POLITECNICO DI TORINO

Department of Mechanical and Aerospace Engineering (DIMEAS)

Master of Science in Mechanical Engineering

Thesis of Master of Science

Experimental validation of topology optimization and lattice design for additive manufacturing of a polymeric bracket

Supervisor:

Prof. Paolo Minetola

Candidate:

Mutahir Qadeer

October 2018

To my Parents, without the support, drive and inspiration that they have given me, I might not be the person I am today.

POLITECNICO
DI TORINO

Author Mutahir Qadeer	
Title of Thesis Experimental validation of topology optimization and lattice design for additive manufacturing of a polymeric bracket.	
Degree Programme MSc Mechanical Engineering (Politecnico di Torino)	
Thesis supervisor Prof. Paolo Minetola	
Date 15/10/2018	**Language** English

Abstract

The thesis work discusses the structural optimization methodology for additive manufacturing – (Topology Optimization and Lattice Optimization) of the ALCOA Bracket (a component used in the aircraft industry) with the objective of reducing weight while full filling the design conditions.

The design process is explained starting with a static linear analysis of the component for analyzing the stress and then performing Topology optimization on the component with specific design constraints in order to reduce the weight of the component. The procedure is repeated separately on the same component by performing Lattice optimization on the component by using a combination of various Lattice beam length and diameter. After which a comparative study with the objective of reducing the weight of the component based on software results is carried out between the Topology Optimized part and Lattice Optimized part.

The last part of the thesis explains the experiment part in which Topology and Lattice optimized designs were printed on the 3d printer and were tested on the tensile testing machine and the test results obtained from the experiments were compared with the results obtained by software.

Keywords: Additive Manufacturing, Topology Optimization, Lattice Optimization

Preface

I would like to express my deep gratitude to my supervisor, Prof. Paolo Minetola for introducing me to the world of design for additive manufacturing, and whose expertise, understanding, support and patience, added considerably to my graduate experience. I would also like to thank Mr. Giovanni Marchiandi for his valuable time and technical support during my work in the laboratory.

Finally, I would like to acknowledge, Politecnico di Torino for providing me a profound environment for study and all the equipment needed for the research.

October 15, 2018

Mutahir Qadeer

Table of Contents

List of figures

List of Tables

Chapter 1 - Introduction

1.1 Background

Additive Manufacturing (AM), also popularly known as 3D printing is a layer-based manufacturing approach in which a complete three-dimensional part is fabricated by adding materials layer by layer. Due to this layer based additive approach, parts with higher geometrical complexity can be fabricated with ease and increase in complexity does not affect the cost of the process as in the case of conventional methods. This capability provides the designer with higher design freedom to optimize the part design towards physics of the problem for optimum performance rather than being limited by manufacturing constraints.

Recently, generative design has been introduced as an innovative approach to build up a 3D model. It has allowed the users to utilize the capabilities of additive manufacturing by producing an optimum model in accordance with design objectives. One of the software in the market is Solid Thinking Inspire which is the most inventive software in producing generative design of parts.

Solid Thinking Inspire reflects a 3D computational designing tool basing on Topology and Lattice optimization. The optimization process is conducted in accordance with the required specification such as product material, constraints and loading condition. The initial structure will transform into the ideal layout by analyzing the applied preference.

Solid Thinking Inspire provides engineers a shorter way to approach the design in the most efficient mechanism. This software can escalate the product optimization. The application of optimized design can be for prototyping of parts or fully functional end user parts which in comparison to others are light in weight and have an efficient design with better quality.

1.2 Objectives

The objective of this thesis is to describe a complete process, starting from the realization of the component to its design optimization in consideration of the main phases of additive manufacturing. The optimized structure will be fabricated using FDM technology and reproduced to perform the necessary test to verify the performance of the component to validate the virtual simulation.

The component that will be optimized is a product of Alcoa fastening systems & rings(AFSR). The airplane bearing bracket is a component originally made of metallic material and is used in control surfaces of various airplanes. In this thesis we will optimize the bracket using Solid Thinking Inspire software. Once the optimized design is obtained than it will be 3D printed by the FDM printing technology and later tested on a tensile testing machine.

The process of printing and testing will be repeated number of times for reproducibility and accuracy of experiment results. At last the results obtained from software and results obtained from tensile testing will be observed and analyzed.

The main objectives are as follows:

- Component realization
- FEM Analysis
- Topology and Lattice optimization
- 3D printing
- Tensile testing

CHAPTER 2 – Topology and Lattice Optimization

2.1 Introduction

After introducing theoretically, the main features of additive manufacturing and its benefits this chapter explains the logic and concept about Topology and Lattice optimization that will be applied on the ALCOA bracket using Solid Thinking Inspire software.

2.2 Topology optimization

The Topology Optimization is a numerical technique that allows users to optimize the shape of a mechanical component provided designated volume as design space is defined. The goal is to analyze and evaluate the optimum distribution of the material in the design space with respect to loads and constraints applied to it. Constraints can be geometric or functional such as displacement constraints, resistance constraints, or stiffness constraints. Through topology optimization it is possible to obtain innovative forms of the components with the lowest possible weight which exhibit desired performance in terms of rigidity while respecting manufacturing constraints.

Solid Thinking Inspire uses algorithms of Hyperworks solver Optistruct and does Topology optimization based on SIMP Theory – Solid Isotropic Material with Penalization.

The solid isotropic material with penalization also known as the density method or the power law method. The SIMP method does discretization of the structure provided the structure previously is composed of the mesh. It assigns a value $\rho = 0$ or 1 to all the elements in the mesh. This value represents the material density of that element. If the value assigned is $\rho = 0$ then it means that the element is assigned a 0% material density therefore the element will be empty. On contrary if the value assigned to it is $\rho = 1$ then it means that the element is assigned a 100% material density and the element will be full.

The goal of the optimization using SIMP theory is to assign at each iteration a density at each element and remove those elements with density equal to zero. The representation of the procedure is as illustrated in figure (2.1). Represented in red color are the parts that cannot be eliminated and those in green are the one that can be subtracted. It can be noted that the color configuration changes at each iteration and that some green areas in the first iteration have not been eliminated in the final solution.

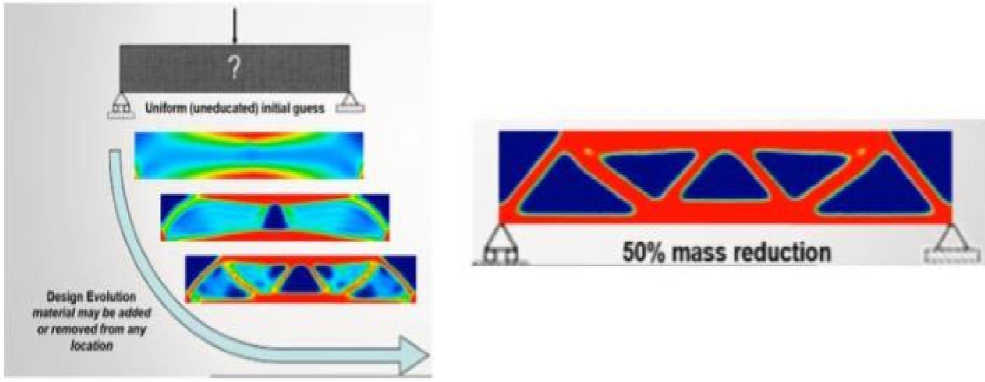


Figure 2. 1 SIMP Process

2.3 Lattice Optimization

The Lattice Optimization is a technique that allows users to optimize the shape of a mechanical component through Lattice structures provided designated volume as design space is defined. The goal is to analyze and evaluate the optimum placement of lattice structure in the material in the design space with respect to loads and constraints applied to it. Constraints can be geometric or functional such as displacement constraints, resistance constraints or stiffness constraints. Through Lattice Optimization it is possible to obtain innovative forms of the components with the lowest possible weight which exhibit desired performance while respecting manufacturing constraints.

Solid Thinking Inspire uses algorithms of Hyperworks solver Optistruct and does Lattice optimization based on algorithms of Optistruct. In regular topology optimization, the intermediate density elements are treated as fictitious material and are penalized into voids and pure solids. Whereas in lattice optimization, they are converted into lattice structures. During the topology optimization, the intermediate density elements are not penalized and are retained within the model whereas in lattice optimization the intermediate density elements are converted into lattice structures and the end diameters are sized based on a stress constraint for further finetuning. Lower and upper bound for intermediate densities and stress constraint values for lattice sizing need to be specified. Density values below the lower bound-(LB) will be converted into void and values above upper bound-(UB) will be converted into solids. Elements between LB and UB are converted into 1D simple beam elements (Type-Rod) and its diameter is proportional to the density of the intermediate density elements which were replaced.

Chapter 3 – Design Optimization

3.1 Introduction

After introducing the concept review about Topology and Lattice optimization in chapter number 2, This chapter explains the implementation of procedure and steps carried out to perform Topology and Lattice optimization on ALCOA bracket by using Solid Thinking Inspire software. The aim of optimization is to minimize the weight of the ALCOA bracket whilst full filling loading condition and constraints applied to it.

This chapter describes the details of the optimization procedure in the following topics:

- Component description
- FEM Analysis
- Topology Optimization
- Lattice Optimization

3.2 Component Description

The component that will be optimized is an ALCOA bracket, a component used in the aircraft industry as shown in figure (3.1).

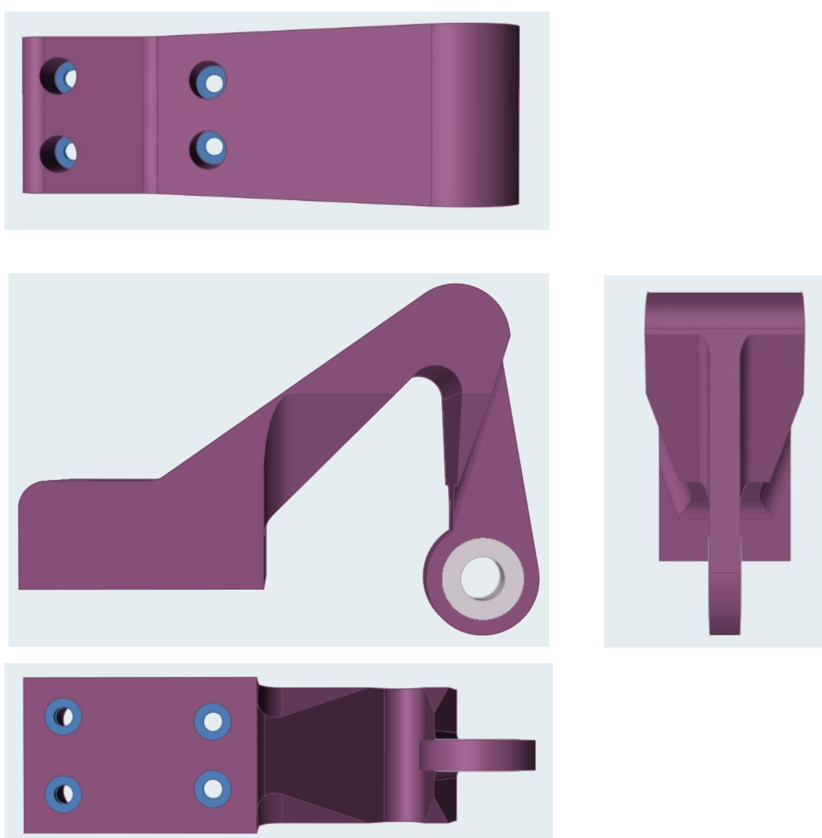


Figure 3. 1 ALCOA Bracket

The bracket has a complicated geometry with external dimensions in millimeters as (38x85x125). It is important to keep in consideration that the bracket will be 3D Printed using an FDM printer and later tested on the tensile testing machine, therefore the first step will be to analyze the C-Shape Bracket on which the component will be fastened for tensile testing. Figure (3.2) shows the structure of the C-Shape Bracket on which the component will be fastened for testing on a tensile testing machine.

