Summer 8-2021

Shape Optimization of a Sandwich Composite Plate with Low-Velocity Impact Loading and Effect of Finite Difference Sensitivity Analysis

Utsav Shah
shahu@my.erau.edu

Follow this and additional works at: <https://commons.erau.edu/edt>

Part of the [Structures and Materials Commons](#)

Scholarly Commons Citation

Shah, Utsav, "Shape Optimization of a Sandwich Composite Plate with Low-Velocity Impact Loading and Effect of Finite Difference Sensitivity Analysis" (2021). *PhD Dissertations and Master's Theses*. 604.
<https://commons.erau.edu/edt/604>

This Thesis - Open Access is brought to you for free and open access by Scholarly Commons. It has been accepted for inclusion in PhD Dissertations and Master's Theses by an authorized administrator of Scholarly Commons. For more information, please contact commons@erau.edu.

SHAPE OPTIMIZATION OF A SANDWICH COMPOSITE PLATE WITH LOW-
VELOCITY IMPACT LOADING AND EFFECT OF FINITE DIFFERENCE
SENSITIVITY ANALYSIS

By

Utsav B Shah

A Thesis Submitted to the Faculty of Embry-Riddle Aeronautical University
In Partial Fulfillment of the Requirements for the Degree of
Master of Science in Aerospace Engineering

August 2021

Embry-Riddle Aeronautical University

Daytona Beach, Florida

SHAPE OPTIMIZATION OF A SANDWICH COMPOSITE PLATE WITH LOW-VELOCITY IMPACT LOADING AND EFFECT OF FINITE DIFFERENCE SENSITIVITY ANALYSIS

By

Utsav B Shah

This Thesis was prepared under the direction of the candidate's Thesis Committee Chair, Dr. Mandar Kulkarni, Department of Aerospace Engineering, and has been approved by the members of the Thesis Committee. It was submitted to the Office of the Senior Vice President for Academic Affairs and Provost, and was accepted in the partial fulfillment of the requirements for the Degree of Master of Science in Aerospace Engineering.

THESIS COMMITTEE

Chairman, Dr. Mandar Kulkarni

Member, Dr. Ali Tamijani

Member, Dr. Alberto Mello

Graduate Program Coordinator,
Dr. DaeWon Kim

8/6/2021

Date

Dean of the College of Engineering,
Dr. Maj Mirmirani

08/06/2021

Date

Associate Provost of Academic Support,
Dr. Christopher Grant

8/9/21

Date

ACKNOWLEDGEMENTS

I would like to give a special thanks to Dr. Mandar Kulkarni for helping me out on this difficult and fast-paced journey. He has helped me to grow my critical thinking skills by asking difficult questions that can help me clear my doubts and can focus on the important concepts of this research.

I would also like to thank Dr. Namilae for providing me access to the labs in Micaplex for sandwich composite plate fabrication and using various machines and equipment. Dr. Sypeck of Instruments lab also taught me the UTM and LVI testing machines for weeks and was readily available when needed to conduct multiple experiments.

Another thank you goes to Dr. Kulkarni's Ph.D. student, Naresh Ahuja who helped me with ANSYS and FEMAP software for modeling and was also available when needed to clear any doubts during this research.

Finally, I wish to thank my mom, dad and brother for their love, encouragement, and support, without whom I would have never enjoyed the opportunities provided. I am thankful to my friends for supporting and helping me out for the past 2 semesters and understanding the importance of this research to me.

This project was sponsored by National Aeronautics & Space Administration through the University of Central Florida's NASA Florida Space Grant Consortium and Space Florida.

ABSTRACT

The structural design of spacesuits is essential in an advancing future of both Lunar and Martian space exploration. A typical spacesuit is made of sandwich composite material and designed to withstand various pressure and loading conditions while also considering the safety and comfort of the astronauts. One of the critical load cases in spacesuit design is a low-velocity impact (LVI) which may occur due to tool drop and other similar scenarios. The objectives of this work were (a) to create a finite element (FE) model of a component of a spacesuit, (b) validation of the FE model through experiments, (c) creating an optimization framework to design the spacesuit component, and (d) investigate the effect of finite difference step size on the final optimized design. A FE model of a plate was created to represent a part of the spacesuit's hard upper torso (HUT), which is made of a sandwich structure with S2 glass fiber composite (outer layers) and carbon fiber composite (core) materials. The FE model was used to simulate the nonlinear LVI response of the plate using MSC Nastran and ANSYS software. LVI experiments were performed in the materials lab using the Instron Impact Test instrument. The FE models were validated against the displacement and contact force history obtained for two impact velocities. Further, the sandwich plate was optimized for an impact load case with sizing variables (thickness and ply orientation) and shape variables (linear, quadratic, sinusoidal and Hicks-Henne bump shape function). The objective was to minimize weight while being subject to displacement or stress constraints. During the optimization process, it was found that the change in fiber orientation and the thickness of plies, reduced the deformation by 37.63% and increased the strength-to-weight ratio of the coupon sample, while indirectly decreasing the

maximum stress. The effect of the finite difference step size variation on the shape optimization of the plate was studied. It was found that choosing an appropriate step size is not intuitive. After varying the step size over four orders of magnitude, the best step size led to a design with a deformation reduction of 30% and a total weight reduction of 34% compared to the initial design.

TABLE OF CONTENTS

ACKNOWLEDGEMENTS.....	iii
ABSTRACT.....	iv
LIST OF FIGURES.....	viii
LIST OF TABLES.....	xii
NOMENCLATURE.....	xiii
1. Introduction.....	1
1.1. Background.....	2
1.2. Motivation and Scope of Thesis	3
2. Review of the Relevant Literature.....	5
2.1. Spacesuit Requirements and Materials.....	5
2.1.1. Space Material.....	5
2.1.2. Spacesuit Requirement and Criteria.....	6
2.2. Structural Optimization.....	8
3. Experimental Approach.....	12
3.1. Fabrication of Sandwich Composite Plate.....	12
3.2. Ultimate Tensile Testing.....	15
3.2.1. Experimental Analysis.....	19
3.3. Low Velocity Impact Experiment.....	23
3.3.1. Analysis.....	28
4. Numerical Simulation.....	32
4.1. Ultimate Tensile Machine Testing Simulation.....	34
4.2. Low Velocity Impact Finite Element Simulation.....	36
4.2.1. Modal Analysis.....	36
4.2.2. Linear Transient Analysis.....	38
4.2.3. Non-linear Transient Analysis.....	42
5. Structural Optimization.....	47
5.1. Sizing Optimization.....	47
5.2. Shape Optimization.....	53
5.3. Shape Sensitivity.....	65
5.3.1. Identifying Correct Finite Difference Step Size.....	66
6. Conclusions and Recommendations.....	69
6.1. Conclusion.....	69
6.2. Recommendations and Future Works.....	71

REFERENCES.....	73
APPENDIX A.....	75

LIST OF FIGURES

Figure	Page
1.1 3D Printed Spacesuit Component (Graziosi, 2016).....	1
2.1 Low Velocity Impact Test samples for S glass (Ross, 2014).....	7
2.2 Low Velocity Impact Test samples for IM-10 (Ross, 2014).....	8
2.3 Flow process of structural optimization (MSC Nastran user's guide design sensitivity and optimization, 2017)	10
3.1 Sandwich composite layup.....	13
3.2 a) Bottom layup b) Final layup.....	13
3.3 Cured sandwich composite laminate plate.....	14
3.4 a) LVI testing specimen b) UTM testing specimen.....	15
3.5 AL 6061 gripping tab for one sample.....	16
3.6 Air sand blaster.....	16
3.7 MeltWeld metal bonding epoxy.....	17
3.8 Cured specimen with tabs for UTM testing.....	17
3.9 a) Tinius Olsen 150ST Electromechanical Universal Testing Machine b) Hydraulic gripping unit.....	18
3.10 Position of LVDT Extensometer on specimen.....	19
3.11 Stress vs strain plot for sample 1 using 2D line.....	20
3.12 Stress vs strain plot for sample 2 using 2D line.....	20
3.13 Sample 3 after UTM testing.....	21
3.14 Stress vs strain plot for sample 3 using 2D line.....	22
3.15 Cured specimen after UTM testing.....	22

Figure	Page
3.16 Components of the drop tower.....	24
3.17 a) INSTRON 9250 HV Impact Test instrument, b) Hemispherical indenter.....	24
3.18 INSTRON 9250 pneumatic clamp fixture.....	25
3.19 Set Impactor baseline and fix rebound brakes.....	26
3.20 LVI experiment at 2 m/s with visible fiber crack a) Top face b) Bottom face...	26
3.21 LVI experiment at 2.5 m/s with visible fiber crack a) Top face b) Bottom face.....	27
3.22 LVI experiment at 2 m/s, 3 m/s and 3.5 m/s a) Top face b) Bottom face.....	27
3.23 Deflection vs time for 2 different composite.....	28
3.24 Impact load vs time plot for two different composites at 2 m/s.....	29
3.25 Impact energy vs time plot for two different composites at 2 m/s.....	29
3.26 Force vs time plot for different speeds on sandwich composite panel.....	30
3.27 Deflection vs time at different speeds for sandwich composite material.....	31
4.1 Maximum stress in ply of symmetry or ply 8/9.....	35
4.2 Stress vs strain plot for UTM simulation data.....	35
4.3 Mode shape 1 at 1560.8Hz.....	37
4.4 Mode shape 2 at 2290.5Hz.....	37
4.5 Mode shape 3 at 3680.8Hz.....	38
4.6 Plate with nodal forces applied to the center elements.....	39
4.7 Maximum Displacement contour plot for T800s carbon fiber.....	39
4.8 Displacement vs time plot for T800s Carbon Fiber ten layers.....	40
4.9 Displacement vs time for sandwich composite plate.....	41
4.10 Contour plot of maximum displacement for sandwich composite plate.....	42

Figure	Page
4.11 LVI model of SOL 400 (Zhennan, 2020)	43
4.12 Displacement vs time for same load case (CFRP)	44
4.13 Displacement vs time plot comparing SOL 400, SOL 112 and LVI experiment.....	45
4.14 Displacement vs time plot comparing two initial velocity simulations and experimental data.....	46
5.1 Objective history of SOL 200.....	50
5.2 Normalized constraint history of SOL 200.....	50
5.3 Deformation comparison of SOL 200.....	51
5.4 Ply thickness history of SOL 200.....	51
5.5 Ply angle history of SOL 200.....	52
5.6 Objective history for length-width optimization.....	55
5.7 Constraint history for length-width optimization.....	56
5.8 Deformation comparison of length-width optimization to original data.....	56
5.9 Quadratic shape of the plate using MATLAB.....	57
5.10 Constraint history for quadratic function shape optimization.....	58
5.11 Objective history for quadratic function shape optimization.....	58
5.12 Deformation comparison of quadratic shape.....	59
5.13 Sine shape of the plate using MATLAB.....	60
5.14 Optimized Sine shape in FEMAP.....	60
5.15 Deformation comparison of sine shape.....	61
5.16 Hicks-Henne Bump shape of the plate using MATLAB.....	62
5.17 Optimized Hicks-Henne Bump shape in FEMAP.....	62

