transfer across the bonded interfaces. The contact definitions are illustrated in Figure 5.4 and Figure 5.5.

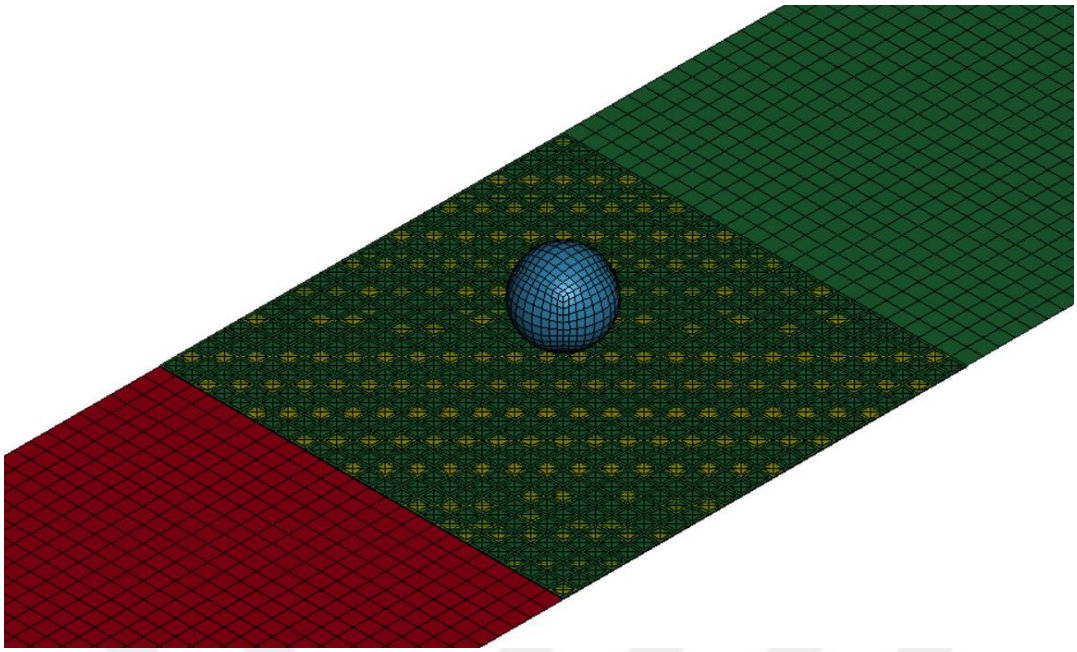

Figure 5.4 : Upper shell (composite plate) to cohesive zone element contact.