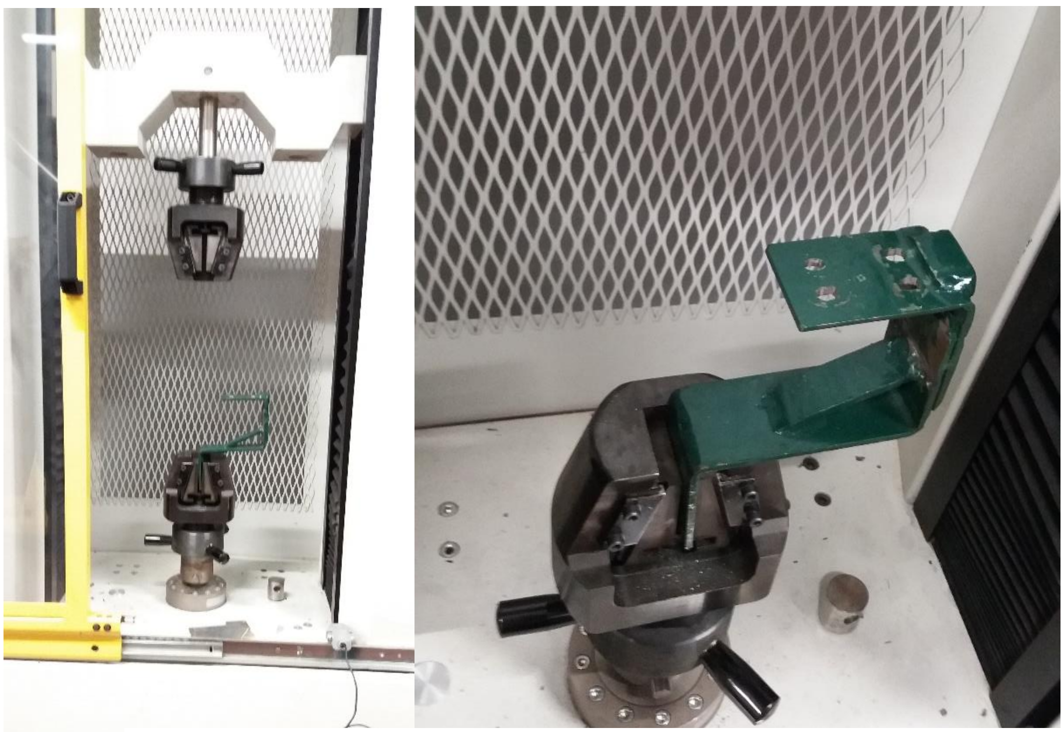


Figure 3. 2 C-Shape Bracket

3.3 FEM (Finite Element Model)

3.3.1 Introduction

We need to evaluate the stresses inside the component before doing Topology and Lattice optimization of the component. Doing a structural analysis before optimization enables us to identify the maximum applicable load on the initial component in consideration of loads and constraints imposed on it.

The structural analysis of the bracket is solved using a finite element method and carried out in Solid Thinking Inspire software. Other than structural analysis the software also provides extensions for doing Topology and Lattice optimization. To conclude it can be said that Solid Thinking Inspire is all in one software that allows users to do analysis and optimization in the same working environment.

3.3.2 FEM Analysis in Inspire

The first step is to import the CAD model of the bracket in the working environment of the software. Figure (3.3) shows the main layout of the software. It can be clearly observed that at the top there is a toolbar which includes functions grouped under heading: edit, geometry, structure, motion and manufacture.

The most important module for us now is the structure module as it contains the functions necessary to set loads and constraints for the FEM analysis and also for Topology and Lattice optimization analysis that will be carried out later.

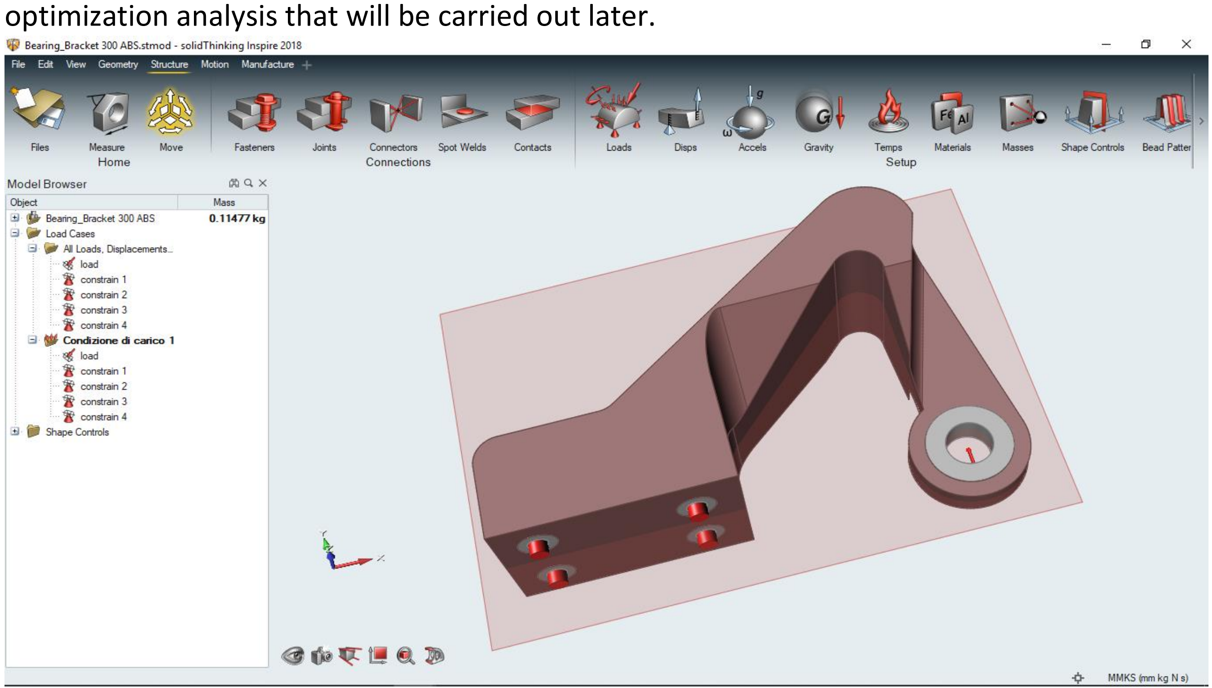
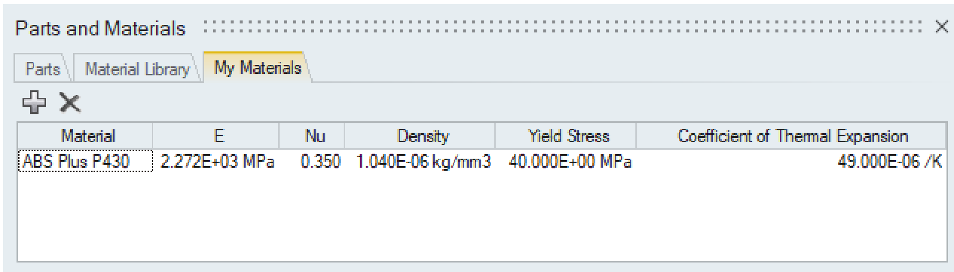


Figure 3. 3 Solid Thinking Inspire layout

After importing the CAD in software, we will assign the material to the geometry. The material assigned is ABS Plus P-430. Figure (3.4) illustrates the material properties of ABS Plus P-430.



Material	E	Nu	Density	Yield Stress	Coefficient of Thermal Expansion
ABS Plus P430	2.272E+03 MPa	0.350	1.040E-06 kg/mm3	40.000E+00 MPa	49.000E-06 /K

Figure 3. 4 Material properties

Now we proceed by defining Loads and Constraints. Before defining loads and constraints it is important to consider that all Topology and Lattice optimized components will be manufactured by using 3D Printing and later will be tested on the tensile testing machine, therefore keeping the fact in consideration the regions of the component where loads and constraints are applied will remain the same throughout. The bracket will be pulled along Y-Axis and will be constrained through four holes of the base.

Figure (3.5) shows the region of the bracket where load and constraints were applied.