Figure	Page
5.18 Objective history for Hicks-Henne Bump Function shape optimization.....	63
5.19 Constraint history for Hicks-henne Bump Function shape optimization.....	64
5.20 Deformation comparison of Hicks-Henne Bump Function.....	64

LIST OF TABLES

Table	Page
3.1 Specimen dimension for horizon software input.....	18
3.2 Mechanical Properties of Sandwich Composite Plate through UTM Testing.....	21
4.1 Mechanical properties of T800s carbon/3900 Epoxy and S2 glass/381 Epoxy lamina under 22°C.....	34
4.2 Geometric properties of sandwich composite plate/coupon.....	36
4.3 Modal frequency of Sandwich Composite Plate.....	38
4.4 Stacking sequence of composite sandwich plate.....	41
4.5 LVI model boundary constraints (Zhennan, 2020)	42
4.6 Error % comparison between LVI Experiment, SOL 112 and SOL 400.....	45
5.1 Bulk Data Entry Nastran card (Zhennan, 2020)	49
5.2 Results of sizing optimization using SOL 200 for quarter plate.....	52
5.3 Results of shape optimization using SOL 200 for quarter plate.....	55
5.4 Results of shape optimization using hicks-henne bump function for quarter plate.....	65
5.5 Optimized displacement for different step size scale comparing two different sensitivity analysis method.....	68
5.6 Optimized total weight for different step size scale comparing two different sensitivity analysis method.....	68

NOMENCLATURE

ACP	Ansys Composite PrePost
ASTM	American Society for Testing and Material
CFRP	Carbon Fiber Reinforced Polymer
CSA	Continuum Sensitivity Analysis
DCM	Display Command Module
E	Young's Modulus
EMU	Extravehicular Mobility Unit
G_{12}	Shear Modulus
GFRP	Glass Fiber Reinforced Polymer
HEO	Highly inclined Elliptical Orbit
HUT	Hard Upper Torso
LVDT	Linear Variable Differential Transformer
LVI	Low-Velocity Impact
MOGA	Multi-Objective Genetic Algorithm
NASA	National Aeronautics and Space Administration
PLSS	Primary Life Support System
S	Shear Stress
UTM	Ultimate Tensile Machine
X	Stress in Longitudinal direction
Y	Stress in Transverse direction
ν	Poisson's ratio
ρ	Density

1. Introduction

Spacesuits play a vital role in space exploration while providing astronauts a safety barrier from the harsh environment of the Moon or Mars. Designing the spacesuit for each mission and testing them in simulation takes a large amount of time. To alleviate this, NASA started 3D printing HUT prototypes, shown in Figure 1.1, to verify the life cycle of joint design, using state-of-the-art 3D scanning devices. These 3D prints are developed in reference to the specific target or person, which can take multiple iterations to design and optimize accurately. To overcome this, a fast and accurate spacesuit design is proposed in this research by creating a high-fidelity model that can accommodate multiple design variables and simulate the response of the spacesuit's structure due to various loading conditions. This model can then be used to optimize the external shape and internal reinforcements of the spacesuit to minimize the weight of the overall assembly.

Figure 1.1 3D Printed Spacesuit Component (Graziosi, 2016)

The development of the model will be done using the Hard Upper Torso as a specimen, which will be subjected to low velocity impacts to simulate the forces applied during manufacturing and space exploration. This can be done by identifying static and dynamic design load cases for the structural analysis of spacesuits. The test specimen will later use Continuum Sensitivity Analysis (CSA) to generate accurate sensitivities of stress and displacement, especially with respect to various shape configurations. This proposed scheme can be used to design multiple components of NASA's HEO missions such as Human Landing Systems, Lunar, and Martian vehicles.

1.1 Background

Composites have become an integral part of today's manufacturing industry. The need for composites has increased due to their high strength to weight ratio when compared to metal materials of the same strength. The growth of the astronautics industry requires a strong and stable material, which does not degrade in the extreme temperatures found in future missions. This requirement calls for a sandwich composite material consisting of two or more different material properties that combine to form a high-performance material (Rajak et. al., 2019).

It is known that the sandwich composites are mostly anisotropic materials in nature, which means structural strength properties change with the application of the load with change in fiber orientation. This makes it possible to develop component materials in the desired direction depending on the requirements (Krzyzak et. al., 2016). Sandwich composite panels are used due to their load transfer capabilities through bending and shear. The exterior surface transfers load caused by bending, whereas the core transfers loads caused by shear. Due to this, the sandwich composites are used in many space

vehicles like the SLS casing, Orion head shield, Artemis XEMU spacesuit, Mars 2020 rover aeroshell, Mars helicopter, etc (Francis, 2020).

1.2 Motivation and Scope of Thesis

The low-velocity impact often occurs on spacesuits during fabrication, space-walk missions, lunar missions, etc. This impact can cause serious damage to the spacesuit if hit repeatedly and can result in catastrophic failure/accident. Thus, this research aims to create a numerical model that can be used for various materials in the future and can be readily optimized to develop a structure that is light in weight but maintains the structural strength while reducing displacement. A sub-goal is to inspect the damage tolerance using LVI transient load, by comparing the two different plate coupons: CFRP composite and Sandwich composite.

Chapter 2 discusses the literature review of the materials and design requirements of the spacesuits, and how they changed over time to make them lighter and stronger. The development of structural optimization through various methods and algorithms is discussed along with the selected process for the current research.

In chapter 3, two different experiments are performed and discussed. First, the fabrication of the sandwich composite plate for testing is discussed. Secondly, experiment 1: Ultimate Tensile Machine testing is discussed where the ultimate tensile strength of the fabricated sandwich composite is measured using ASTM standards. Finally, experiment 2: Low-Velocity Impact testing is performed at various initial velocities to understand the plate's overall damage and energy absorption. This experiment is then compared with the CFRP composite plate experiment performed by a previous researcher.

Chapter 4 comprises the numerical simulation of both the experiments, which are then verified using the experimental data. UTM simulation is computed in ANSYS 2020 which outputs the stress and strain data at various applied loads. MSC. Nastran SOL 400 is used to compute the LVI simulation which is then used by SOL 112 to output the contact force, energy and deflection vs time data.

Once the simulations are verified, the LVI model is used for the structural optimization in chapter 5. First, size optimization aims to minimize the weight of the plate while maintaining the structural strength and satisfying the required constraints. Secondly, shape optimization is computed using the shape vectors to optimize the new plate using displacement as the constraint. After shape optimization is completed, the feasible shape vector providing optimum design is selected. Sensitivity analysis is computed using two different approximation methods: forward difference and central difference, for two optimizing methods: Sequential Linear Programming and Sequential Quadratic Programming.

Finally, the conclusion of the research is discussed in the last chapter. Some recommendations and future work are also provided to continue the research and use the state-of-the-art spacesuit materials for experiments and optimization.

2. Review of the Relevant Literature

National Aeronautics and Space Administration (NASA) has been designing spacesuits for many years. They act as protection for the astronauts or crew in different scenarios like launch, entry, and abort. Hard Upper Torso (HUT) is the main component of the spacesuit as it comprises of the Display and Control Module (DCM) and the Primary Life Support Subsystem (PLSS) that helps the crew to stay alive (Dunbar, B et. al., 2013).

2.1 Spacesuit Requirements and Materials

2.1.1 Space Material

Research conducted by Ferl et.al (2006) states that the spacesuit must be worn unpressurized and that it can remain operable for 96 hours, which allows the crew to complete missions in 0-G and be comfortable (Dunbar, B et. al., 2013). The spacesuit materials are layered with a water-cooled Nylon undergarment, a multi-layered pressure suit with an internal layer of lightweight nylon with fabric vents, a middle layer with neoprene-coated nylon to hold pressure, and an outer nylon layer to restrain the pressurized layers beneath. This is then followed by five layers of Aluminized Mylar interwoven with four layers of Dacron for heat protection, two layers of Kapton for additional heat protection, a layer of Teflon-coated cloth (non-flammable) for protection from scrapes and, finally, a layer of White Teflon-coated cloth (non-flammable).

HUT is made up of different materials and thicknesses to accommodate different spacesuits over time. For example, fiberglass reinforced epoxy material was used for the Shuttle EMU HUT and the ZPS Mark III Brief Transition at 90 mil (2.286 mm) and 60 mil (1.524 mm) thickness respectively, aluminum was used for the ZPS Mark III at 80

mil (2.032 mm) thickness and double layer denier Dacron with urethane-coated nylon was used for the Rear Entry I-Suit SUT at 33 mil (0.8382 mm) thickness (Ferl, J. et. al., 2006). To achieve the objective of making the spacesuit lighter, new composite material must be introduced to manufacture HUTs based on the requirements and criteria which would later be optimized.

According to research conducted by NASA and ILC Dover, the HUT material is chosen based on its impact resistance, strength, and mass requirements. The Z-2 HUT was developed with a hybrid composite material that has layers of S2 glass/epoxy resin prepreg that sandwich a thick IM10/epoxy resin prepreg core. This HUT composition provides a great balance of the requirements with a total weight of 17 lbs. for the HUT shell. The research also mentions that the Z-2 hatch was changed to an entirely composite material build like that of the HUT. But during the demolding, the hatch was damaged due to the large ply that was not peeled during installation of prepreg and was later changed back to the original material of hybrid aluminum and composite assembly (Graziosi, 2016).

2.1.2 Spacesuit Requirement and Criteria

To accommodate all the important and complex systems, the HUT must be composed of a strong tensile strength material while able to resist the pressure exerted by the external forces. The HUT has a structural integrity requirement of 3.75 psi nominal operation pressure, with an 8-psi static pressure to achieve a factor of safety of 2 (Ferl, J. et. al., 2006). They also stated that the Shuttle EMU HUT is made to withstand 15.8 psi pressure, with a safety factor of 2.

A research paper by Ross et. al. (2014) mentioned another requirement of withstanding all the possible impact loading conditions that arises during the lunar missions. In order to achieve all the above-mentioned requirements, they developed a finite element model of Z-2 space suit which was analyzed in LS-DYNA for dynamic impact analysis and Nastran for static analysis. The static and impact analysis were conducted using the low velocity impact experiments at different energy levels for S2 glass and IM10 carbon fiber shown in Figure 2.1 and Figure 2.2, following the ASTM standards. This data was compared to the properties given by MIL-HDBK-17. It was also mentioned that based on the LS-DYNA simulation results, parametric studies were conducted to determine the stacking sequence and sizing of the composite laminas to minimize the damage and deformation (Ross, 2014). Due to lack of research available on optimization of the composite material using Nastran rather than LS-DYNA, various optimizing methods are suggested in this research paper and are discussed in detail.

Figure 2.1 Low Velocity Impact Test samples for S glass (Ross, 2014)

Figure 2.2 Low Velocity Impact Test samples for IM-10 (Ross, 2014)

2.2 Structural Optimization

It is presumed that the metallic materials are isotropic in nature when modeled (i.e. the properties: strength and stiffness are very close in all directions), whereas composite materials, for example, CFRP and GFRP are anisotropic, and their properties vary in all directions. These composite materials are layered with different stacking sequences and fiber orientations depending on the type of load applied to the laminate/structure (Mian, 2013). This provides the structure with a reduced weight while increasing the strength of the component. Thus, to optimize the coupon sample, weight and shape optimization is to be conducted by ply thickness or stacking sequence, fiber orientation, and geometric shape.