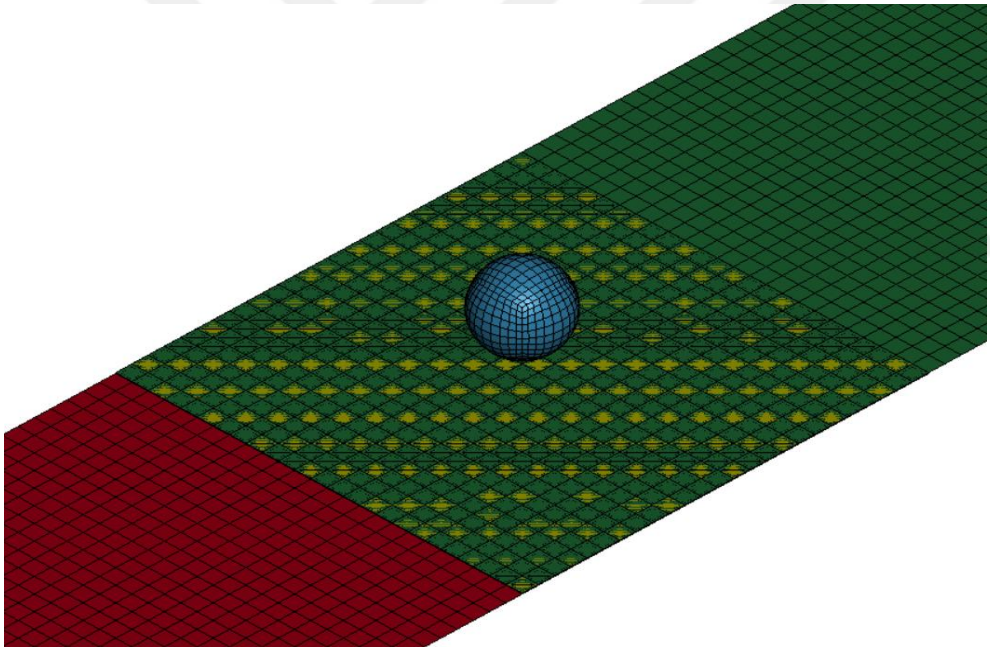

Figure 5.5 : Cohesive zone element to lower shell (aluminum plate) contact.

AS4 8552 woven composite is used as the composite material, while AL7050 is selected for the aluminum plate. FM300K.05 film epoxy is chosen as the adhesive which material properties are given in Table 5.1. The properties are taken from 3M [23], Alkoç et al. [24], Cytec Solvay Group [25], and Jokinen [26].

Table 5.1 : Properties of FM300K.05 film epoxy adhesive. [23-26].

Symbol	Property	Value	Unit
ρ	Density	875	kg/m^3
t	Thickness	0.2	mm
E	Elastisity Modulus	3.10	GPa
ν	Poisson Ratio	0.38	-
G	Shear Modulus	58	GPa
t_1^0	Nominal Tensile Stress (Mode I)	2.3	MPa
t_2^0	Nominal Shear Stress (Mode II)	36.8	MPa
t_3^0	Peel Stress (Mode III)	4.0	kN/m
G_{Ic}	Mode I Fracture Toughness	1600-2000	J/m^2
G_{IIc}	Mode II Fracture Toughness	4897	J/m^2

The composite structure is modeled as a symmetric laminate. In the baseline input model, the aluminum shell thickness is specified as 0.0004 m, and the composite layup is defined as $[0]_{16}$, as presented in Figure 5.6.

```
*SECTION_SHELL_TITLE
al_shell
$#   secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
         1         2       1.0         2       1.0         0         0         1
$#      t1        t2        t3        t4      nloc     marea      idof    edgset
    0.0004    0.0004    0.0004    0.0004       1.0       0.0       0.0         0
*PART_COMPOSITE_LONG
$#                                                                         title
composite
$#     pid    elform      shrf      nloc     marea      hgid    adpopt    thshel
         3         2       0.0       1.0       0.0         0         0         0
$#    mid1    thick1        b1     tmid1     plyid    shrfac    unused    unused
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
         32.80000E-4       0.0         0         0       0.0
```

Figure 5.6 : Input model al_shell and COMPOSITE_LONG card.

To build and train the metamodel, a variety of input and output files must be generated based on different design configurations. This process begins with the creation of a design space that incorporates all feasible combinations of stacking sequences and aluminum plate thicknesses.

The study considers five distinct aluminum thickness values: 0.0004, 0.0006, 0.0008, 0.001, and 0.0012 meters. Each ply can take one of two fiber orientation angles, 0° or 45°. Additionally, laminates composed of 8, 10, 12, 14, and 16 total plies are investigated. To maintain symmetry in the layup, only half of the plies are independently varied, which significantly limits the number of valid stacking combinations.

As an example, an 8-ply symmetric laminate results in $2^4 = 16$ unique stacking configurations. When combined with the five thickness values, this yields 80 possible design cases. Extending this logic to all ply counts and thicknesses, the complete design space consists of 2480 configurations.

To reduce computational cost, a subset of 496 configurations representing approximately 20% of all combinations for each ply count is randomly selected for training the metamodel. A sample of these selected input cases is shown in Table 5.2.

Table 5.2 : Examples of stacking sequences and aluminum plate thickness configurations.