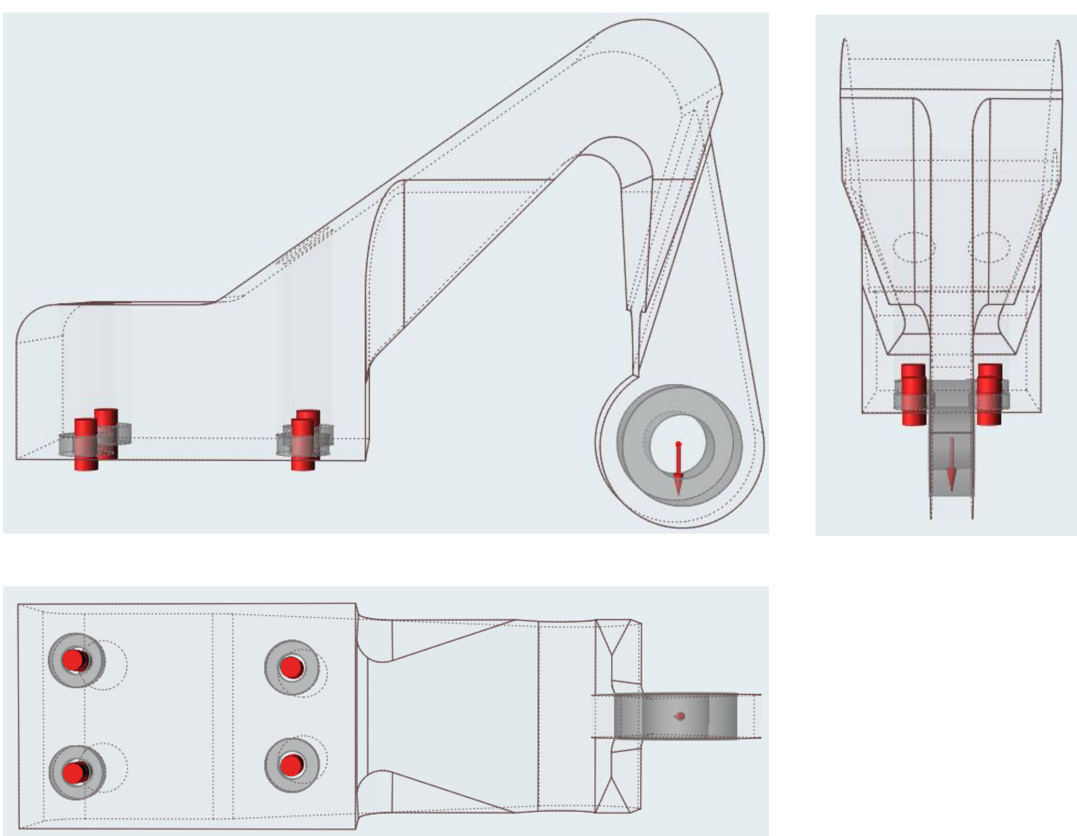


Figure 3. 5 Bracket regions

3.3.3 FEM Analysis Result

Once the load and constraints are applied to the model we can proceed with the FEM Analysis by defining the parameters required to proceed with the simulation. Figure (3.6) shows the example of input parameter required for the simulation.

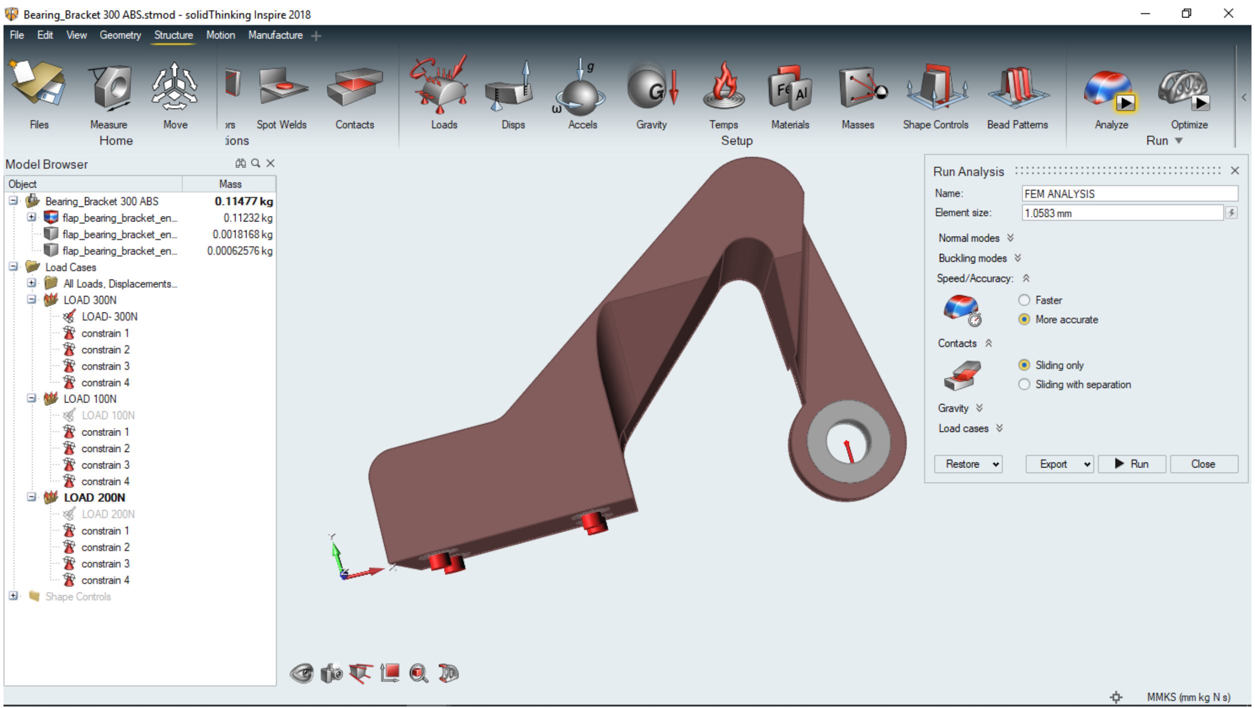


Figure 3. 6 Simulation input parameter

After defining the parameters, we can proceed with the simulation. Figure (3.7) shows the result of the FEM analysis of the bracket subjected to a load of 300 N.

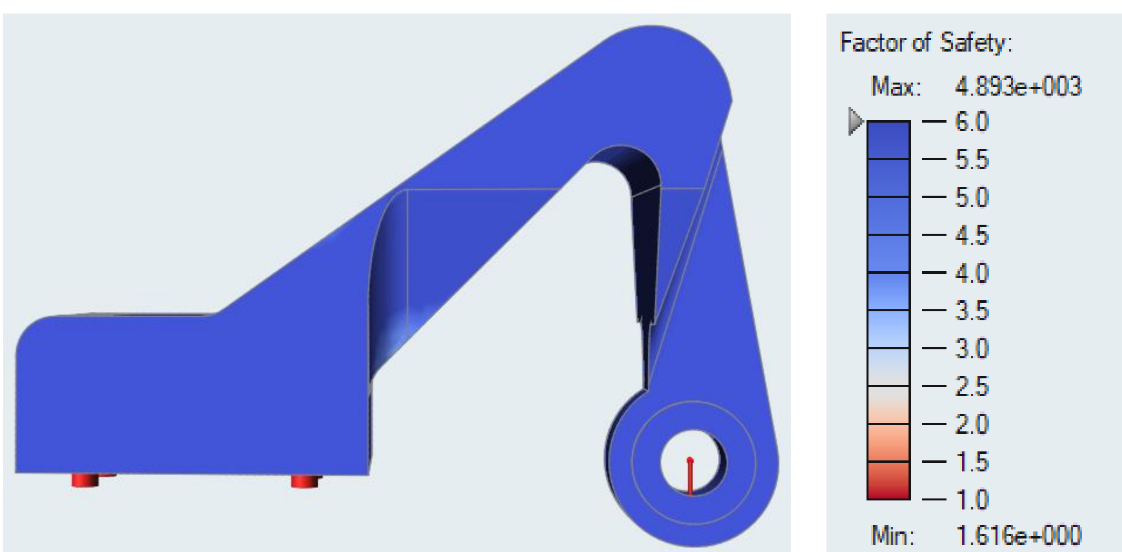


Figure 3. 7 FEM Result

The FEM analysis was conducted with various load cases to observe the region with the lowest Safety Factor. The critical regions of the bracket are as illustrated in figure (3.8).

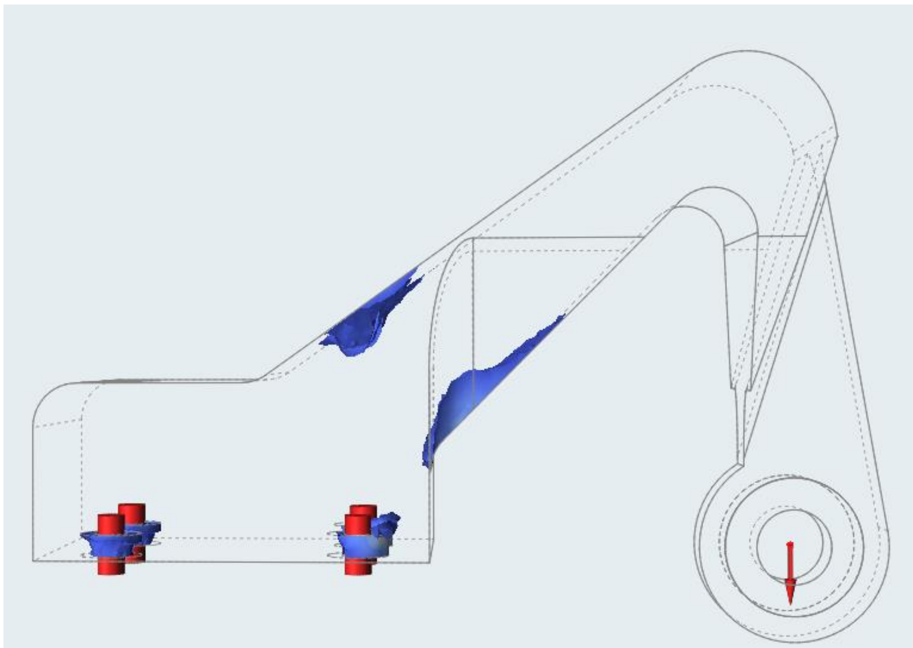


Figure 3. 8 Bracket Critical Regions

3.4 Topology Optimization

As explained in chapter number 2 the concept review about Topology Optimization this topic explains the procedure to perform Topology Optimization on the bracket using Solid Thinking Inspire software.

3.4.1 Topology Optimization in solid thinking inspire software

The procedure followed to perform Topology Optimization is as follows:

1. Definition of optimization regions (Design and Non-Design space)
2. Definition of shape constraints
3. Definition of optimization goals
4. Simulation and calculation

Definition of optimization regions (Design and Non-Design space)

The first step is to differentiate the regions of the bracket which we want to optimize and the one which we do not want to optimize. It is necessary to differentiate the regions because the regions on which we will apply load and constraints cannot be optimized and will not be changed by the software, therefore the regions on which we want to perform topology optimization are stated as Design space and the regions on which we will apply load and constraints will be stated as non- Design space. The regions that will not be optimized by the software and will remain unchanged (non- Design space) for Topology optimization are as follows:

1) The region in the vicinity of the 4 holes which will be kept intact with the support and constrained in all direction.
2) The region in the vicinity where the load will be applied

Figure (3.9) shows the design space and non-design space of the bracket. The maroon color is for design space and grey color is for non-design space.