Structural optimization can be conducted using various optimization theories and algorithms using some of the classic methods like size, shape, and topology. Most of the optimization these days tend to be complex in nature and thus requires commonly used

software like MSC Nastran, Genesis, OptiStruct and OptiSLang (to a limited extent). In this research, MSC Nastran is being used to conduct the size and shape optimization of the sandwich composite plate.

In 1960, Schmit first brought the method of structural optimization mathematically, where the one or more design variables are defined, and the objective is to minimize or maximize the response of the surface, while satisfying the design constraint (Schmit, 1960). Later, the optimization was integrated by the FEM for some basic analysis. According to the MSC Nastran user guide design sensitivity and optimization, the design task can be expressed in a quantitative form of a linear optimization statement as follows: Minimize/Maximize X,

$$F(X) \text{ Objective function} \quad (1)$$

$$g_j(X) \leq 0; j = 1, \dots, n_g \text{ Inequality constraint} \quad (2)$$

$$h_k(X) = 0; k = 1, \dots, n_h \text{ Equality constraint} \quad (3)$$

$$x_i^L \leq x_i \leq x_i^U; i = 1, \dots, n \text{ Side constraint} \quad (4)$$

$$X = \{x_1, x_2, \dots, x_n\} \text{ Design variables} \quad (5)$$

The first optimization algorithm used by MSC Nastran is a nonlinear programming method (NPL), which uses gradient based optimization and is comprised of a direct and an approximation method. This method transforms design variables into an explicit function of variables, which can be controlled by the NPL and later conducts shape and size optimization. Another method is sequential quadratic programming (SQP), where the initial design point can be in feasible or infeasible region. SQP can handle equality constraint efficiently (Choi et al., 2016). According to the MSC Nastran Design Sensitivity and Optimization User's Guide, when using Nastran SOL 200, the program

automatically uses MSCADS for size optimization and IPOPT for topology optimization.

Figure 2.3 shows the procedures of the methodology being used, which helps to

summarize all the steps take to finish the optimization in MSC. Nastran.

Figure 2.3 Flow process of structural optimization (MSC Nastran user's guide design sensitivity and optimization, 2017)

Design sensitivity analysis shows sensitivities/change in response of design model in reference to the change in design parameter for structural optimization. Sensitivity analysis is performed automatically before the optimization as it requires the response function in form of gradient of objective and constraint function. This helps avoid the high cost of repetition during optimization and can also perform the parametric study using the response functions. The optimization can use the direct and adjoint sensitivity techniques, finite difference analysis scheme, as well as finite element methods.

In this report, optimization is first conducted for the size of the plate. The total thickness of the plate and fiber orientation of each ply is considered as the design variables along with weight minimization as objective and deflection as constraint. Once

optimized, shape optimization take place, where force displacement method is used to generate shape vectors using initial shape basis vector.

3. Experimental Approach

The goal of this section is to conduct experiments like ultimate tensile machine testing (UTM) and low velocity impact testing (LVI). UTM is used to understand the behavior of the sandwich composite under tremendous load, whereas LVI is used to understand how small handheld tools, such as a hammer, can cause changes to the affected area and how much energy is absorbed when it impacts the HUT. This section will discuss the fabrication of the sandwich composite plate, followed by UTM and LVI testing.

3.1 Fabrication of Sandwich Composite Plate

The sandwich composite plate is fabricated using two different materials: S2 glass CYCOM 381 unidirectional epoxy prepreg from Solvay and T800s UD carbon fiber 3900 series epoxy prepreg from HEATCON Composite. The data sheet for both the materials are provided in Appendix B. Both the prepreps were kept under 10° F until the start of the fabrication process. The sandwich composite plate was fabricated using the S2 glass as both top and bottom skin, while the T800s carbon fiber was used as a core. The core has ten layers of carbon fiber, whereas the top and bottom skin has three layers of S2 glass, all in a symmetric 0/90 fiber orientation as shown in Figure 3.1. Each of the layers were trimmed to 250 mm x 250 mm (9.84 in x 9.84 in) dimension using the electric cutter.

The materials were then placed as described above and were fabricated in a Wabash Compression Press (Wabash MPI). An aluminum plate was used as a bottom platform, with the releasing agent applied for easy removal after the curing process. Next, the stacked composite plate was placed and pressed using the rollers until the sample stopped

moving. Later, peel ply from FiberGlast corporation (Brookville, OH) was placed for easy removal, followed by breathable fabric, which can be seen in Figure 3.2 (a). The fabric functions as a two-fold during the curing process. By absorbing the excess epoxy through peel ply when the clamp pressure load is applied, the fabric can strengthen the laminate. Finally, aluminum foil was used to cover the fabric and the upper platen was lowered to apply uniform load to the entire composite stacked laminate. The final laminate curing layup is shown in Figure 3.2 (b).

Figure 3.1 Sandwich composite layup

a)

b)

Figure 3.2 a) Bottom layup b) Final layup

The composite stacked laminate was cured using the Wabash Compression Press (Wabash MPI) under a continuous loading of 2 tons (19,613 N) at 160°C (320°F) for 140 minutes. This curing temperature and time was selected by trial-and-run method as the curing temperature and time for both the materials are different: 177°C (350°F) for 120 minutes for carbon fiber and 126.67°C (260°F) for 100 minutes for S2 glass. Once the curing time was completed, the laminate was cooled with hydraulics and the air-cooling system until it reached room temperature, and the cured laminate plate shown in Figure 3.3 was removed. The laminate plate was later trimmed to a 250 mm x 250 mm (9.84 in x 9.84 in) square plate, removing excess hardened epoxy. This dimension was used to obtain two samples of 101.6 mm x 152.4 mm (4 in x 6 in) for LVI testing and three samples of 250 mm x 25 mm (9.84 in x 0.98 in) for tensile testing, as required by the ASTM standards, and are shown in Figure 3.4 a) and b) respectively.

Figure 3.3 Cured sandwich composite laminate plate

Figure 3.4 a) LVI testing specimen b) UTM testing specimen

3.2 Ultimate Tensile Testing

Tensile testing is important to understand the mechanical behavior of a material undergoing continuously applied axial loads. This test will determine only the ultimate tensile strength of the sandwich composite plate as, behaving as a brittle material, it will exhibit no yielding before fracture. A total of three samples, with dimensions of 250 mm x 25 mm, were tested. These samples were cut from the 250 mm x 250 mm fabricated plate.

To conduct this experiment, gripping tabs must be fabricated such that it can be clamped at 27.58 MPa (4000 psi) of pressure. The AL 6061 material was chosen for the tabs as its easily available and recommended for such kind of testing. The gripping tabs shown in Figure 3.5 were cut to the dimensions of 85 mm x 25 mm with a thickness of 3 mm and a beveled edge at 45°, in accordance with the UTM ASTM standards. Once the tabs were prepared, the composite panels were marked to obtain the same lengths as the

tabs, and were sanded using the central pneumatic air sand blaster at 861.84 kPa (125 psi) pressure of air shown in Figure 3.6. This is done to roughen both the surfaces so that they can be glued using the high strength two-part metal bonding epoxy, manufactured by MeltWeld, shown in Figure 3.7. The final cured specimen for UTM testing is shown in Figure 3.8.

Figure 3.5 AL 6061 gripping tab for one sample

Figure 3.6 Air sand blaster

Figure 3.7 MeltWeld metal bonding epoxy

Figure 3.8 Cured specimen with tabs for UTM testing

The Tinius Olsen 150ST Electromechanical Universal Testing Machine, shown in Figure 3.9 a), is used for this experiment with loadcell accuracy of 0.2 % at 0.2 – 100 % capacity. The specimen is first inserted into the bottom flat grip face and clamped using the MTS hydraulic gripping unit at 27.58 MPa (4000 psi) pressure, shown in Figure 3.9 b). Next, the Tinius Olsen LVDT Extensometer is attached to the center of the specimen without touching the grips, as shown in Figure 3.10, and then the upper grip is closed. The extensometer has a calibration of 20 % at 10:1 strain gage range, and resolution up to 50 mm strain measurement. Finally, the Horizon software is used to record and plot the stress vs strain data, along with force and position. The software is customized to require

input of cross-sectional area of the specimen and the extensometer measures the required gauge length to calculate strain. Table 3.1 shows the iteration process of calculating the average area for each sample.

Table 3.1

Specimen dimension for horizon software input

Sample	Properties	Iteration					Average
		1	2	3	4	5	
1	Width (mm)	22.80	22.98	22.86	23.01	23.18	22.97
	Thickness (mm)	2.26	2.21	2.27	2.22	2.23	2.24
	Area (sq.mm)	51.53	50.79	51.89	51.08	51.69	51.40
2	Width (mm)	25.85	25.66	25.57	25.38	25.41	25.57
	Thickness (mm)	2.27	2.28	2.33	2.28	2.23	2.28
	Area (sq.mm)	58.68	58.50	59.58	57.87	56.66	58.26
3	Width (mm)	25.85	25.92	25.89	25.66	25.36	25.74
	Thickness (mm)	2.26	2.22	2.24	2.26	2.25	2.25
	Area (sq.mm)	58.42	57.54	57.99	57.99	57.06	57.80

a)

b)

Figure 3.9 a) Tinius Olsen 150ST Electromechanical Universal Testing Machine b) Hydraulic gripping unit

Figure 3.10 Position of LVDT Extensometer on specimen

3.2.1 Experimental Analysis

Once the test was completed, stress vs strain data for all three samples were plotted. During the testing, it was found that sample 1 broke in between the tabs. This is possible when the epoxy resin hardener does not cure properly, or the applied quantity was less than required to glue the metal and composite. Figure 3.11 is the plot of the stress vs strain for sample 1, and it shows the maximum stress at 1470 MPa, which occurred before the break. This resulted in a Young's modulus, E , of 96.5 GPa using linear regression.

Figure 3.11 Stress vs strain plot for sample 1 using 2D line

Figure 3.12 Stress vs strain plot for sample 2 using 2D line

Sample 2 resulted in a Young's Modulus of 77.77 GPa from the stress vs strain plot shown in Figure 3.12, but during the experiment the gripping tabs started slipping and the experiment had to be immediately stopped. Sample 3 behaved properly under the continuous load as expected, and can be seen in Figure 3.13. The UTM testing outputted a Young's modulus value of 78.894 GPa with an ultimate stress of 1570 MPa, which can be observed in Figure 3.14, the stress vs strain plot. Table 3.2 summarizes the ultimate stress and the Young's modulus for each sample, along with the total average for easy comparison. This can also be seen in Figure 3.15.