	Input_0001	**Input_0002**	**Input_0003**	**Input_0004**	**Input_0005**
Ply1	0	-1	-1	-1	-1
Ply2	45	0	-1	-1	-1
Ply3	0	45	0	-1	-1
Ply4	45	0	45	45	-1
Ply5	0	0	45	0	0
Ply6	0	45	0	45	0
Ply7	45	45	0	0	0
Ply8	45	0	45	45	45
Ply9	45	0	45	45	45
Ply10	45	45	0	0	0
Ply11	0	45	0	45	0
Ply12	0	0	45	0	0
Ply13	45	0	45	45	-1
Ply14	0	45	0	-1	-1
Ply15	45	0	-1	-1	-1
Ply16	0	-1	-1	-1	-1
Aluminum Plate Thickness (mm)	0.0004	0.0006	0.0008	0.001	0.0012
Number of Plies	16	14	12	10	8
Number of Combinations	1280	640	320	160	80
Number of Randomly Selected Combintions	256	128	64	32	16

5.1.1 Mass

Mass is considered as the first objective function, and it directly depends on the total number of plies in the laminate and aluminum plate thickness. Since this relationship is straightforward, there is no strict requirement to construct a metamodel for mass prediction. Nonetheless, validation results indicate that the trained surrogate model for mass yields results that are highly consistent with the actual values. Therefore, using a metamodel remains a practical and efficient option.

To determine the mass corresponding to each randomly selected stacking configuration, data is retrieved from the “d3hsp” output file generated by LS-DYNA. This file provides detailed mass information for individual parts and the overall model. In this study, only the mass of the adhesively bonded plates is extracted and used for analysis.

5.1.2 Von – Mises stress

The stress in the aluminum plate is defined as the second objective function. Stress values are extracted from all elements located in the bonding region of the aluminum plate. To minimize the influence of localized anomalies such as stress singularities the average of the nine highest von Mises stress values is used instead of a single peak value.

For this purpose, von Mises stress is computed for each element using the principal stress components. The von Mises criterion is a scalar value that combines the effects of normal and shear stresses into a single equivalent stress, representing the yield condition in ductile materials. By averaging the top nine values, a more stable and representative stress measure is obtained for use in the optimization process.

5.1.3 Hashin criterion

For the Hashin failure criterion, stress values are extracted from all elements located on the contact surface of the composite plate. Similar to the aluminum stress calculation, the average of the nine highest stress values is taken to represent the stress state. This approach is used to reduce the influence of localized spikes or singularities that may occur at specific elements due to sharp geometrical transitions, mesh refinement, or contact interactions. The detailed calculation procedure has been presented in the previous section.

5.2 Metamodel Based Multiobjective Optimization

Trained surrogate models were developed using input–output data obtained from simulations. A key step in this process was the selection of appropriate training methods. Various algorithms were tested, and for each output variable, the method that provided the best agreement with the simulation results and the lowest root mean square error (RMSE) was selected. For the von Mises stress surrogate model, the bagged trees method was chosen, achieving an RMSE of 18.96. The mass surrogate model was trained using the stepwise linear method with 4.01e-4 while the Hashin index model also employed bagged trees with 0.148.

Bagged Trees Method, Breiman proposed the bagged trees method, also known as bootstrap aggregation, as an ensemble technique that enhances prediction accuracy by reducing variance. He claimed that combining multiple decision trees trained on bootstrapped subsets and averaging their outputs could significantly improve model stability and prevent overfitting. Bagging is particularly effective when applied to high-variance, low-bias models such as decision trees [27].

Stepwise Linear Regression, Draper and Smith explained that stepwise linear regression is a practical technique for selecting a subset of predictor variables in multiple linear regression. They said that this method incrementally adds or removes variables based on statistical criteria, such as p-values or information criteria, to create a parsimonious model. The authors also noted that it is particularly useful when dealing with datasets containing many candidates input features [28].

Response plot for Mass, von Mises and Hashin Index is given in Figure 5.7, Figure 5.8 and Figure 5.9, respectively.