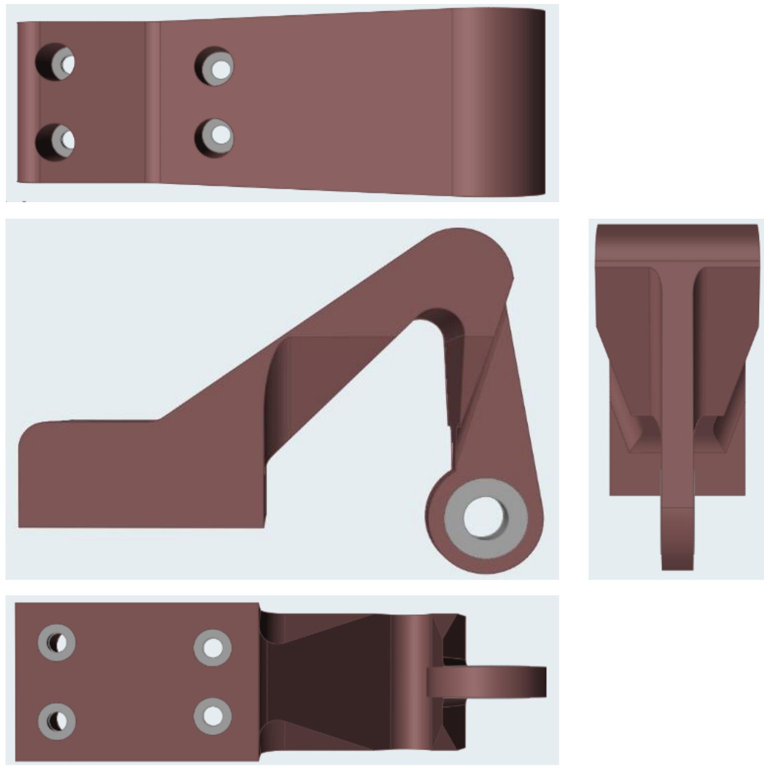


Figure 3. 9 Design and Non-design space of Bracket

Once define the design and non-design space it is now clear that only the design space will be optimized, and the non-design space will remain unchanged.

Definition of shape constraints

Shape constraints in Inspire software are basically fabrication constraints that are present in various manufacturing processes. In our case study we will use additive manufacturing technology by 3D Printing the Topology Optimized bracket. The use of additive manufacturing for the production of topology optimized bracket is advantageous as compared to conventional manufacturing technologies in terms of manufacturing constraints, therefore we will proceed by defining the form of constraining that exhibit the symmetry of the bracket. The symmetry is set with respect to the X-Axis and will remain unchanged for all topology optimized components as illustrated in figure (3.10).

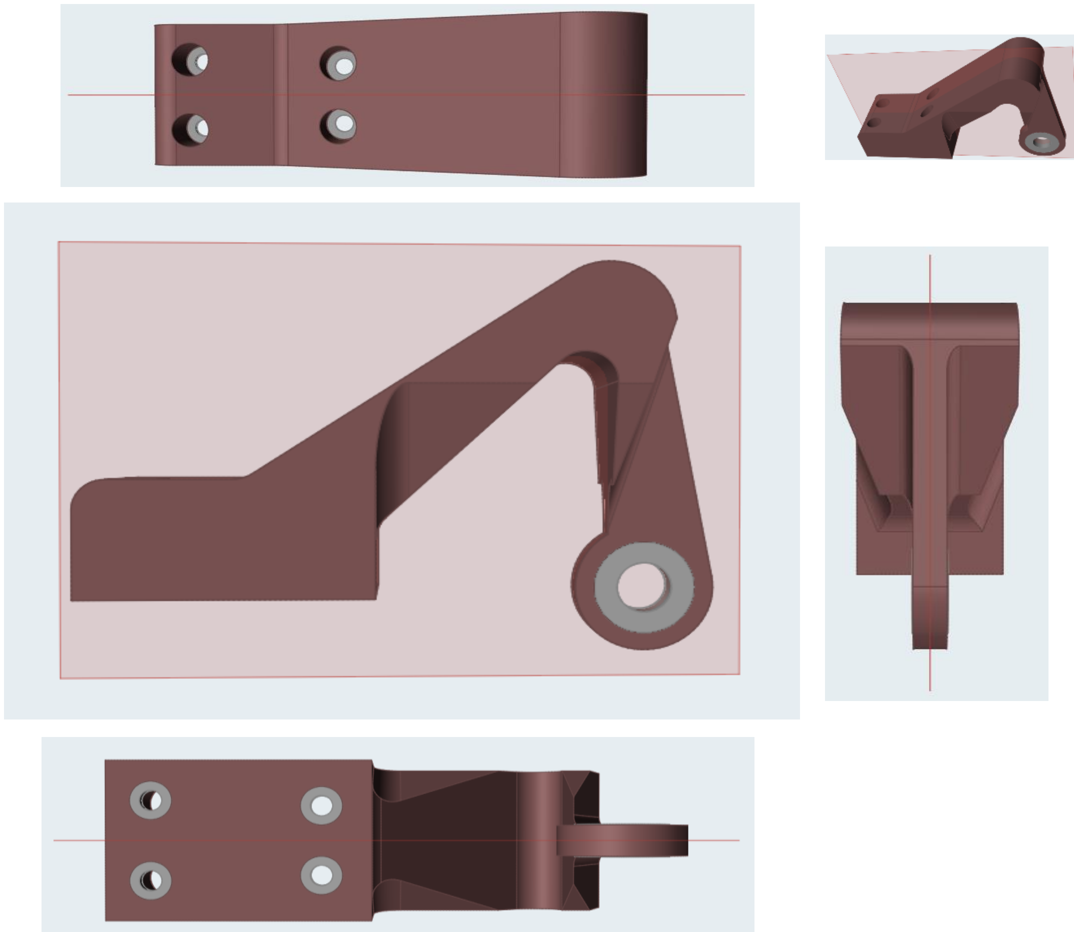


Figure 3. 10 Symmetry Constraint of Bracket

Applying a symmetry plane to the bracket is advantageous as the load applied on the bracket will be equally distributed. It also reduces the computation time and gets faster results from the computation thus allowing a more robust structure by giving a greater resistance and stability to the optimized piece.

Defining optimization goals

The third step is to define the objectives of the analysis. Figure (3.11) shows the parameters required to do simulation for topology optimization.

In order to perform the topology optimization provided the load and constraints are applied to the model, the first thing is to define the objective of the optimization. As per the scope of the thesis the objective of our optimization is to minimize mass.

Second thing to define is the parameter of minimum safety factor for which we want to Topology optimize the component. Other parameters include minimum and maximum thickness constraints for the optimization.

In this example, the component was topology optimized for a minimum safety factor of 1.5 with minimum thickness constraints of 5 mm.

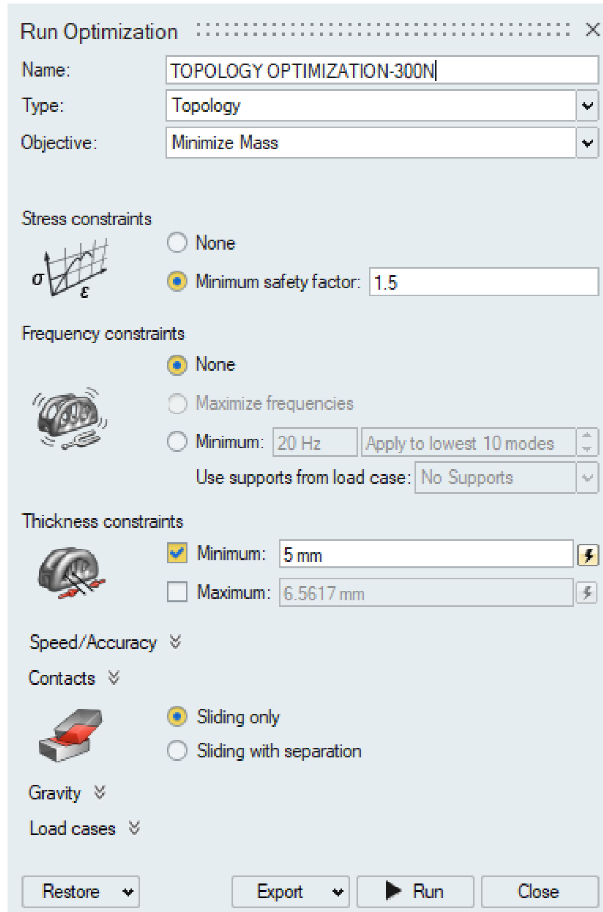


Figure 3. 11 Simulation parameters for Topology optimization

Simulation and Calculation

Once Definition of optimization regions (design and non-design space), Definition of shape constraints and Definition of optimization goals are defined we can proceed with the simulation for topology optimization of the component. Following as shown in figure (3.12) is the example of simulation performed in order to topology optimized the component with the objective of minimizing the weight of the component. The material of the component is ABS plus P40 and the component was subjected to a load of 100 N.

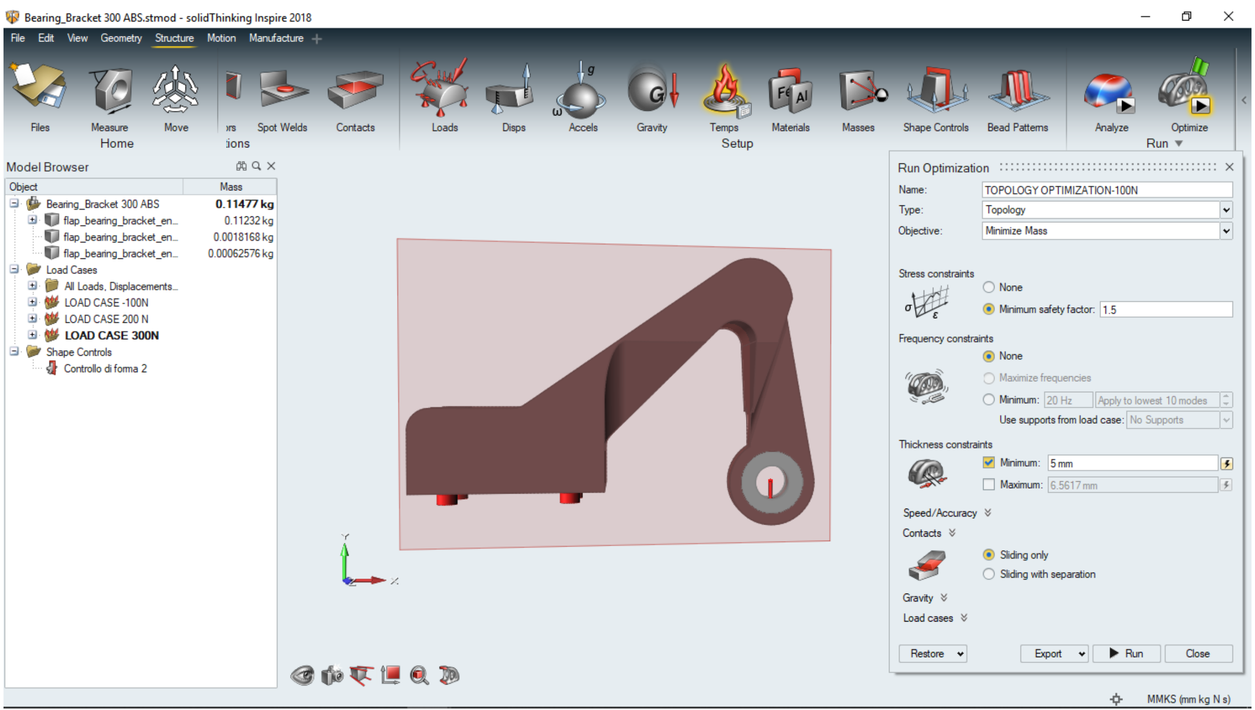


Figure 3. 12 Simulation

The results obtained after the completion of the simulation is as shown in figure (3.13) below. It can be clearly observed that the material has only been removed from the region assigned as design space and the regions assigned as non-design space remains unchanged.