Figure 3.13 Sample 3 after UTM testing

Table 3.2

Mechanical Properties of Sandwich Composite Plate through UTM Testing

Sample	Properties	
	Ultimate Strength, σ_U (MPa)	Young's Modulus, E (GPa)
1	1470	96.503
2	1520	77.77
3	1570	78.201

Figure 3.14 Stress vs strain plot for sample 3 using 2D line

Figure 3.15 Cured specimen after UTM testing

3.3 Low Velocity Impact Experiment

To achieve the objective of making the spacesuit lighter, a new composite material must be introduced to manufacture the HUT based on the given requirements and criteria. The composite material must undergo low velocity impact testing, to understand damage caused by foreign objects in the material fibers, which reduces the strength of the plate.

A research paper by Ahmed et al (2020). discussed the impact response of composite laminates with LVI in various stacking and resins. They performed experiments on three different types of composite laminates: carbon-fiber, glass-fiber, and mixed-fiber. The impact is studied through the energy absorption, with several structural parameters, by measuring the total energy input to the composite plates and the energy absorbed by the specimen. It was also stated that developing hybrid composite laminates with new fibers enhances the impact resistance of the material. It was concluded that “increasing or decreasing the number of layers by 5 %, while keeping the total laminate thickness constant, results in an increased impact energy absorption” (Alomari, 2020, pg. 146).

LVI testing was performed using the two 101.6 mm x 152.4 mm (4 in x 6 in) composite plate. The impact drop test was performed according to the ASTM D7136 (American Society for Testing and Materials, West Conshohocken, PA) test standard using the INSTRON 9250 HV Impact Test Instrument. Figure 3.16 shows the other components included in drop tower: a loadcell of mass 6.67 kg mounted on top of indenter, a pneumatic clamp, a rebound brake and a velocity flag (this measures the initial velocity before impact) which are connected to the impulse DAQ system and a controller to output the raw data. Figure 3.17 a) shows and the INSTRON 9250 HV impact test

instrument and Figure 3.17 b) shows the hemispherical steel indenter of diameter 15.75 mm that was used to hit the sandwich composite plate.

Figure 3.16 Components of the drop tower

Figure 3.17 a) INSTRON 9250 HV Impact Test instrument, b) Hemispherical indenter

To conduct this experiment, the plate was first placed in between the pneumatic clamps shown in Figure 3.18 that fixed the edges of the plate at 85 psi shop air. The rebound brakes height was adjusted to avoid the second impact on the plate along, which was done by lowering the impactor close to the plate as shown in Figure 3.19. The velocity flag was then adjusted to measure the impactor velocity before the impact. Next, impulse DAQ system controller interface was used to input initial velocity of 2 m/s, 2.5 m/s, 3 m/s and 3.5 m/s, which would be used to adjust the maximum height of the impactor using Equation 6. Finally, the test was ran, and the raw data was collected and processed to output various plots, discussed in section 3.3.1.

$$V^2 - U^2 = 2gS \quad (6)$$

Where V is the finial velocity, U is the initial velocity (0 m/s for impact), S is the total height and g is the gravitational acceleration relative to sea level. Rearranging this equation for height, we get Equation 7.

$$S = \frac{V^2}{2g} \quad (7)$$

Figure 3.18 INSTRON 9250 pneumatic clamp fixture

Figure 3.19 Set Impactor baseline and fix rebound brakes

After the experiment was setup, both plate samples were first impacted at 2 m/s and 2.5 m/s respectively. At 2 m/s, S2 glass fiber damage was seen in Figure 3.20 a) and cracks in T800s carbon fiber can also be presumed due to the visible S2 glass fiber damage at the bottom surface in Figure 3.20 b) without any complete failure. At 2.5 m/s, S2 glass fiber damage was seen in both the direction on the top face, Figure 3.21 a) but does not have a complete penetration or failure, when seen through the bottom face in Figure 3.21 b).

Figure 3.20 LVI experiment at 2 m/s with visible fiber crack a) Top face b) Bottom face

Figure 3.21 LVI experiment at 2.5 m/s with visible fiber crack a) Top face b) Bottom face

Once the small bump was visible at 2.5 m/s, it was decided to continue the experiment with 3 m/s and 3.5 m/s. Due to lack of time and machine availability, no new plate samples were fabricated and the decision of using the used plate was taken. The plates were clamped in a way such that the previous experiment sections on plate do not interfere with the new experiment. Figure 3.22 shows the reused plate with new experiments performed. At 3 m/s, a bump was seen with major fiber damage and a tiny hole, whereas at 3.5 m/s a complete penetration of the impactor was achieved.

Figure 3.22 LVI experiment at 2 m/s, 3 m/s and 3.5 m/s a) Top face b) Bottom face

3.3.1 Analysis

After the experiment was conducted, the raw data was converted to various plots, including displacement vs time, load vs time, velocity vs time and energy vs time. The first run was conducted using an initial velocity of 2 m/s. This was chosen to compare and verify the data achieved by the previous researcher. Figure 3.23 shows the total deflection of 4.948 mm at 3.85 ms for the sandwich composite plate whereas the composite plate with ten plies of T800s carbon fiber deforms at 4.64 ms for 6.7 mm. This difference is valid as the layers of S2 glass on the top and bottom surface restrict the load transfer and provides more strength.

Figure 3.23 Deflection vs time for 2 different composites

Figure 3.24 Impact load vs time plot for two different composites at 2 m/s

Figure 3.25 Impact energy vs time plot for two different composites at 2 m/s

The plot of force vs time shown in Figure 3.24 compares the carbon fiber composite plate with that of sandwich composite plate to confirm the validity of the experiment at the same speed. Upon comparison it was found that the impact force in the sandwich composite plate is a bit higher than that of carbon fiber plate an approximate error of 28 %. Figure 3.25 compares the energy level between both the plates with an approximate error of 2.5 %. The energy level shown in the plot is made up of two components: E_e , elastic energy and E_a , absorbed energy, which adds up to total kinetic energy with some minor energy loss.

Later, multiple tests were performed at differing initial velocities of 2.5 m/s, 3 m/s and 3.5 m/s, whose force vs time and deflection vs time plots are shown in Figure 3.26 and Figure 3.27 respectively. The drop shown in Figure 3.26 depicts the crack in the sandwich composite plate and then the rebound of the impactor.

Figure 3.26 Force vs time plot for different speeds on sandwich composite panel

Figure 3.27 Deflection vs time at different speeds for sandwich composite material

4. Numerical Simulation

This section is important to compare and validate the numerical simulation model with that of the experimental results presented in Section 3 using ANSYS. The low velocity impact testing is modeled as nonlinear dynamic loading / explicit dynamics for the model, and the UTM testing is modeled as static load analysis. The output from this simulation will be deformation, contact force and energy absorption plots for LVI testing, and stress vs strain plot for UTM testing, which are then compared to the values from the DAQ system.

To model the simulations, all the material properties of the specific metal are required. The value for shear moduli, G_{23} and Poisson's ratio, ν_{23} , are missing from the material properties of both the material in Appendix A, and thus its required to calculate an estimated value. These values can be calculated using the semi-empirical Halpin-Tsai model (Hull & Clyne, 1996) using Equation 8 and 9 below.

$$G_{23} = \frac{E_2}{2(1+\nu_{23})} \quad (8)$$

$$\nu_{23} = 1 - \nu_{12} - \frac{E_2}{3K} \quad (9)$$

Where, E_2 is the elastic modulus in transverse direction and K is the bulk modulus, which can be expressed a follows:

$$K = \left(\frac{f}{\nu_f} + \frac{1-\nu_f}{K_m} \right)^{-1} \quad (10)$$

Here, ν_f is volume fraction of fiber, K_f and K_m are bulk modulus for fiber and matrix respectively and are expressed as:

$$K_f = \frac{E_f}{3(1-2\nu_f)} \text{ and } K_m = \frac{E_m}{3(1-2\nu_m)} \quad (11)$$

The parameters, Young's modulus and volume fraction of fiber and matrix denoted as E_f , E_m , v_f , and v_m , respectively, are not provided in the material property data sheet, which are required to calculate G_{23} and v_{23} . To overcome this problem, another method is proposed using the wave equation and UD orthotropic material criteria (Kuo et. al., 2007). This method uses Classical Laminate Theory (CLT), which requires the derivation of stiffness matrix $\mathbf{Q}$ using Equation 12.

$$[\sigma] = \begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{31} \\ \sigma_{12} \end{bmatrix} = \begin{bmatrix} Q_{11} & Q_{12} & Q_{13} & 0 & 0 & 0 \\ Q_{12} & Q_{22} & Q_{23} & 0 & 0 & 0 \\ Q_{13} & Q_{23} & Q_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & Q_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & Q_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & Q_{66} \end{bmatrix} \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{23} \\ \gamma_{31} \\ \gamma_{12} \end{bmatrix} \quad (12)$$

where,

$$\begin{aligned} Q_{11} &= E_{11} \frac{1-v_{23}v_{32}}{\Delta} & Q_{12} &= E_{11} \frac{v_{21}-v_{31}v_{23}}{\Delta} & Q_{13} &= E_{11} \frac{v_{23}-v_{23}v_{32}}{\Delta} \\ Q_{22} &= E_{22} \frac{1-v_{13}v_{31}}{\Delta} & Q_{23} &= E_{22} \frac{v_{32}-v_{12}v_{31}}{\Delta} & Q_{33} &= E_{33} \frac{1-v_{12}v_{21}}{\Delta} \\ Q_{44} &= G_{23} & Q_{55} &= G_{13} & Q_{66} &= G_{12} \\ \Delta &= 1 - v_{12}v_{21} - v_{23}v_{32} - v_{13}v_{31} - 2v_{21}v_{32}v_{13} \end{aligned} \quad (13)$$

Now, combining the wave equation and stiffness matrix, $\mathbf{Q}$, (Zhennan, 2020) we get,

$$v_{23} = \frac{-E_{22} \left[E_{11} \left(\frac{1}{2} - v_{12} \right) + 2G_{12}v_{12}^2 \right] + \Delta}{2E_{11}G_{12}} \quad (14)$$

$$G_{23} = \frac{E_{22}}{2(1+v_{23})} \quad (15)$$

where,

$$\Delta = \sqrt{E_{22}^2 [E_{11}(0.5 - v_{12}) + 2G_{12}v_{12}^2]^2 - 4E_{11}G_{12} [E_{11}E_{22}(0.5 - v_{12}) - G_{12}(E_{11} - 2E_{11}v_{12}^2)]} \quad (16)$$

The composite material properties of the CFRP and GFRP ply are shown in Table 4.1 and the values are derived from Appendix A. After inputting these values in Equation 14 and Equation 15, we get $G_{23} = 2.548$ GPa and $\nu_{23} = 0.628$ for CFRP material and $G_{23} = 2.4202$ GPa and $\nu_{23} = 0.652$ for GFRP material.