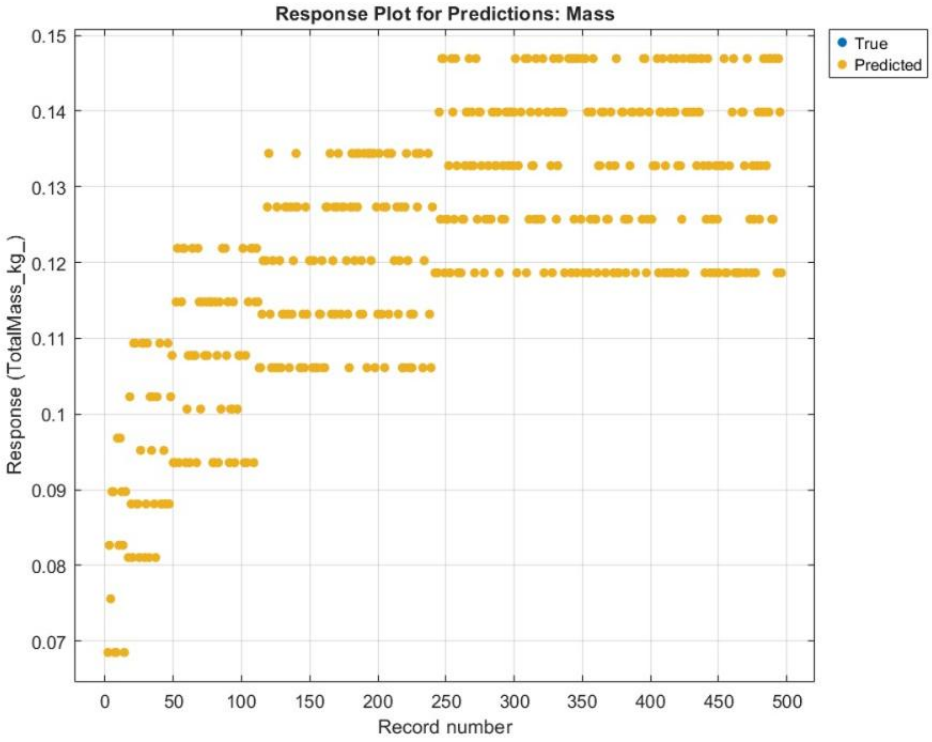

Figure 5.7 : Response of actual and metamodel prediction for Mass.