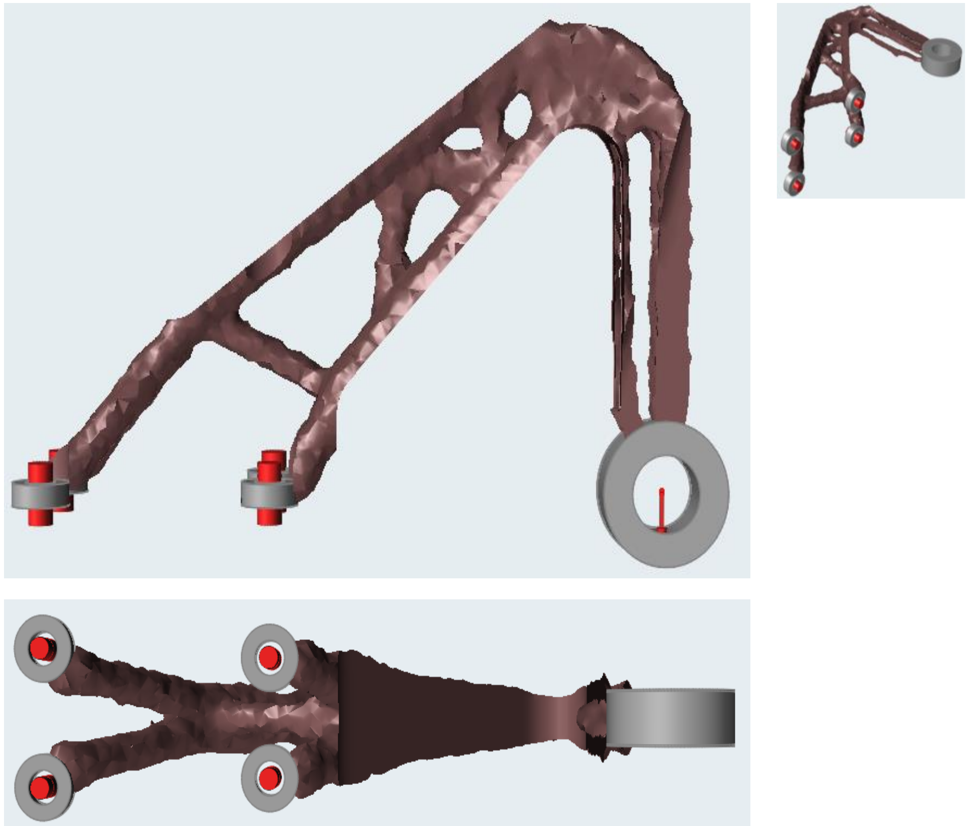


Figure 3. 13 Topology optimized result

Once the topology optimized model is attained a further investigation on the topology optimized model is carried out by performing FEM Analysis on it to evaluate the factor of safety of the component. Figure (3.14) shows the FEM analysis of the optimized component.

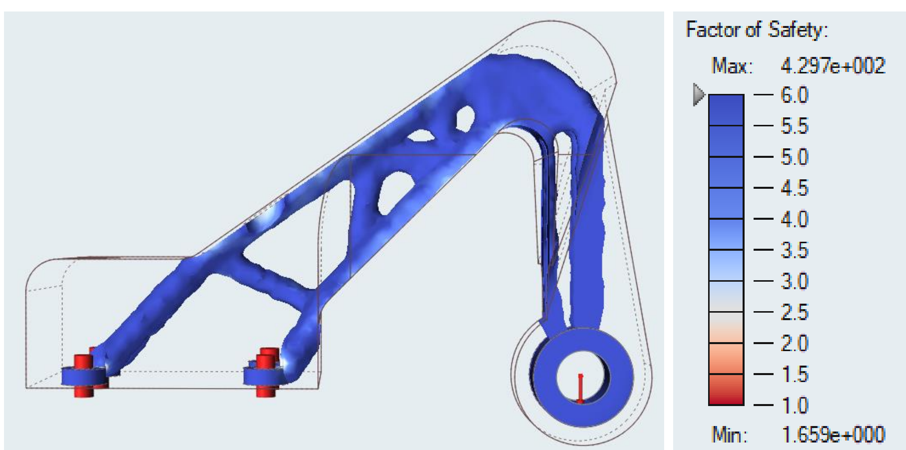


Figure 3. 14 FEM Analysis Result

3.4.2 Topology optimization case study

The intent of doing topology optimization of the bracket is to evaluate the reliability of the software, therefore various simulation with different load cases and a factor of safety were performed to best know the scenario.

After performing several simulations and having a range of results it was decided to select the simulations performed with FS = 1.5 for load cases 100 N, 200 N and 300 N. The selected cases are as follow in Table (3.1).

Table 3. 1 Topology Optimization Cases

Case	Description
Case A	Includes the bracket optimized with FS = 1.5 and load of 100 N.
Case B	Includes the bracket optimized with FS = 1.5 and load of 200 N.
Case C	Includes the bracket optimized with FS = 1.5 and load of 300 N.

The results of the Topology optimized model with FEM analysis of Case A, Case B and Case C after optimization are as follow:

CASE A

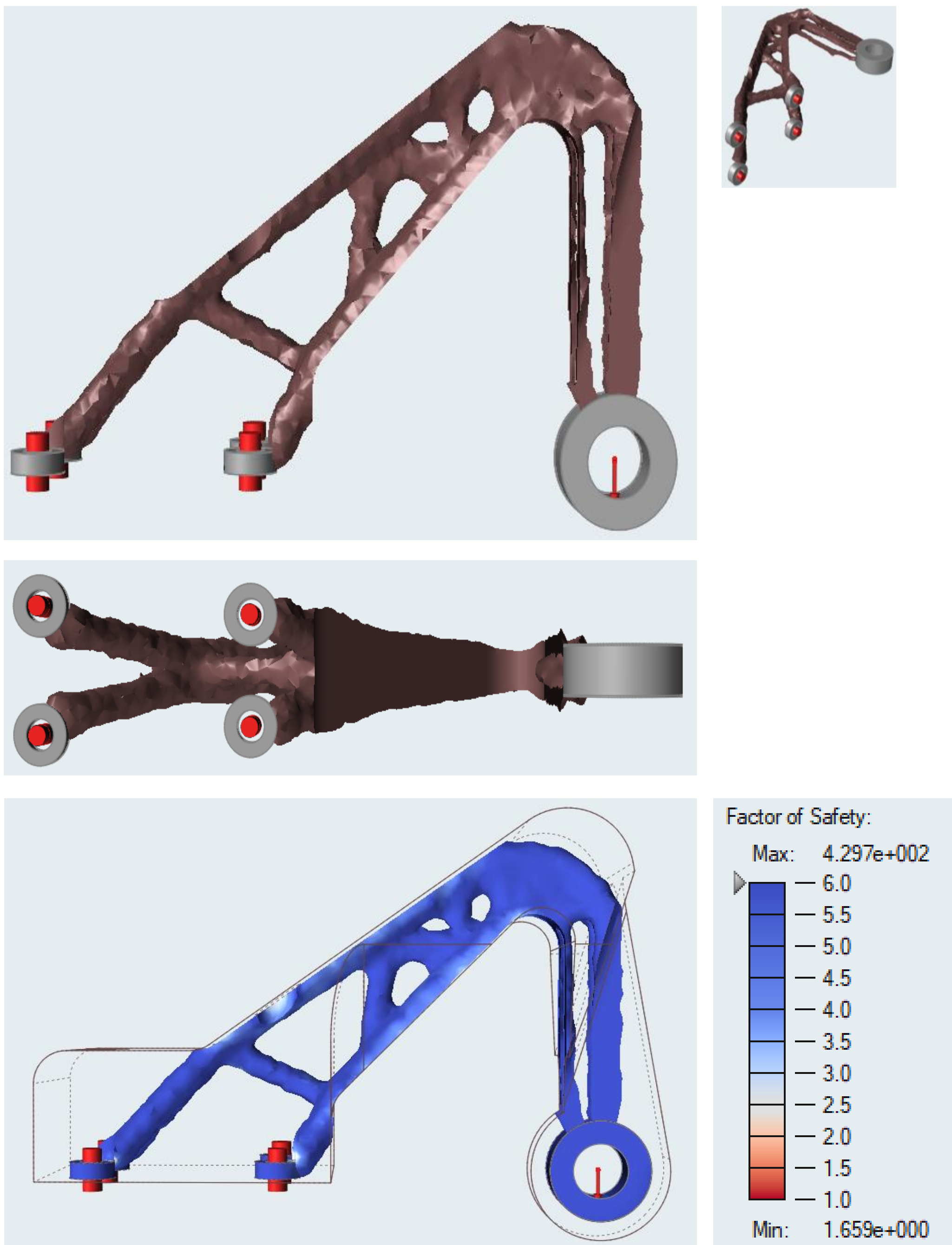


Figure 3. 15 Case A Results

CASE B

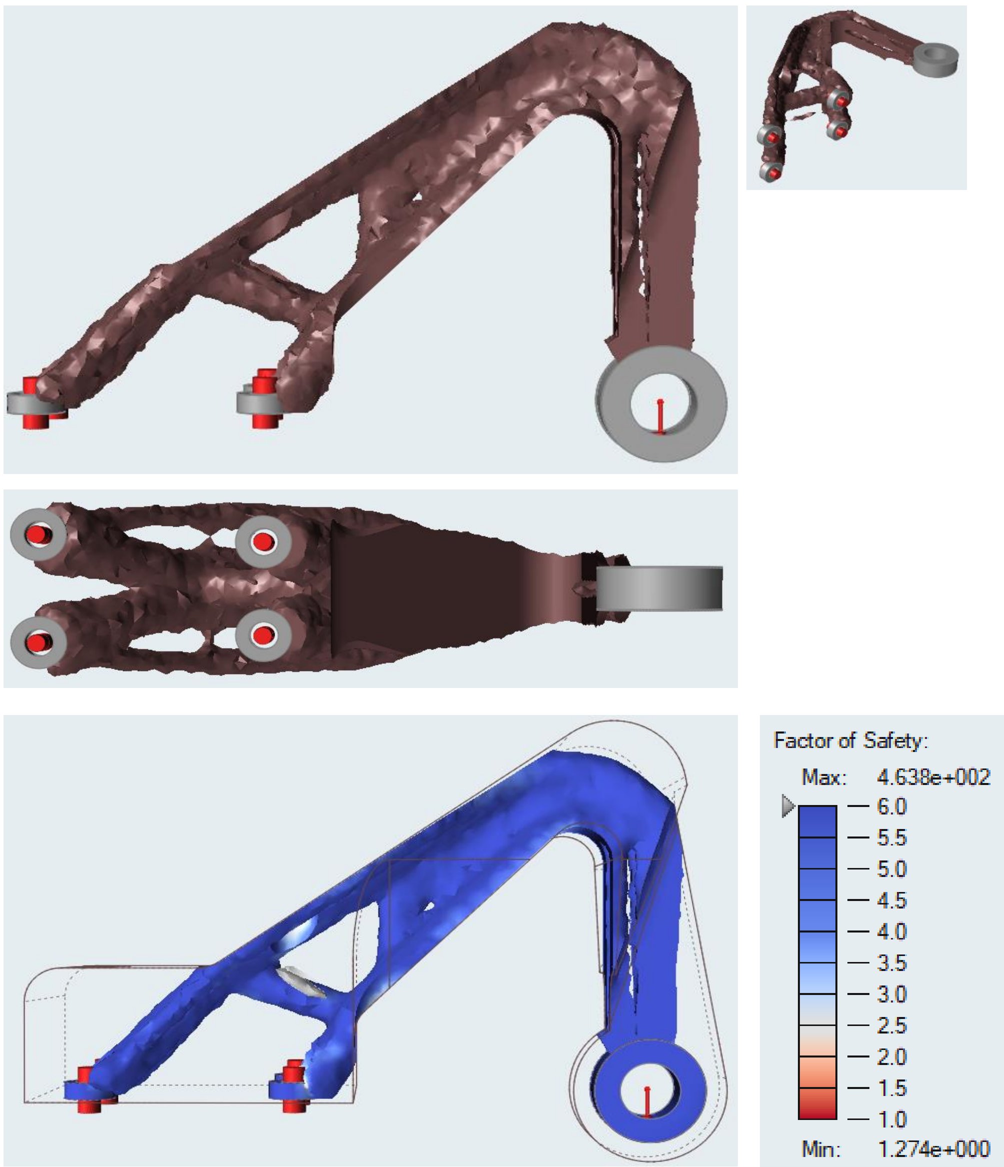


Figure 3. 16 Case B Result

CASE C

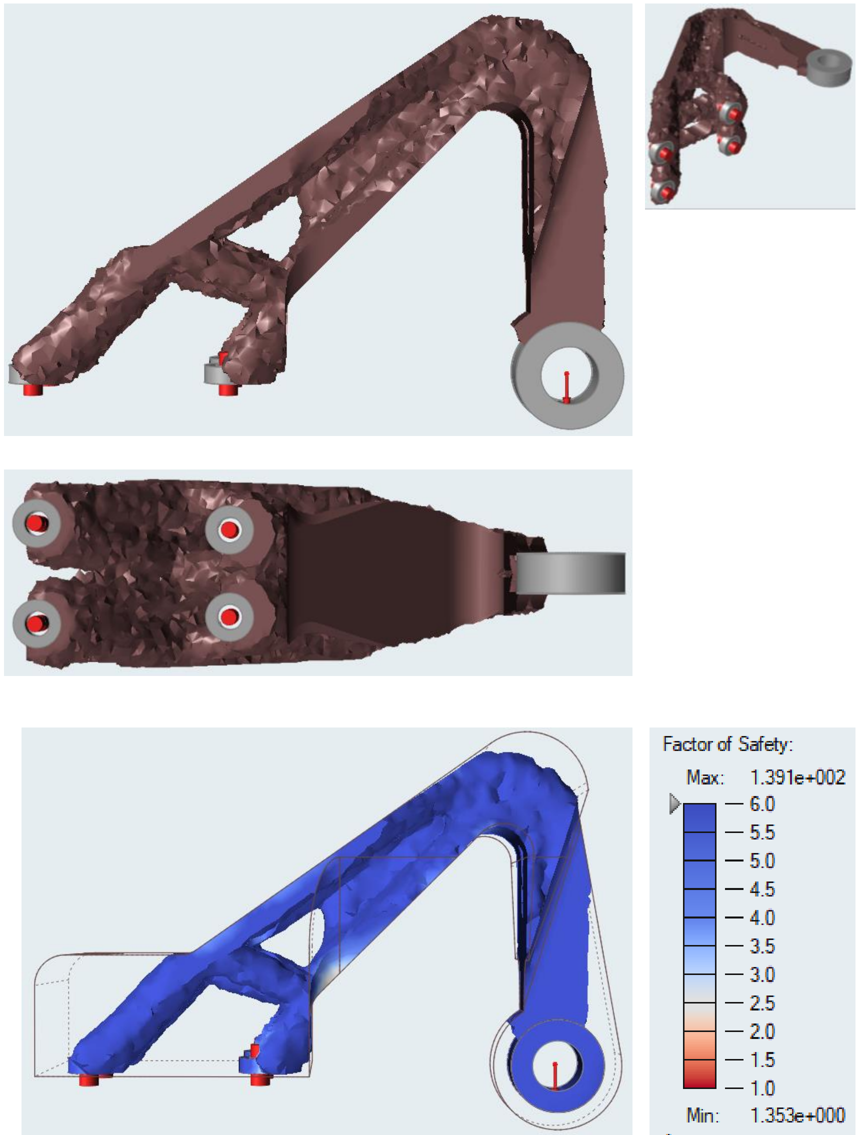


Figure 3. 17 Case C Result

The results of the Case A, Case B and Case C are presented in table (3.2).

Table 3. 2 Topology Optimization Case Result

Case	Load (N)	(FS) simulated	Inspire Mass (grams)	Mass reduction (%)
A	100	1.659	9.74	94
B	200	1.274	20.80	84
C	300	1.353	38.72	70

3.5 Lattice optimization

As explained in chapter number 2 the concept review about Lattice optimization this topic explains the procedure to perform Lattice optimization on the bracket using Solid Thinking Inspire software.

3.5.1 Lattice optimization in solid thinking inspire software

The procedure followed to perform Lattice optimization is as follows:

1. Definition of optimization regions (design and non-design space)
2. Definition of shape constraints
3. Definition of optimization goals
4. Simulation and calculation

Definition of optimization regions (design and non-design space)

The first step is to differentiate the regions of the bracket which we want to optimize and the one which we do not want to optimize. It is necessary to differentiate the regions because the regions on which we will apply load and constraints cannot be optimized and will not be changed by the software, therefore the regions on which we want to perform lattice optimization are stated as design space and the regions on which we will apply load and constraints will be stated as non- design space.

As evident from the FEM analysis and later stated in topic lattice optimization case study it was decided to do lattice optimization in two phases. In the first scenario the regions with low factor of safety will be assigned as design space and the rest will be assigned as non-design space as shown in figure (3.18).

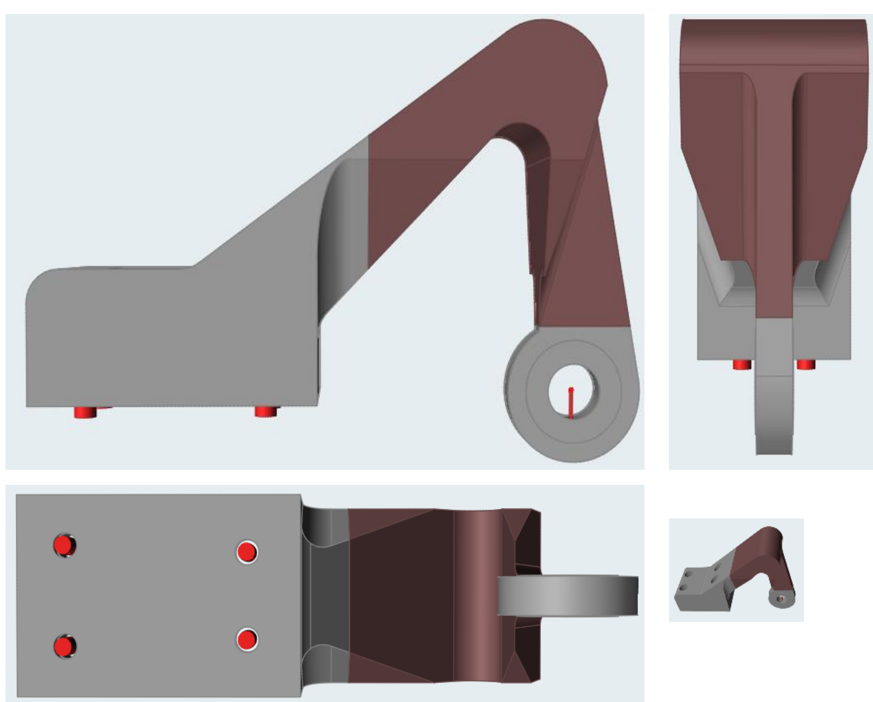


Figure 3. 18 Design and Non-design space first scenario

In the second scenario, the maximum regions of the bracket will be assigned design space and the remaining regions where force and constraints are applied will be non-design space as shown in figure (3.19).