Table 4.1

Mechanical properties of T800s carbon/3900 Epoxy and S2 glass/381 Epoxy lamina under 22°C

Properties	T800s Carbon Prepreg	S2 Glass Prepreg
ρ (kg/m ³)	1580	1860
E_1 (GPa)	148	47.9
E_2, E_3 (GPa)	8.3	8
ν_{12}	0.33	0.32
G_{12} (GPa)	3.93	4.6
X_T (MPa)	2965	1765
X_C (MPa)	1779	1227
Y_T (MPa)	60.3	60
Y_C (MPa)	214	122.7
S_T, S_C (MPa)	68.9	86.2, 46.2

4.1 Ultimate Tensile Machine Testing Simulation

The UTM model is created to understand the experimental procedure and compare the resultant data. A plate of 250 mm x 25 mm is modeled with the sandwich composite material defined earlier in ACP pre. The model is transferred as a solid element to the static structural module, where the bottom face is fixed and a max load of 90 kN is applied to the top face resembling the tensile testing experiment performed. The tensile load is applied at increments of 5 kN to output the stress and strain data for each simulation. Later, all the data is compiled, and a plot of stress vs strain is derived. During the simulation, it was noticed that the maximum stress occurred at the center ply, or ply

of symmetry, shown in Figure 4.1. This is valid as during the experiment core plies were stretched more than the outer skin ply.

Figure 4.1 Maximum stress in ply of symmetry or ply 8/9

Figure 4.2 Stress vs strain plot for UTM simulation data

Figure 4.2 shows the stress vs strain plot for the UTM simulation and yields the Young's modulus of 63.68 GPa, which is an underestimation in comparison to the Young's modulus of 78.2 GPa for the sample 3 experimental data. This error of approximately 18 % might have occurred due of the usage of 4-year-old S2 glass, after completely using its shelf life. This leads to matrix degradation, which renders it unable to cure with another ply.

4.2 Low Velocity Impact Finite Element Simulation

To understand the behavior and compare the accuracy of the LVI experiment, a non-linear transient analysis, or explicit dynamic analysis, was done in MSC Nastran by modeling the impact between the plate and the hemispherical impactor. This was then compared with the data obtained from the LVI experiment conducted. The geometry of the plate was defined according to the properties defined in Table 4.2. This model will then be used for the future analysis and optimization process.

Table 4.2

Geometric properties of sandwich composite plate/coupon

Properties	Value
Length (mm)	152.4
Width (mm)	101.6
Thickness (mm)	2.762

4.2.1 Modal Analysis

Modal analysis is important to understand, where the maximum vibration occurs, and at what natural frequencies. Mode shapes tell us how the structure tends to deform at the specific natural frequencies and which regions would experience high stresses if the

deformed shape were like that of the mode shape. This analysis can be useful, as it would provide information of any high stress regions on the bearing connection or welded region of HUT, which is important as it can affect the fatigue life of the structure.

The analysis was conducted by modeling the coupon as a shell element when transferring the data from the ACP (pre) module to the Modal Analysis module. The plate was then fixed from all four edges to resemble the LVI testing. The result of modal analysis is shown in Table 4.3 for the first ten modes shapes, along with plot of the first three mode shapes in Figure 4.3, Figure 4.4 and Figure 4.5 respectively.

Figure 4.3 Mode shape 1 at 1560.8Hz

Figure 4.4 Mode shape 2 at 2290.5Hz

Figure 4.5 Mode shape 3 at 3680.8Hz

Table 4.3

Modal frequency of Sandwich Composite Plate

Modes	Frequencies (Hz)
1	1560.8
2	2290.5
3	3680.8
4	3822.5
5	4264.7
6	5255.3
7	5616.7
8	6857.8
9	7116.2
10	7432.9

4.2.2 Linear Transient Analysis

The linear transient analysis was conducted using the contact force vs time data for an impacted single element of a quarter plate, which was extracted using SOL 400 in MSC PATRAN (Zhennan, 2020). The same data is used in this analysis, but it is distributed over the entire plate, i.e., nodes 1, 2, 17, and 18 for the quarter plate is redistributed to the other 3 elements connecting the middle node (node 1), shown in Figure 4.6, to cause symmetry and resemble the impact area of hemispherical impactor.

Figure 4.6 Plate with nodal forces applied to the center elements

Figure 4.7 Maximum Displacement contour plot for T800s carbon fiber

The plate was first modeled with a ten ply of T800s carbon fiber material, and the force vs time data was applied to compare the displacement data achieved previously. The contour plot on the full plate is shown in Figure 4.7. The maximum displacement of 6.469 mm occurred at the central node (node 1) of the plate at 5.2 ms, which is shown in Figure 4.8. This data can be confirmed through the contour plot.

Figure 4.8 Displacement vs time plot for T800s Carbon Fiber ten layers

To compare the basic composite structure, a sandwich composite coupon model was designed as stated in Section 3.1. The stacking sequence for this sandwich composite plate is symmetric and is shown in Table 4.4. The sandwich composite plate was placed under the same force vs time data on the same nodes and computed via linear transient analysis. It was found that the maximum deflection occurred at the center node with deflection of 3.388 mm at 4.8 ms, which can be seen in the Figure 4.9 and can be compared with its contour plot, which is shown in Figure 4.10. Thus, it can be concluded that the sandwich composite structure is stiffer and can absorb more impact energy than multiple layers of a single material (carbon fiber).

Table 4.4

Stacking sequence of composite sandwich plate

Number of Ply	Material	Angle (deg)	Thickness (mm)
1	S2 Glass	0	0.0567
2	S2 Glass	90	0.0567
3	S2 Glass	0	0.0567
4	T800s	90	0.191
5	T800s	0	0.191
6	T800s	90	0.191
7	T800s	0	0.191
8	T800s	90	0.191
9	T800s	90	0.191
10	T800s	0	0.191
11	T800s	90	0.191
12	T800s	0	0.191
13	T800s	90	0.191
14	S2 Glass	0	0.0567
15	S2 Glass	90	0.0567
16	S2 Glass	0	0.0567
Total Thickness			2.2502

Figure 4.9 Displacement vs time for sandwich composite plate

Figure 4.10 Contour plot of maximum displacement for sandwich composite plate

4.2.3 Non-linear Transient Analysis

After modal analysis, non-linear transient analysis was performed in MSC. Nastran to understand and simulate the low velocity impact experiment. This can be done by modeling the plate and impactor in the center while fixing all the sides of the plate. To simplify the optimization process, the model was sectioned in quarters due to symmetry with new boundary conditions shown in Table 4.5, where edge 1 and 2 are simply supported and edge 3 and 4 are fixed.

Table 4.5

LVI model boundary constraints (Zhennan, 2020)

	Translation (T)	Rotation (R)
Node 1	X, Y	X, Y, Z
Edge 1	Y, Z	Y
Edge 2	X, Z	X
Edge 3	X, Y, Z	X, Y, Z
Edge 4	X, Y, Z	X, Y, Z

Here, SOL 400 was used to simulate the LVI experiment. It is selected because of its ability to perform nonlinear analysis as it uses advanced element technology with the ability to handle large deformations and rotations, while maintaining an accurate nonlinear strain-deformation relationship (Zhennan, 2020). Another reason of choosing SOL 400 is its ability to perform transient dynamic structural analyses.

The impactor was modeled as a rigid element and the quarter plate was modeled as a Quad4 shell element to simulate the laminate properties using S2 glass and CFRP material as sandwich composite structure. The impactor was given initial speed of 2 m/s in -z direction to simulate the experiment and environment. Figure 4.11 shows the LVI model for SOL 400.

Figure 4.11 LVI model of SOL 400 (Zhennan, 2020)

To understand and compare the difference of using sandwich composite structure, displacement vs time data was plotted. SOL 112 was used to output the displacement vs time data. First, the data of CFRP material was compared with the sandwich composite using the CFRP load case at 2 m/s initial impactor velocity. It can be seen in Figure 4.12

that the displacement at the same load for two different composite plate varies by approximately 26 %. The max displacement recorded for CFRP material is 6.9 mm at 5ms and 5.05 mm at 5.2 ms for sandwich composite plate. This proves that the sandwich structure can resist more impact force than that of a composite material by itself. It was calculated that the displacement reduced by 26.81 % from the original composite material used by previous researcher.

Figure 4.12 Displacement vs time for same load case (CFRP)

Now, SOL 400 was ran to get the contact force vs time data for the sandwich composite plate which was then used as an input for SOL 112 to output the displacement vs time data at 70 % scaled force load. Figure 4.13 shows the displacement vs time plot of new load case compared with the LVI experiment and SOL 400 data. This suggests

that the high-fidelity numerical model developed in Nastran precisely simulates the real-time experiment with an error of 1 %.

Furthermore, another validation of the Nastran model was made for initial velocity of 2.5 m/s. Figure 4.14 shows that the SOL 400 at 2.5 m/s behaves like the experimental value at same speed with an error of 1.7 %.

Table 4.6

Error % comparison between LVI Experiment, SOL 112 and SOL 400

Model	Displacement (mm)	Error % (w.r.t. LVI)	Error % (w.r.t SOL 112)
LVI Experiment	4.9485	0	2
SOL 112	5.05	2	0
SOL 400	4.995	0.94	1.1

Figure 4.13 Displacement vs time plot comparing SOL 400, SOL 112 and LVI experiment

Figure 4.14 Displacement vs time plot comparing two initial velocity simulations and experimental data

5. Structural Optimization

In this chapter, the optimization of the thickness of the composite plate for minimizing the weight of the plate was conducted with reduction of plate deflection i.e., the deflection the plate that would undergo when subjected to LVI testing as a constraint. MSC. Nastran was chosen to perform this optimization process. MSC. Nastran offers a wide range of optimization algorithms as it has abilities to perform optimization in the domains of shape, size, topology, topometry and ability to carry out multi model optimization as well. For this work, shape optimization would be used along with multi model optimization. The shape optimization helps in obtaining the optimum minimum weight of the composite plate being the primary objective of this thesis. Further, the multi model optimization algorithm would be incorporated as the primary and secondary objectives must be processed simultaneously.

5.1 Sizing Optimization

This optimization process was solved using MSC. Nastran SOL 200. The output from SOL 400 will be then used as input for SOL 200 to complete the process. Due to the inability of SOL 200 to simulate nonlinear dynamic response the data from SOL 112 is going to be used. Zhennan et al (2020) used SOL 200 to perform shape and sizing optimization which would be used in this research. Size optimization would be carried out for the design variables being the thickness of plate and the other being the fiber orientation of the composite plies. The optimization problem would have 32 design variables in total which would constitute 16 design variables for the thickness of plies with each variable representing the thickness of a single ply of the 16-ply composite

plate. The other 16 design variables would be the fiber orientation variables for each of the 16 plies.

The primary objective of the optimization process was to minimize the mass of the composite laminate plate. The mass of the plate depends on the volume of the plate and the density of the material used. The plate having the shape of a rectangular block, the mass of the plate majorly depends on the length (L), width (W), thickness (T), and the density of the material (ρ). Two different materials have been used to prepare the laminates. Hence, the objective function must be modified accordingly. The objective function for mass minimization is presented below,

$$\text{Min } m = L * W(10 * \rho_1 * T_1 + 6 * \rho_2 * T_2) \quad (17)$$

Subjected to,

$$0 < X_i \leq 0.4, i = 4, \dots, 13 \quad (18)$$

$$0 < X_j \leq 0.15, i = 1, 2, 3, 14, 15, 16 \quad (19)$$

$$\delta \leq \delta_{allowable} \quad (20)$$

And / or,

$$\sigma \leq \sigma_{allowable} \quad (21)$$

Where, T_1 and ρ_1 are the thickness and density of the carbon-fiber ply, T_2 and ρ_2 are the thickness and density of the S2 glass ply, in Equation 17.