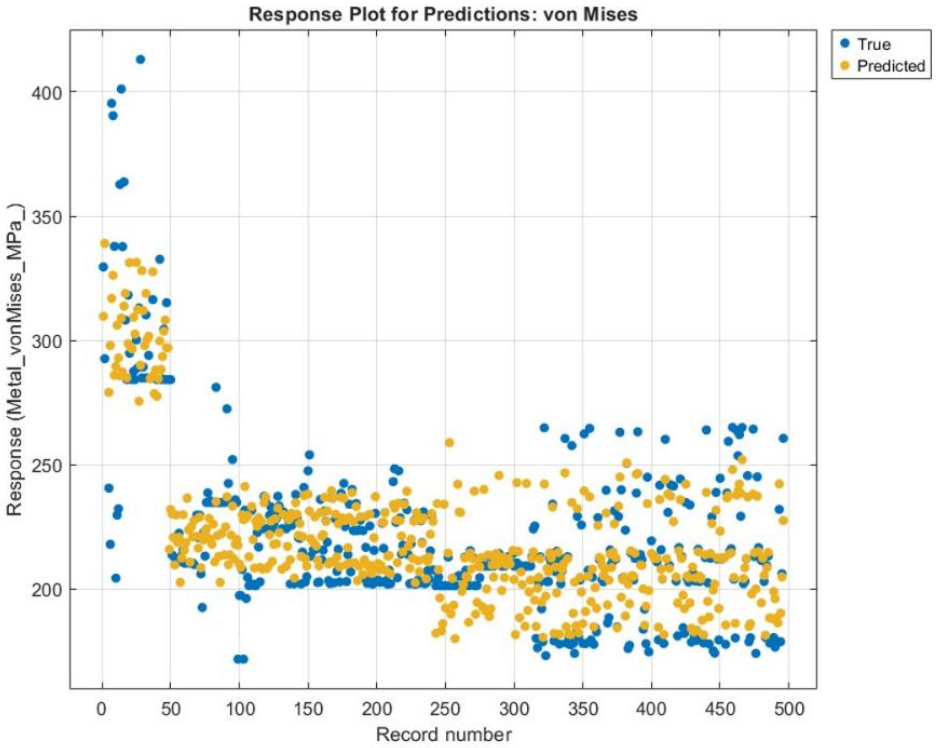

Figure 5.8 : Response of actual and metamodel prediction for von Mises stress.

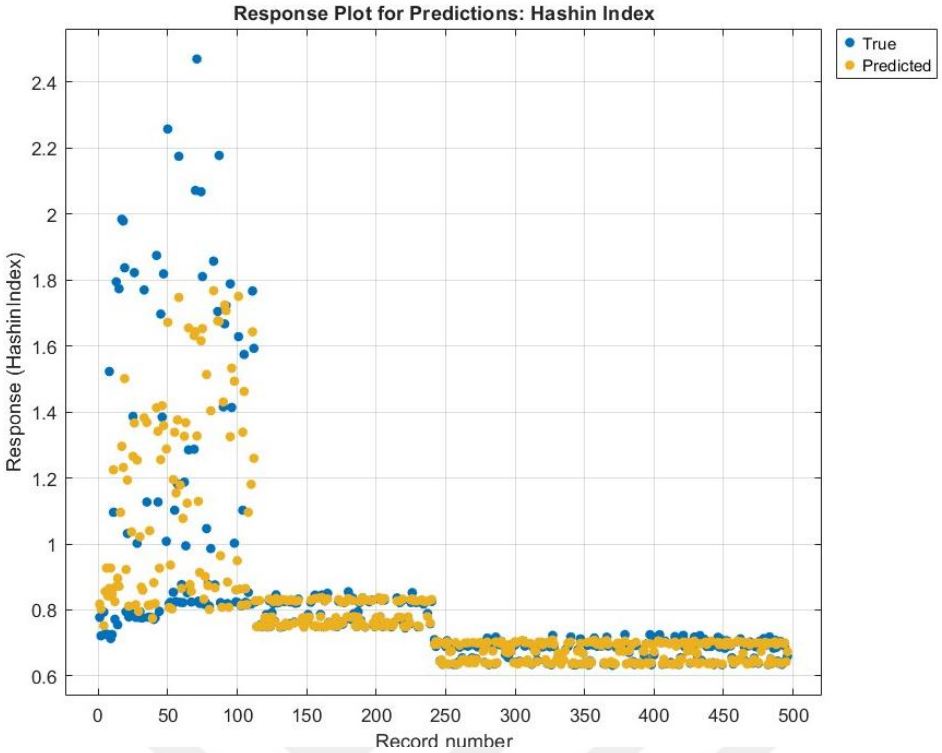

Figure 5.9 : Response of actual and metamodel prediction for Hashin index.

The optimization was done in MATLAB using the “gamultiobj” function. This tool uses a genetic algorithm to solve problems with more than one goal. In this study, the goals were to reduce the mass and decrease the von Mises stress that occurs in the aluminum plate. The algorithm uses steps like selection, crossover, and mutation to search for good results. Instead of giving only one solution, it finds many good options, called Pareto-optimal solutions. These solutions show the best balance between the two goals. Then, one of these solutions was selected as the optimum stacking sequence and aluminum plate thickness. The optimal stacking sequence is shown in Figure 5.10.