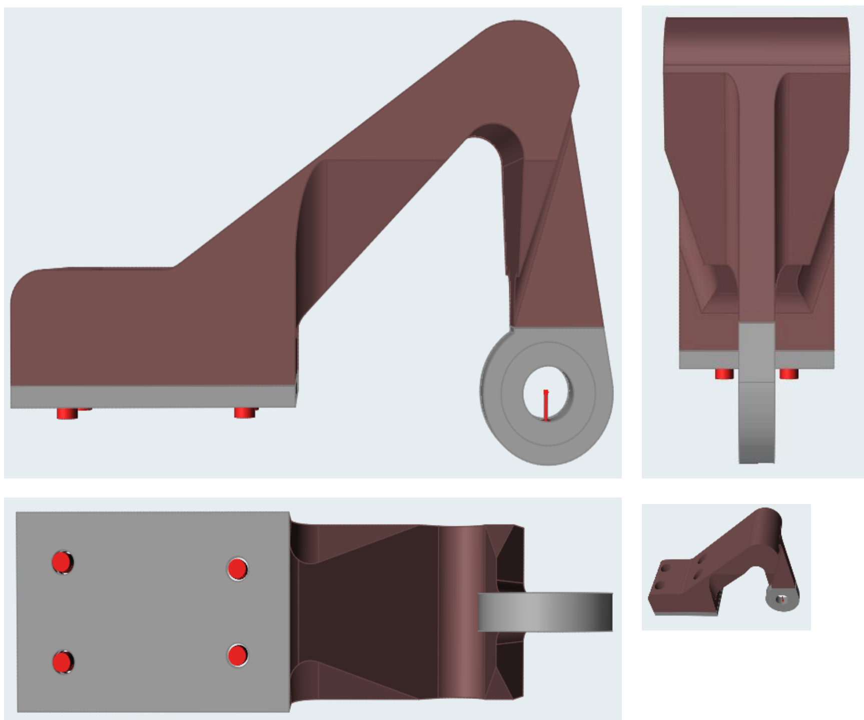


Figure 3. 19 Design and Non-design space second scenario

Once defined the design and non-design space for the above stated scenarios, it is now evident that only the design space will be optimized, and the non-design space will remain unchanged.

Definition of shape constraints

Shape constraints remains the same as that defined in Topology optimization. The symmetry constraint is set with respect to the x-axis and will remain unchanged for both the scenarios stated in Definition of optimization regions. Figure (3.20) and figure (3.21) shows symmetry constraint applied to the bracket.

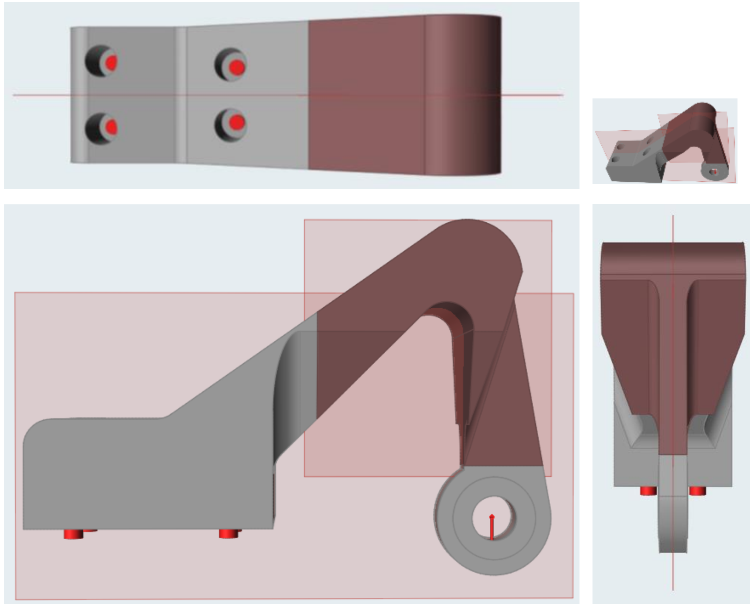


Figure 3. 20 Symmetry constraint first scenario

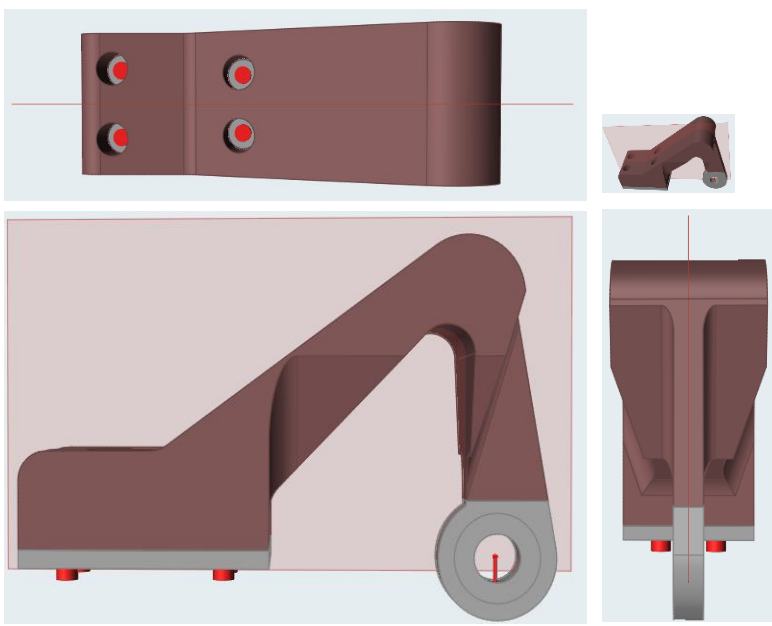


Figure 3. 21 symmetry constraint second scenario

Defining optimization goals

The third step is to define the objectives of the analysis. Figure (3.22) shows the parameters required to do simulation for lattice optimization.

To perform the lattice optimization provided the load and constraints are applied to the model, the first thing is to define the objective of the optimization. As per scope of the thesis the objective of our optimization is to minimize mass.

Second thing to define are the parameters related to dimensions of the Lattice – (length, minimum diameter and maximum diameter). We also need to define the percentage of lattice that needs to be filled in design space. In this example lattice size selected was (7-2-4) with 100% lattice fill in design space.

Other parameters include stress constraints, speed/accuracy and gravity.

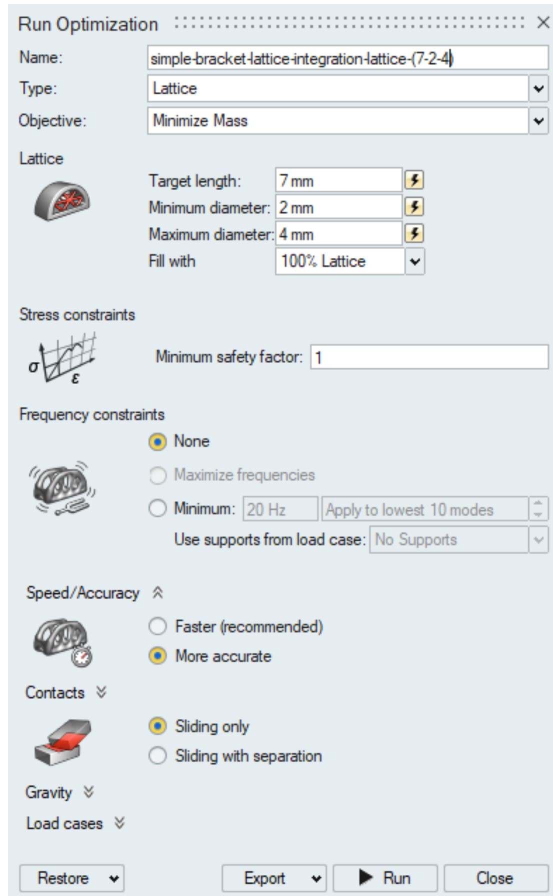


Figure 3. 22 Input parameters for lattice optimization

Simulation and calculation

Once Definition of optimization regions (design and non-design space), Definition of shape constraints and Definition of optimization goals are defined we can proceed with the simulation for Lattice optimization of the component. Following as shown in figure (3.23) is the example of simulation performed to do lattice optimization of the component with the objective of minimizing the mass of the component. The material of the component is ABS plus P40. The component was subjected to a load of 300 N and the lattice size selected for the optimization is (7-2-4) with 100% lattice fill in design space.

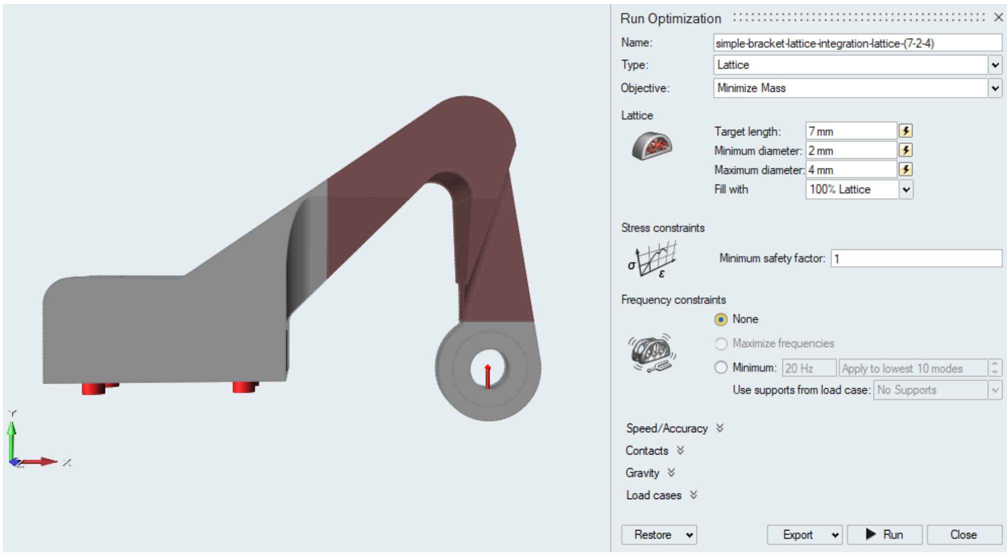


Figure 3. 23 Simulation for lattice Optimization

Solid Thinking Inspire software produce the Lattice optimization in the form of FEM analysis. The FEM results obtained after the completion of the simulation is as shown in figure (3.24). It can be clearly observed that the lattice optimization has only been carried out in the region assigned as design space and the regions assigned as non-design space remains unchanged.