Using the material properties from Table 4.1 and the geometric parameters of the plate from Table 4.2, the initial mass of the composite laminate plate comes out to be 56.28 g. The 32 design variables were defined in SOL 200 algorithm using the DVPREL1 entries. The upper and lower boundaries for the design variables were defined using DESVAR entries. As the composite plate has same number of plies across its

neutral plane, only sixteen design variables i.e., half the plate, are selected and the rest are linked with DLINK entry. The optimization for the secondary case i.e., the fiber orientation of the plies is carried out with a 15° increment in the value ranging from 0° to 90° . The other entries used in the algorithm are presented in Table 5.1.

Table 5.1

Bulk Data Entry Nastran card (Zhennan, 2020)

Entry name	Descriptions
DVPREL1	Defines the relation of the model property with the variables
DESVAR	Defines the design variables
DLINK	Relates a variable with more others
DDVAL	Defines discrete design variables values
DRESP1	Defines the structural responses as an objective or as constraints
DCONSTR	Define design constraints
DCONADD	Defines the constraints as a union of DCONSTR entries
DOPTPRM	Override default values of the parameters

The maximum number of iteration that the algorithm would undergo was set to 90. The optimization converges after four iterations. Figure 5.1 and Figure 5.2 shows the change in objective and normalized constraint value. The objective value or the total weight of the quarter plate increased from 13.95 g to 15 g, which is a violation of the optimization process, but the normalized constraint reached negative value which resulted in decrease in max displacement of the plate from 4.947 mm to 3.085 mm at 2m/s initial velocity, shown in Figure 5.3. The increase in weight was due to the change in the thickness of each ply. As seen in Figure 5.4 and Figure 5.5, the thickness of ply 1, 3, 5 and 8 increased by an average of 15 % of original thickness, and the ply angles have changed to 15/45 orientation instead of 0/90 orientation. The final ply thickness and

angles for the symmetric sandwich composite plate after the size optimization is shown in Table 5.2.

Figure 5.1 Objective history of SOL 200

Figure 5.2 Normalized constraint history of SOL 200

Figure 5.3 Deformation comparison of SOL 200

Figure 5.4 Ply thickness history of SOL 200

Figure 5.5 Ply angle history of SOL 200

Table 5.2

Results of sizing optimization using SOL 200 for quarter plate

Property		Initial Value	Final Value	Change in value
Thickness (mm)	X ₁ , X ₁₆	0.0526	0.0614	16.73 %
	X ₂ , X ₁₅	0.0526	0.0067	(-) 87.26 %
	X ₃ , X ₁₄	0.0526	0.0546	3.80 %
	X ₄ , X ₁₃	0.191	0.0950	(-) 50.26 %
	X ₅ , X ₁₂	0.191	0.2364	23.77 %
	X ₆ , X ₁₁	0.191	0.2850	49.21 %
	X ₇ , X ₁₀	0.191	0.1806	(-) 5.44 %
	X ₈ , X ₉	0.191	0.2850	49.21 %
Angle (deg)	X ₁ , X ₃ , X ₅ , X ₇ , X ₁₀ , X ₁₂ , X ₁₄ , X ₁₆	0	15	
	X ₂ , X ₄ , X ₆ , X ₈ , X ₉ , X ₁₁ , X ₁₃ , X ₁₅	90	45	
Objective	Total Weight (g)	13.95	15	7.52 %
Constraint	Displacement (mm)	4.947	3.085	(-) 37.63 %

It is known that the deformation of the composite can only be decreased if the number of ply or the thickness is increased, making the panel stiff. With a little price to pay, the maximum deflection of the sandwich composite plate reduced by 37.6 % in comparison to the LVI experiment conducted in section 3.3.1 and by 38 % in comparison to the Nastran model developed in section 4.2.3. This increase in objective can be minimized through the process of shape optimization.

5.2 Shape Optimization

The shape optimization process is like that of the weight optimization but with different design variables. Here, the maximum deflection and stress induced by the plate are to be minimized using geometric shape parameters and the shape basis vector for grid points/nodes as the design variable.

The computational cost of shape optimization is comparatively higher than the size optimization. The shape of the structure in terms of grid nodes is considered as the design variable for this problem and its optimal value is determined using the shape basis vector, implementing the line search using Sequential Quadratic Programming (SQP) algorithm. Gradient based methods are used to achieve the improved design from the line search which make use of the first derivative of the objective function. SOL 200 supports manual grid variation, direct input of shapes, geometric boundary shapes and analytical boundary shapes methods. For the object with simple shape such as a plate, the direct input of shapes and geometric boundary shapes provides reasonable accuracy to the optimized solution. But with the sandwich composite panel, geometric boundary shape method will be more applicable with the optimization formulation mentioned in Equation 22 through 26.

$$\text{Min } m = L * W(10 * \rho_1 * T_1 + 6 * \rho_2 * T_2) \quad (22)$$

Subjected to,

$$0 < L \leq 76.2 \quad (23)$$

$$0 < W \leq 50.8 \quad (24)$$

$$\delta \leq \delta_{allowable} \quad (25)$$

And / or

$$\sigma \leq \sigma_{allowable} \quad (26)$$

Geometric Boundary Shapes use method of generating shape basis vector based on the geometric data of boundaries (MSC Nastran user's guide). This was done using the BNDGRID entries for the boundary grid points which changes based on the data provided to DVGRID. DVGRID provides variations to the grid points in any shape defined, which then creates the shape basis vector associated with the design variable.

The first shape optimization is considered using the length and width as design variable represented using X and Y direction of the shape variables. This can be done using the BNDGRID entry which defines that edge 1 and edge 3 are free in T2 (Y-direction) whereas edge 2 and edge 4 are free in T1 (X-direction). This method is chosen to understand how the SOL 200 incorporates the boundary grid constraints for optimization. The optimization was initiated with the symmetric base model of 0/90 orientation of fiber angle shown previously in Table 4.4.

After the optimization it was found that the objective and constraint were achieved in four design cycle with hard convergence and this can be seen in Figure 5.6 and Figure 5.7. The objective or the total weight decreased by 19.57 % from 13.95 g to 11.22 g along with the decrease in normalized constraint to -0.05, achieving the feasible design criteria.

The new optimized data shown in Table 5.3 was used as an input for SOL 112 to output the displacement vs time data at node 1, shown in Figure 5.8. It was found that the max displacement was achieved to be 3.814 mm which is a 22.9 % decrease from the original displacement. The decrease in displacement using length-width optimization was less than that of the size optimization but fulfilled both the objective and constraint criteria successfully.

Figure 5.6 Objective history for length-width optimization

Table 5.3

Results of shape optimization using SOL 200 for quarter plate

Property	Initial Value	Final Value	Change in value
Length (mm)	76.2	69.95	(-) 8.2 %
Width (mm)	50.8	44.5	(-) 12.4 %
Total Weight (g)	13.95	11.22	(-) 19.57%
Displacement (mm)	4.947	3.814	(-) 22.9 %

Figure 5.7 Constraint history for length-width optimization

Figure 5.8 Deformation comparison of length-width optimization to original data

The second method used for shape optimization for shape basis vector is by using quadratic function. MATLAB is used to calculate the vectors for the top and right boundary using Equation 27 and the normalized coordinates of those boundaries.

$$y = (x)^2 \quad (27)$$

Figure 5.9 shows the predicted initial shape of the plate after optimization. This was achieved through the DVGRID that furnishes the shape variation of boundary. After running the optimization, it took four design cycles to converge to a feasible point and this can be confirmed by noticing the normal constraint reaching -0.0267 which can be seen in Figure 5.10. The constraint and objective history of the optimization is shown in Figure 5.10 and Figure 5.11 respectively.

Figure 5.9 Quadratic shape of the plate using MATLAB

Figure 5.10 Constraint history for quadratic function shape optimization

Figure 5.11 Objective history for quadratic function shape optimization

In Figure 5.12, the max displacement for node 1 is reduced by 5.1 % and total weight by 1 % from 14.13 g to 13.99 g, fulfilling the constraint and objective. As the change is small relative to the length-width optimization, , new shape functions were explored.

Figure 5.12 Deformation comparison of quadratic shape

The third method used is the sine function in form of

$$y = x * \sin(x) \quad (28)$$

where x is the normalized displacement for each grid point of respective boundaries.

Figure 5.13 shows the predicted shape vector for top and right boundary whereas Figure 5.14 shows the optimized shape of the plate. It can be noticed that the optimized plate shape follows the predicted shape resulting in max displacement of 4.4 mm which is a reduction by 11.10 % with respect to original plate shown as black curve in Figure 5.15.

The optimization process converged in four design cycles providing hard compromised

converged feasible point whose design is shown in Figure 5.14. The constraint of the optimization problem was fulfilled but the objective was increased by 2.5 % which is a violation of the given problem and thus requires to move to a new shape function.

Figure 5.13 Sine shape of the plate using MATLAB

Figure 5.14 Optimized Sine shape in FEMAP

Figure 5.15 Deformation comparison of sine shape

The fourth function used for shape optimization is Hicks-Henne Bump Function. The Hicks-Henne Bump Function is particularly useful for aerodynamic purpose or to model uncertainties on the geometries of an airfoil. This function creates a set of y-coordinates as a function of x coordinate, location, width, and magnitude of bumps. Figure 5.16 shows the shape of this function in comparison to the base model, where three different shapes functions are shown for both top and right boundary caused by the three different magnitudes: 0.2, 0.5 and 0.7. This results in six design variables for this method providing flexibility for optimization.

Figure 5.16 Hicks-Henne Bump shape of the plate using MATLAB

Figure 5.17 Optimized Hicks-Henne Bump shape in FEMAP

The optimization process took five design cycles to have a hard convergence at a feasible design point. Figure 5.17 shows the optimized shape of the quarter plate where

the top boundary follows the purple curve, and the right boundary follows the orange curve from Figure 5.16 with move limits of 1. Figure 5.18 and Figure 5.19 shows the objective and normalized constraint history where the normalized constraint is at -0.45 proving the successful completion of design optimization. Maximum stress of 1570 MPa was used as constraint for this function using SLP method to conduct the optimization process. The total weight using Hicks-Henne Bump Function was reduced by 24.49 % and the stress was reduced by 2.03 % as described in Table 5.4. Even with less change in stress, displacement was reduced by 28.09 % shown in Figure 5.20, making this method the most feasible than all the above functions and to be used for shape sensitivity analysis.

Figure 5.18 Objective history for Hicks-Henne Bump Function shape optimization

Figure 5.19 Constraint history for Hicks-henne Bump Function shape optimization

Figure 5.20 Deformation comparison of Hicks-Henne Bump Function

Table 5.4

Results of shape optimization using Hicks-Henne Bump Function for quarter plate

Property Type	Initial Value	Final Value	Change in value
Total Weight (g)	14.13	10.62	(-) 24.49
Displacement (mm)	4.95	3.56	(-) 28.09
Stress (MPa)	962.59	943.086	(-) 2.03

5.3 Shape Sensitivity

Shape sensitivity for the shape basis vector can be performed using the semi-analytical method as default option for NASTRAN that approximates the stiffness matrix and force vector derivatives using the finite difference method for the element used. Feneys et. al. noticed that the error in displacement sensitivity for plate element increases with increase in different dimensions of the stiffness matrix and suggested a solution by using a very small-time step or applying various finite difference scheme. The small step-size for the finite difference approximation can be achieved by using the DELB entry card in the DOPTPRM bulk data entry card. The default step-size for this relative finite difference move parameter in MSC. Nastran is 0.0001 and 0.001 for fatigue response (MSC Nastran user's guide) when used for property optimization using STATIC analysis in SOL 200.