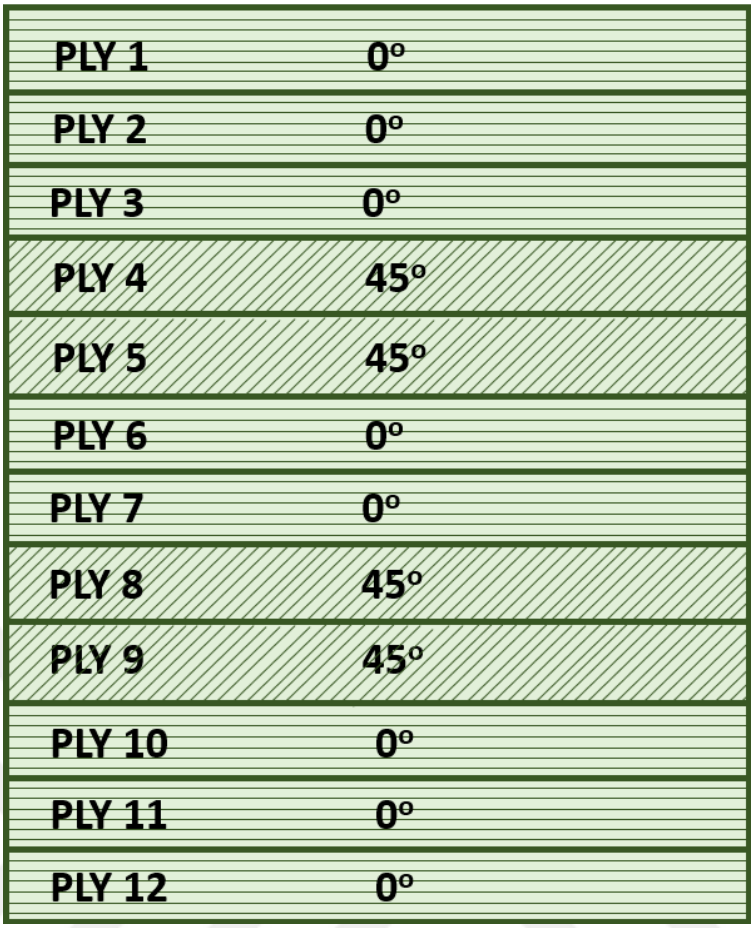

Figure 5.10 : Optimum stacking sequence.

Furthermore, the predicted, actual, and error values of mass, von Mises stress, and Hashin index corresponding to the optimal point are presented in Table 5.3.

Table 5.3 : Optimum stacking sequence properties.

Optimum Stacking Sequence	$[0^o\ 0^o\ 0^o\ 45^o\ 45^o\ 0^o]_s$
Optimum Ply Number	12
Optimum Aluminum Plate Thickness	0.8 mm
Optimum von Mises Stress (Predicted)	203.769 Mpa
Optimum von Mises Stress (True)	272.563 Mpa
The von Mises Stress Metamodel Error	25.20%
Optimum Total Mass (Predicted)	0.108 kg
Optimum Total Mass (True)	0.108 kg
The Mass Metamodel Error	0.03%
Optimum Lamina Hashin Index (Predicted)	0.819
Optimum Lamina Hashin Index (True)	0.818
The Hashin Index Metamodel Error	0.09%

6. RESULT AND DISCUSSION

This thesis presents a comprehensive study on the multi-objective optimization of adhesively bonded metal and composite plates under out-of-plane impact loading. The main objective was to develop lightweight and impact-resistant structural designs by minimizing mass and critical stress values, while satisfying damage and safety constraints.

The study was divided into three main stages. In the first stage, the thickness of an AL7050 aluminum plate was optimized under impact loading. LS-DYNA simulations were used to collect data, and Kriging surrogate models were developed to estimate von Mises stress and total mass. A genetic algorithm implemented in MATLAB was then used to find optimal solutions. The results showed that metamodel-based optimization can effectively reduce computational cost while identifying designs that meet strength and weight requirements.

In the second stage, optimization of an AS4 8552 woven composite plate was performed. The aim was to maximize absorbed energy and minimize mass. A large number of stacking sequence combinations were generated using Python scripts, and LS-DYNA was used to perform impact simulations. The Hashin failure index was used as a constraint to ensure structural safety. Gaussian Process Regression (GPR) models were trained to predict absorbed energy and mass, enabling efficient optimization without excessive simulations. The results highlighted that symmetric stacking sequences with well-balanced ply angles provided the best trade-off between energy absorption and lightweight design.

The final stage focused on the optimization of an adhesively bonded hybrid structure composed of the previously studied aluminum and composite plates. The design variables included aluminum thickness, stacking sequence, and the number of plies. Objective functions were defined as minimizing total mass and von Mises stress on the aluminum plate, while the Hashin index acted as a constraint. Cohesive zone modeling was applied to represent the adhesive layer between the plates. Surrogate models were trained using selected simulation data, and a genetic algorithm was

employed to explore the design space. The results demonstrated that the proposed method can identify optimal hybrid joint configurations that are both lightweight and structurally safe under impact conditions.

Overall, this thesis shows that combining finite element analysis with surrogate modeling and genetic algorithms offers a practical and efficient approach to optimizing adhesively bonded structures. The methods used here can support the early design phases of aerospace or automotive components, helping engineers reduce computational time while achieving performance and safety targets. Future work may include experimental validation or extension to include temperature effects, fatigue, or other failure modes.

REFERENCES

[1] **Hart-Smith, L. J.** (1974). Analysis and design of advanced composite bonded joints (NASA Contractor Report No. CR-2218). National Aeronautics and Space Administration.

[2] **Banea, M. D., & da Silva, L. F. M.** (2009). Adhesively bonded joints in composite materials: An overview. Proceedings of the Institution of Mechanical Engineers, Part L: Journal of Materials: Design and Applications, 223(1), 1–18. https://doi.org/10.1243/14644207JMDA219

[3] **da Silva, L. F. M., Öchsner, A., & Adams, R. D. (Eds.).** (2011). Handbook of adhesion technology. Springer.

[4] **Abbott, R.** (2019). Analysis and design of composite and metallic flight vehicle structures (3rd ed.). *https://www.abbottaerospace.com/aa-sb-001/12-joints/12-1-adhesive-joints/12-1-1-introduction/*

[5] **Ozel, A., Yazici, B., Akpinar, S., Demir Aydin, M., & Temiz, S.** (2014). A study on the strength of adhesively bonded joints with different adherends. International Journal of Adhesion and Adhesives, 54, 1–10.

[6] **Marques, E. A. S., & da Silva, L. F. M.** (2008). Bonded patches. The Journal of Adhesion, 84(10), 915–934. https://doi.org/10.1080/00218460802505275

[7] **Machado, J. J. M., Marques, E. A. S., & da Silva, L. F. M.** (2017). Adhesives and adhesive joints under impact loadings: An overview. The Journal of Adhesion. https://doi.org/10.1080/00218464.2017.1282349

[8] **He, B., & Pan, Y.** (2021). Failure mechanism of single lap, adhesively bonded composite–titanium joints subjected to solid projectile impact. Journal of Adhesion Science and Technology. https://doi.org/10.1080/01694243.2021.1922213

[9] **Deka, L. J., Bartus, S. D., & Vaidya, U. K.** (2006, June). Damage evolution and energy absorption of FRP plates subjected to ballistic impact using a numerical model. 9th International LS-DYNA Users' Conference, Dearborn, MI. CD-ROM proceedings.

[10] **Talagani, M. R.** (2014). Impact analysis of composite structures (Doctoral dissertation, Delft University of Technology). Retrieved from https://repository.tudelft.nl/

[11] **Amouzgar, K., Rashid, A., & Stromberg, N.** (2013). Multi-objective optimization of a disc brake system by using SPEA2 and RBFN. Proceedings of the ASME 2013 International Design Engineering Technical Conferences & Computers and Information in Engineering Conference (IDETC/CIE), Portland, Oregon, USA. American Society of Mechanical Engineers (ASME). https://doi.org/10.1115/DETC2013-12537

[12] **Marklund, P. O., & Nilsson, L.** (2000). Optimization of a car body component subjected to side impact. Structural and Multidisciplinary Optimization, Linköping University, Department of Mechanical Engineering.