As described in Section 5.2, shape basis vector uses grid points that are modeled as a design variable which can be changed by relating it to the mathematical function. With the realization of DELB being used for property optimization, another parameter, STPSCL, is used for shape finite difference step-size. STPSCL is a scaling factor which can be applied to all the shape design variable by perturbing the grid coordinates using Equation 29, where i is the variation in shape for the i^{th} design variable, E_i is the

maximum strain energy norm and T_i is the shape basis vector (MSC Nastran user's guide).

$$\{\delta G\}_i = \frac{STPSCL}{E_i} \{T\}_i \quad (29)$$

The maximum strain energy norm is calculated by finding the grid force due to the shape basis vector displacement and later finding the strain energy vector per grid location using Equation 30. The maximum strain energy is calculated using the Equation 31 for the strain energy vector which was used in Equation 29 to yield a set of finite difference move parameters.

$$\{E_i\} = \{F_s\}\{T\}_i \quad (30)$$

$$E_i = \sqrt{(\max\{E\}_i)} \quad (31)$$

5.3.1 Identifying Correct Finite Difference Step Size

As discussed in the previous section, STPSCL parameter is the only option available for changing the step-size of finite difference method indirectly for shape basis vector in MSC. Nastran. This parameter can now be used in various finite difference schemes to calculate the shape sensitivities and provide feasible optimum design points using the Hicks-Henne Bump Function.

Finite difference scheme is an approximation method to calculate the derivatives using the Taylor Series expansion. MSC Nastran uses first-forward difference and central difference methods to provide low-cost derivative approximations where forward difference is a default option for property optimization and central difference is used for shape optimization. These methods can alternatively be used using the PARAM,CDIF,YES and PARAM,CDIF,NO cards where “YES” forces the selection of central difference and “NO” is to use forward difference. The first-forward difference

method calculates the sensitivities by perturbing the design variable in first order of the step-size, making the calculations faster but less accurate. Whereas the central difference method uses both the forward and backward difference methods resulting in more accurate data but at the cost of increase in computational time by using second order derivatives of the step size.

The shape optimization for the sandwich composite quarter plate using the Hicks - Henne Bump Function for the fixed boundaries was calculated using the two different shape sensitivity method first-forward finite difference and central finite difference. These methods are then compared for two different optimizers: sequential linear programming (1st order approximation) and sequential quadratic programming (2nd order approximation). The parametric study between the two different approximation methods with respect to the maximum displacement at ten different scaling points between 0 and 100 spaced logarithmically is to be calculated.

Table 5.5 and Table 5.6 shows the mentioned comparison for max displacement and total weight for two different optimizers. Based on the data, it can be concluded that at 63.5020 step scale with an error of 0.0933 % for SQP and 0.1261 % for SLP, least error can be achieved simultaneously with respect to maximum displacement, and error of 0.0193 % for SQP and 0.04955 % for SLP for total weight. Some step scale provides lower error percentage for SQP where higher error percentage for SLP was noticed. This difference does not meet the requirement of achieving a common step size where both the sensitivity methods reduce to approximately the same value providing a standard feasible design. Thus, the way of choosing this parameter value is not intuitive, hence making shape optimization challenging in Nastran.

Table 5.5

Optimized displacement for different step size scale comparing two different sensitivity analysis method

Step Scale	SLP			SQP		
	Central Difference	Forward Difference	Error %	Central Difference	Forward Difference	Error %
1.2589	3.7818	3.9838	5.3395%	3.5859	3.5859	0.0000%
2.0470	3.8631	3.5871	7.1444%	3.5859	3.5859	0.0000%
3.3283	3.5867	3.5859	0.0225%	4.0363	4.0363	0.0001%
5.4117	3.5878	3.5887	0.0248%	4.0366	4.0367	0.0026%
8.7992	3.6452	3.6419	0.0901%	4.4149	3.8292	13.2661%
14.3072	3.7294	3.7691	1.0645%	4.1186	4.0378	1.9611%
23.2631	3.5871	3.7968	5.8461%	4.0388	4.0388	0.0008%
37.8249	3.4912	3.4921	0.0237%	3.4707	3.4738	0.0902%
61.5020	3.4795	3.4839	0.1261%	3.4435	3.4467	0.0933%
100	3.4371	3.4470	0.2877%	3.8124	3.8117	0.0194%

Table 5.6

Optimized total weight for different step size scale comparing two different sensitivity analysis method

Step Scale	SLP			SQP		
	Central Difference	Forward Difference	Error %	Central Difference	Forward Difference	Error %
1.2589	10.9103	10.7981	1.0284%	10.8930	10.6179	2.5255%
2.0470	10.9286	10.6183	2.8393%	10.8931	10.6179	2.5264%
3.3283	10.9286	10.6177	2.8448%	10.8076	10.8076	0.0000%
5.4117	10.9282	10.6123	2.8907%	10.8077	10.8077	0.0000%
8.7992	11.2044	11.1991	0.0473%	12.3106	10.8935	11.5112%
14.3072	10.6808	10.7729	0.8623%	11.8180	10.8097	8.5319%
23.2631	10.6181	10.6897	0.6743%	10.8081	10.8081	0.0000%
37.8249	9.2296	9.2303	0.0070%	9.1755	9.1772	0.0185%
61.5020	9.1995	9.2040	0.0495%	9.1584	9.1601	0.0193%
100	9.1588	9.1641	0.0575%	10.7977	10.7988	0.0102%

6. Conclusions and Recommendations

It is known that spacesuits are costly and are difficult in processing/manufacturing components of a space mission as they must maintain their structural strength and stability for a long time in space. To minimize the repetitive manufacturing of spacesuit components for testing, a high-fidelity model was created using MSC. Nastran/Patran and ANSYS which can be tested for various loading conditions or real-life environment. In this thesis, the model was created using a sandwich composite material made up of S2 glass fiber prepreg as the top and bottom skin and T800s carbon fiber prepreg as the core. To validate this model, the coupon was first fabricated using the same materials and were later used for LVI testing to simulate the dropping of handheld devices. Lastly, the model was optimized using MSC. Nastran SOL 200 to increase the damage resistance or minimize deflection for the sandwich composite plate.

6.1 Conclusion

The LVI testing was conducted using two sample plates with an initial impactor (hemispherical) speed of 2 m/s and 2.5 m/s. It was noticed that at 2.5 m/s, a small bump formed at the back-face of the plate, which led to conduct another set of experiments at 3 m/s and 3.5 m/s until full penetration was achieved. Later, the numerical model was developed to run the LVI testing using the same material properties as that of the experiment. It was concluded that the numerical model successfully validated with the experimental data with a maximum error of 1 % in displacement.

The numerical model was validated with two different Nastran solutions: SOL 400 and SOL 112 where, SOL 400 is nonlinear transient analysis and resembles the LVI experiment, and SOL 112 is static analysis that inputs force data from SOL 400 and

outputs the displacement vectors. During validation, it was seen that the displacement values using both the solutions approximately match with an error of 1.1 % and 0.94 % when compared with the LVI experiment. Thus, it can be concluded that the numerical model developed works well and is ready to be used to simulate different environment and load cases.

The above validation was conducted to ensure the use of the model for structural optimization. Sizing optimization was first analyzed to increase the damage tolerance while minimizing the weight of the sandwich composite. The objective for this optimization did not converge as desired, but the constraint was minimized by 38 % as expected. This analogy is still correct because increasing the weight will decrease the deflection experienced on the sandwich composite plate. Another way to optimize the sandwich plate is through the shape optimization.

Shape optimization was done to understand the change in deflection and stress behavior of the sandwich composite plate by changing its geometric parameters. For this optimization, the length and width of the plate were chosen as the design variables and were changed using the DVGRID and BNDGRID entries. It was found that the max displacement at node 1 was reduced by 23 % and weight was minimized by 19.2 %. Even if both the objective and constraint were achieved with hard convergence, the displacement was slightly higher than that of size optimization.

A shape basis vector was introduced to overcome this problem, which was changed by relating the grid points in the DVGRID entry card with various mathematical functions like quadratic, sine and Hincks-Henne Bump Functions. After the optimization, it was concluded that the Hicks -Henne Bump Function provided the most feasible design

with 24.49 % decrease in total weight and 28.09 % decrease in the maximum displacement. This function was chosen for the shape sensitivity analysis that compared central finite difference and first-forward finite difference derivative approximation methods using SLP and SQP as an optimization method. The step size scale of 61.5020 was chosen to be the most feasible design point that reduced the max displacement constraint by 30.32 % and total weight by 35.18 % using forward difference method for SQP when compared to the original model.

6.2 Recommendations and Future Work

Several sections of this research could be improved efficiently and be expanded to a wider research field to try and compare different options available. These improvements are listed below as future work due to the time and lab availability during this research period.

- **Change of material:** The core of the sandwich material, T800s carbon fiber must be changed to the IM10 carbon fiber to understand and improve the model with the original material used in the spacesuits. The new sandwich plate is then be used for LVI test and SOL 400 simulation in MSC. Nastran. This change will help to continue the research work paused by NASA until 2028.
- **Continuum Sensitivity Analysis:** The shape optimization could be taken to the next step by using different functions for shape basis vector and their sensitivities and compare them with the Hicks-Henne Bump method. Another sub-step is to include fatigue as another constraint converting the simple problem to a multi-constraint problem and optimizing the plate for better results. The optimization process should be carried with the entire plate to understand the process even though it is computationally

expensive. Another step is to conduct shape sensitivity using static forces and Adjoint Method where step size can be controlled by DELB parameter. Comparison could be made between transient and static forces using different sensitivity analyses to find the appropriate method that provides the most feasible optimum design.

- Compression after impact test: Damage due to the low velocity impact on fiber-reinforced composites is thought to be very risky for the most part as the damage is not detectable to the exposed eye causing Barely Visible Impact Damage. A composite's compressive strength can undergo a loss of about 60% with this type of damage, thus it is required to perform a compressive strength test of the plate before and after the impact using materials used in this research and the Z-2 spacesuit materials. This test can be conducted using the UTM or other machines according to the ASTM standards if available.

- Low Velocity Impact test after optimization: LVI test after optimization is required to validate the data obtained from the SOL 200 and SOL 112 shown in the section 5, structural optimization. This validation will finally prove that the Nastran model is correct and can be used in the future for any optimization process using different materials.

REFERENCES

- Alomari, A. S., Al-Athel, K. S., Arif, A. F., & Al-Sulaiman, F. A. (2020). Experimental and Computational Analysis of Low-Velocity Impact on Carbon-, Glass- and Mixed-Fiber Composite Plates. *Journal of Composites Science*, 4(4), 148. DOI:10.3390/jcs4040148
- Choi, W., Kim, J., & Park, G. (2016). Comparison study of some commercial structural optimization software systems. *Structural and Multidisciplinary Optimization*, 54(3), 685-699. DOI: 10.1007/s00158-016-1429-y
- Dunbar, B. (2013, June 05). Learn About Spacesuits. Retrieved from https://www.nasa.gov/audience/foreducators/spacesuits/home/clickable_suit_nf.html
- Fenyves, P. A., and Lust, R. V., Error analysis of semianalytic displacement derivatives for shape and sizing variables, *AIAA Journal*, vol. 29, 1991, pp. 271–279.
- Ferl, J., Hewes, L., Cadogan, D., Graziosi, D., & Splawn, K. (2006). System Considerations for an Exploration Spacesuit Upper Torso Architecture. *SAE Technical Paper Series*. DOI:10.4271/2006-01-2141
- Francis, S. (2020, March 31). Composites in the race to space. *CompositesWorld*. Retrieved from [https://www.compositesworld.com/articles/composites-in-space\(2\)](https://www.compositesworld.com/articles/composites-in-space(2))
- Graziosi, D., Jones, R., Ferl, J., Scarborough, S., Hewes, L., Ross, A., & Rhodes, R. (2016, July 10). Z-2 Architecture Description and Requirements Verification Results. Retrieved from <http://hdl.handle.net/2346/67653>
- He, Zhennan (2020). Analysis of After Impact Characteristics and Structural Optimization of CFRP Composite Plate. PhD Dissertations and Master's Thesis. 555. Retrieved from <https://commons.erau.edu/edt/555>
- Hull, D., & Clyne, T. W. (1996). *An introduction to composite materials* (Second ed.). Cambridge; New York; Cambridge University Press
- Krzyzak, Aneta & Mazur, Michał & Gajewski, Mateusz & Drozd, Kazimierz & Komorek, Andrzej & Przybyłek, Paweł. (2016). Sandwich Structured Composites for Aeronautics: Methods of Manufacturing Affecting Some Mechanical Properties. *International Journal of Aerospace Engineering*. 2016. 1-10. DOI: 10.1155/2016/7816912.
- Kuo, Y. -, Lin, H. -, Wang, C. -, & Liao, C. -. (2007). Estimating the elastic modulus through the thickness direction of a uni-direction lamina which possesses

transverse isotropic property. *Journal of Reinforced Plastics and Composites*, 26(16), 1671-1679. DOI: 10.1177/073168440708145

Mian, H. H., Wang, G., Dar, U. A., & Zhang, W. (2013). Optimization of composite material system and lay-up to achieve minimum weight pressure vessel. *Applied Composite Materials*, 20(5), 873-889. DOI:10.1007/s10443-012-9305-4

MSC Nastran user's guide design sensitivity and optimization version 2017 (2017b) [Computer Software] MSC Software Co., Santa Ana, CA, US, 11-12

Rajak, D., Pagar, D., Menezes, P., and Linul, E., (2019). Fiber-Reinforced Polymer Composites: manufacturing, properties, and applications, *Polymers*, vol. 11, p. 1667. DOI: 10.3390/polym11101667

Ross, A., Rhodes, R., Graziosi, D., Jones, R., Lee, R., Haque, B., & Gillespie, J. (2014). Z-2 Prototype Space Suit Development. *NASA Technical Reports Server*. Retrieved from <http://hdl.handle.net/2346/59679>

Schmit LA, (1960) Structural design by systematic synthesis. Proceedings, *Second National Conference on Electronic Computation, ASCE, Pittsburgh, PA*, 105–122. Retrieved from <http://www.vrand.com/resources/publications/archive/structural-design-by-systematic-synthesis/>

APPENDIX A

The following figures provides the material properties and the fabrication process of the materials sued in this thesis.

1. Toray 3900: T800s Carbon Fiber

LAMINA/LAMINATE MECHANICAL PROPERTIES: UD T800S-24K - COMMERCIAL READY

PROPERTY	SYMBOL	METHOD	UNITS	CTA	RTA
0° Tensile Strength	F ₁₁	ASTM D3039	ksi (MPa)	410 (2830)	430 (2965)
90° Tensile Strength	F ₂₁	ASTM D3039	ksi (MPa)	8.25 (56.9)	8.75 (60.3)
0° Tensile Modulus	E ₁₁	ASTM D3039	Msi (GPa)	21.5 (148)	21.5 (148)
90° Tensile Modulus	E ₂₁	ASTM D3039	Msi (GPa)	1.40 (9.7)	1.20 (8.3)
0° Compressive Strength	F _{1c}	ASTM D6641	ksi (MPa)	257 (1772)	258 (1779)
90° Compressive Strength	F _{2c}	ASTM D6641	ksi (MPa)	44.0 (303)	31.0 (214)
0° Compressive Modulus	E _{1c}	ASTM D6641	Msi (GPa)	19.0 (131)	19.0 (131)
90° Compressive Modulus	E _{2c}	ASTM D6641	Msi (GPa)	1.40 (9.7)	1.20 (8.3)
In-Plane Shear Strength @ 5% or Ultimate	F ₁₂	ASTM D3518	ksi (MPa)	13.0 (89.6)	10.0 (68.9)
In-Plane Shear Modulus	G ₁₂	ASTM D3518	Msi (GPa)	0.700 (4.83)	0.570 (3.93)
Short Beam Shear Strength	SBS	ASTM D2344	ksi (MPa)	20.0 (138)	14.0 (96.5)
Poisson's Ratio	ν ₁₂	ASTM D3039	-	-	0.33
Poisson's Ratio	ν ₂₁	ASTM D3039	-	-	0.02
Open Hole Tension Strength (25/50/25)	OHT	ASTM D5766	ksi (MPa)	-	73 (503)
Open Hole Compression Strength (25/50/25)	OHC	ASTM D6484	ksi (MPa)	-	43 (296)
Compression After Impact (25/50/25)	CAI	ASTM D7137	ksi (MPa)	-	42
Cured Ply Thickness	CPT	-	in. (mm)	0.0075 (0.191)	

Tension and compression values are normalized to the indicated CPT herein.

Notes:

CTA: -65°F (-54°C), Ambient

RTA: 72°F (22°C), Ambient

ETW: 180°F (82°C), Ambient, Conditioned at 160°F/85% RH until equilibrium

TORAY

Toray Composite Materials America, Inc.

3900

TORAYCA

BAGGING PROCEDURE **STORAGE LIFE**

Out Life*	42 days @ 75 °F
Freezer Life	24 months @ <10 °F

*Tack and drape is optimum at 72°F and 65% RH

CURE CYCLE - AUTOCLAVE

1. Apply full vacuum (minimum of 22"Hg (74.5 kPa) at sea level) to the vacuum bag assembly and check for leak prior to the beginning of the cure cycle. The leak rate shall be less than 2.0"Hg (6.77 kPa) over 5 minutes.
2. Apply 85 + 15/-0 psi (586 + 103/-0 kPa) autoclave pressure to the laminate.
3. Vent the vacuum bag when the autoclave pressure reaches 20psi (138 kPa).
4. Apply the temperature ramp to 350 ± 10°F (177 ± 5°C) at rate of 3.0 ± 2.0 °F (1.7 ± 1.1°C) per minute.
5. Maintain the cure temperature at 350 ± 10°F (177 ± 5°C) for 120 - 180 minutes.
6. Cool vessel to 140°F (60°C) or lower at a maximum rate of 5°F (2.78°C) per minute before removing pressure.
7. Remove the bagged panels from the curing vessel.

2. Cycom 381: S2 Glass fiber

TECHNICAL DATA SHEET
CYCOM® 381
 PREPREG

Table 5 | Mechanical Properties of CYCOM® 381 with S-2 Unidirectional Glass Tape¹

Property	Test Temperature	Value	Test Method
0° Tensile Strength ksi (MPa)	-65°F (-54°C) 75°F (24°C) 160°F (71°C), wet 180°F (82°C) 220°F (104°C)	266 (1834) 256 (1765) 115 (793) 224 (1544) 216 (1489)	ASTM D 638
0° Tensile Modulus msi (GPa)	-65°F (-54°C) 75°F (24°C) 160°F (71°C), wet 180°F (82°C) 220°F (104°C)	6.99 (48.2) 6.95 (47.9) 6.81 (47.0) 6.72 (46.3) 6.63 (45.7)	ASTM D 638
0° Tensile Strain %	-65°F (-54°C) 75°F (24°C) 160°F (71°C), wet 180°F (82°C) 220°F (104°C)	3.80 3.68 1.69 3.34 3.27	ASTM D 638
90° Tensile Strength ksi (MPa)	-65°F (-54°C) 75°F (24°C) 160°F (71°C), wet 180°F (82°C)	10.3 (71) 8.7 (60) 3.6 (25) 6.4 (44)	ASTM D 638
90° Tensile Modulus msi (GPa)	-65°F (-54°C) 75°F (24°C) 160°F (71°C), wet 180°F (82°C)	2.16 (14.9) 1.84 (12.7) 1.09 (7.51) 1.56 (10.8)	ASTM D 638
0° Compression Strength ksi (MPa)	-65°F (-54°C) 75°F (24°C) 160°F (71°C), wet 180°F (82°C) 220°F (104°C)	177 (1220) 178 (1227) 137 (945) 173 (1193) 129 (889)	ASTM D 695
0° Compression Modulus msi (GPa)	-65°F (-54°C) 75°F (24°C) 160°F (71°C), wet 180°F (82°C) 220°F (104°C)	6.9 (47.6) 7 (48.3) 6.94 (47.9) 6.95 (47.9) -	ASTM D 695

¹: Autoclave cure cycle, Cure ply thickness = 0.0053 in (0.142 mm), 33% nominal resin content.

TECHNICAL DATA SHEET
CYCOM® 381
 PREPREG

PROCESSING

Guidelines for Thawing Out

On Removal of sealed prepreg from 0°F (-18°C) storage, ensure that the material is allowed to thaw fully to room temperature prior to unsealing. This avoids condensation. Typically a 60 yd (55m) roll of prepreg requires about 8-12 hours to thaw.

Recommended Cure Cycle

Autoclave Cure Cycle

Apply full vacuum, 24 in Hg (0.081 MPa) minimum.
 Heat from 75°F (24°C) to 260°F ± 10°F (127°C ± 5°C) at 5°F (3°C)/minute.
 Apply 50 psi (0.35 MPa) positive pressure, vent vacuum at 45 psi (0.12 MPa).
 Hold at 260°F (127°C) for 100 +10/-0 minutes.
 Cool under pressure to below 150°F (66°C) at 5°F (3°C)/minute.

Vacuum Cure Cycle

Apply full vacuum, 24 in Hg (0.081 MPa) minimum.
 Heat from 75°F (24°C) to 260°F ± 10°F (127°C ± 5°C) at 5°F (3°C)/minute.
 Hold at 260°F (127°C) for 100 +10/-0 minutes.
 Cool under pressure to below 150°F (66°C) at 5°F (3°C)/minute.

Figure 1 | Recommended Lay-up