[13] **Yin, H., Xiao, Y., Wen, G., Qing, Q., & Deng, Y.** (2015). Multiobjective optimization for foam-filled multi-cell thin-walled structures under lateral impact. Thin-Walled Structures, 94, 51–60. https://doi.org/10.1016/j.tws.2015.03.038

[14] **Saraswat, M., & Sharma, A. K.** (2013). Genetic algorithm for optimization using MATLAB. International Journal of Advanced Research in Computer Science, 4(3), 154–157

[15] **Dong, S., Sheldon, A., Carney, K., & Honda R&D America, Inc.** (2018, June). Modeling of carbon-fiber-reinforced polymer (CFRP) composites in LS-DYNA® with optimization of material and failure parameters in LS-OPT®. 15th International LS-DYNA® Users Conference.

[16] **Albanesi, A., Roman, N. D., Bre, F., & Fachinotti, V.** (2018). A metamodel-based optimization approach to reduce the weight of composite laminated wind turbine blades. *Composite Structures, 194*, 541–552. https://doi.org/10.1016/j.compstruct.2018.04.015

[17] **Smiths Metal Centres.** (2023). 7050 Aluminium Technical Datasheet. https://www.smithmetal.com/pdf/aluminium/7xxx/7050.pdf

[18] **Kleijnen, J. P. C.** (2007). Kriging metamodeling in simulation: A review. European Journal of Operational Research, 192(3), 707-716. https://doi.org/10.1016/j.ejor.2007.10.013

[19] **Ata, T. T., & Coker, D.** (2019). 3D simulation of dynamic delamination in curved composite laminates. Procedia Structural Integrity, 21, 130–137. https://doi.org/10.1016/j.prostr.2019.12.094

[20] **Bartholomeusz, R. A., & Rhead, A. T.** (2003). Design of composite pressure vessels using the Tsai-Wu failure criterion.

[21] **LSTC.** (2023). LS-DYNA Keyword User's Manual Volume II: Material Models (Version R13.0). Livermore Software Technology Corporation.

[22] **Hashin, Z.** (1980). Failure criteria for unidirectional fiber composites. Journal of Applied Mechanics, 47(2), 329–334.https://doi.org/10.1115/1.3153664

[23] **3M** FM300 Adhesive Film Technical Data Sheet. 3M.

[24] **Ahmetcan Alkoç, A. B. H. Y., Mete Bakır, Adem Can Uşak** (2025). "Study on the durability of film adhesively bonded composite joints subjected to environmental aging." Reinforced Plastics and Composites.

[25] **Cytec Solvay Group** (2018). *FM® 300 Film Adhesive Technical Data Sheet*

[26] **Jokinen, J., Wallin, M., & Kanerva, M.** (2020). The influence of the number of adhesive plies (FM 300-2) on fracture properties. *ICAS 2020*. https://www.icas.org/icas_archive/ICAS2020/data/papers/ICAS2020_0295_paper.pdf

[27] **Breiman, L.** (1996). Bagging predictors. *Machine Learning*, *24*(2), 123–140. https://doi.org/10.1007/BF00058655

[28] **Draper, N. R., & Smith, H.** (1998). *Applied regression analysis* (3rd ed.). Wiley.

CURRICULUM VITAE

Name Surname **:** Ezgi BAKIR

EDUCATION **:**

- **B.Sc** **:** 2022, Istanbul Technical University, Faculty of Aeronautics and Astronautics, Department of Aeronautical Engineering

PROFESSIONAL EXPERIENCE AND AWARDS:

- Since 2022, Working as a structural analysis engineer at Turkish Aerospace Industries

PUBLICATIONS, PRESENTATIONS AND PATENTS ON THE THESIS:

- **BAKIR E.,** Vedat Ziya Doğan, Oğuzcan İnal, Zahit Mecitoğlu. 2025: " Optimization of Aluminum Plate Under Impact Loading " 4th International Graduate Research Symposium (IGRS'25), İstanbul, Türkiye, 12-14 May, 2025.