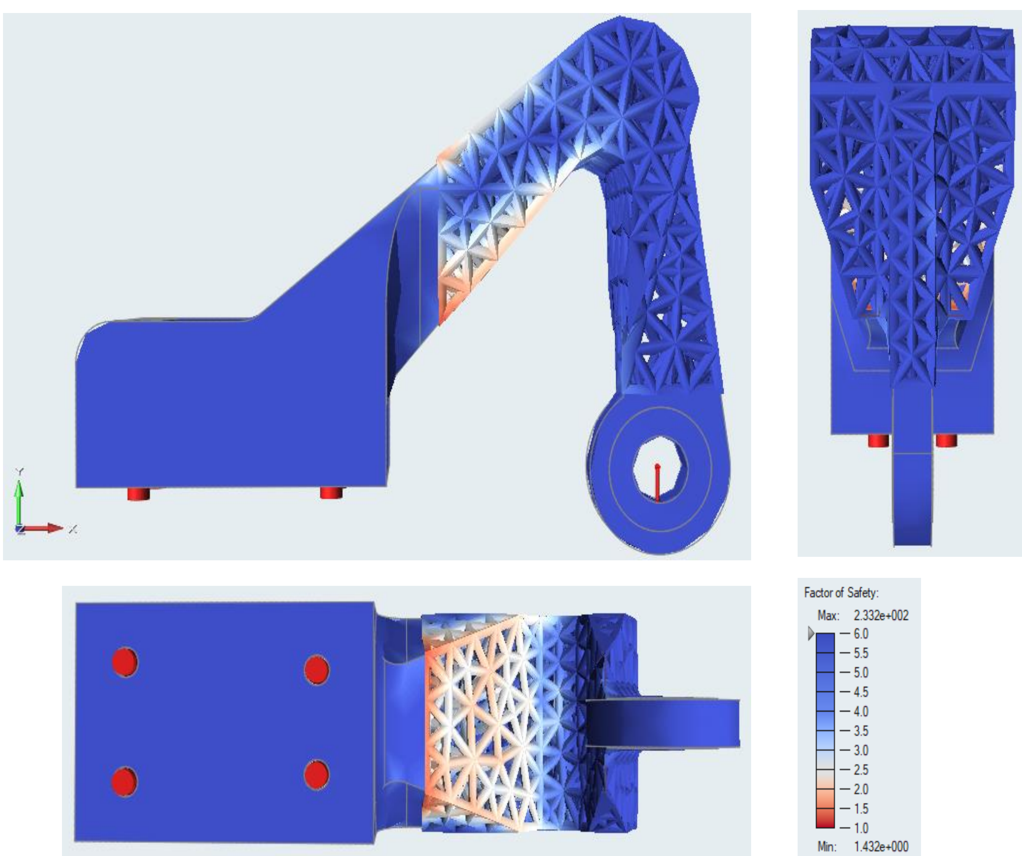


Figure 3. 24 lattice optimization FEM result

3.5.2 Lattice optimization case study

The intent of doing Lattice optimization of the bracket is to evaluate the reliability of the software, therefore various simulations with different lattice size were performed keeping in consideration the two scenarios defined in topic - Definition of optimized regions (design and non-design space).

After performing several simulations and having a range of results it was decided to select the simulations as illustrated in Table (3.3) for validating the results of the software.

Table 3. 3 Lattice Optimization Cases

Case	Description
Case D	Includes lattice optimization of the bracket for a load of 300 N with lattice size (7-2-4) for (design / non-design space) of scenario 1
Case E	Includes lattice optimization of the bracket for a load of 300 N with lattice size (5-2-4) for (design / non-design space) of scenario 1
Case F	Includes lattice optimization of the bracket for a load of 300 N with lattice size (7-2-4) for (design / non-design space) of scenario 2
Case G	Includes lattice optimization of the bracket for a load of 300 N with lattice size (5-2-4) for (design / non-design space) of scenario 2

The optimized models along with FEM analysis after lattice optimization of the case studies are as follows:

CASE D

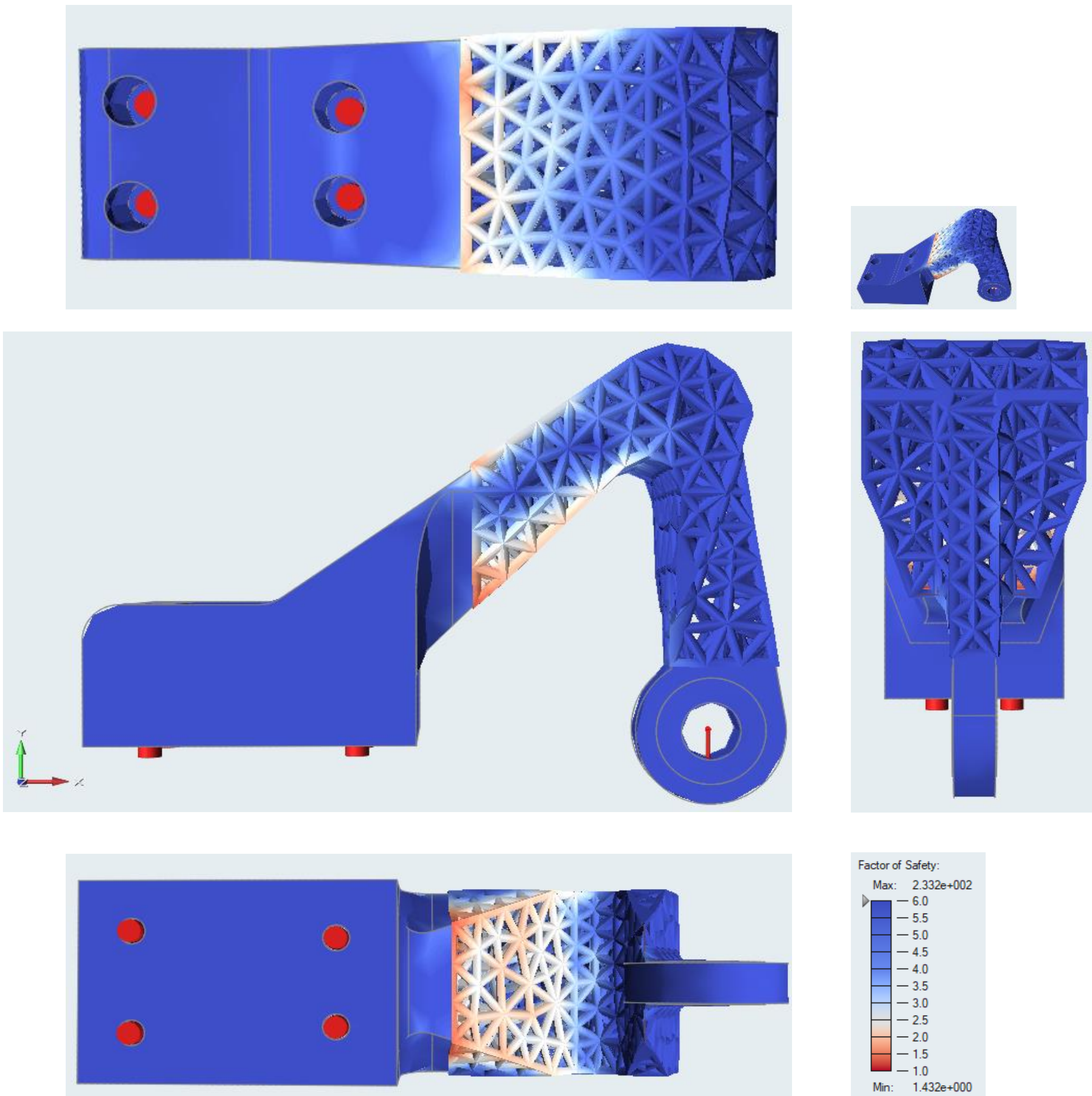


Figure 3. 25 Case D Result

CASE E

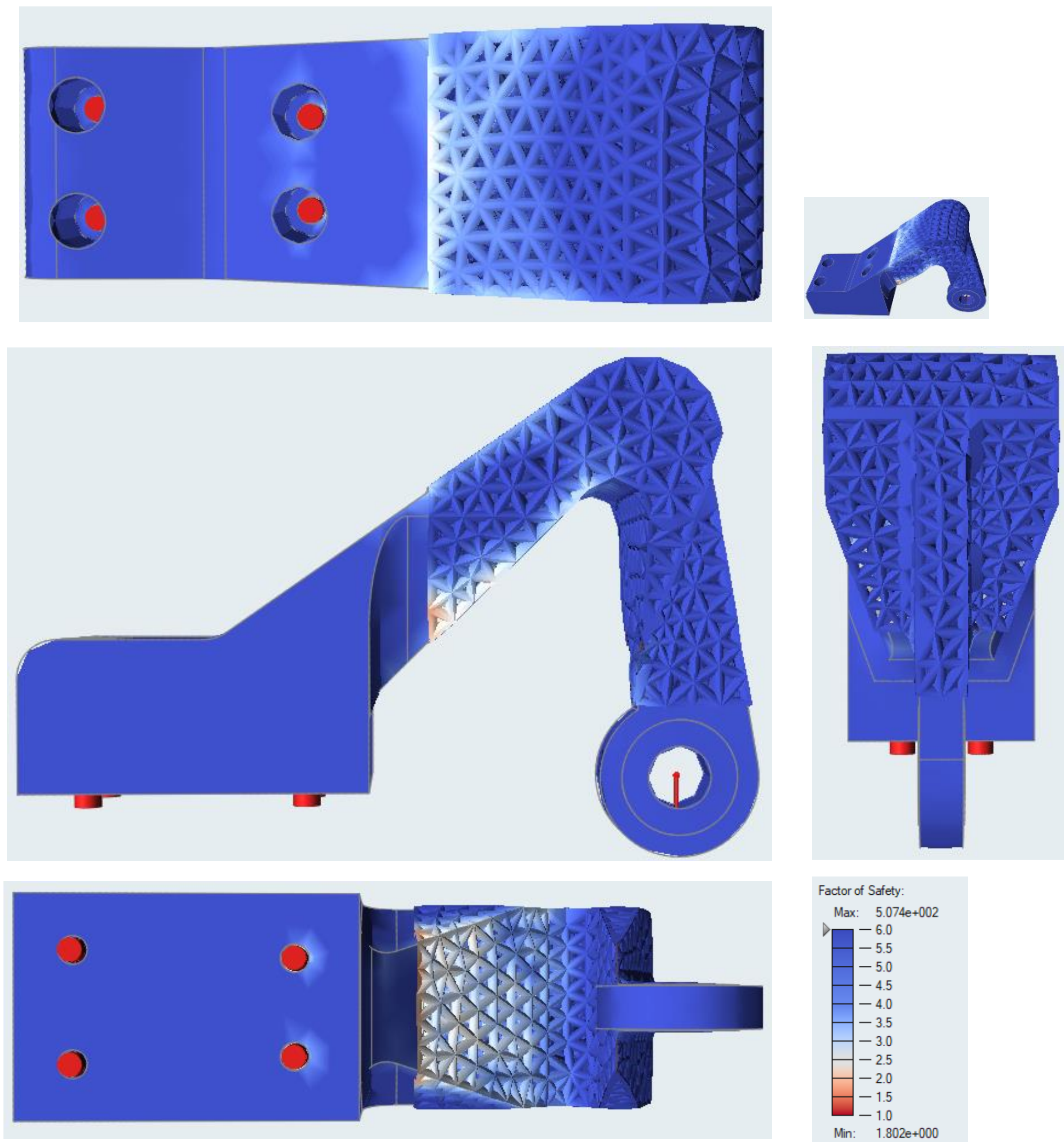


Figure 3. 26 Case E Result

CASE F

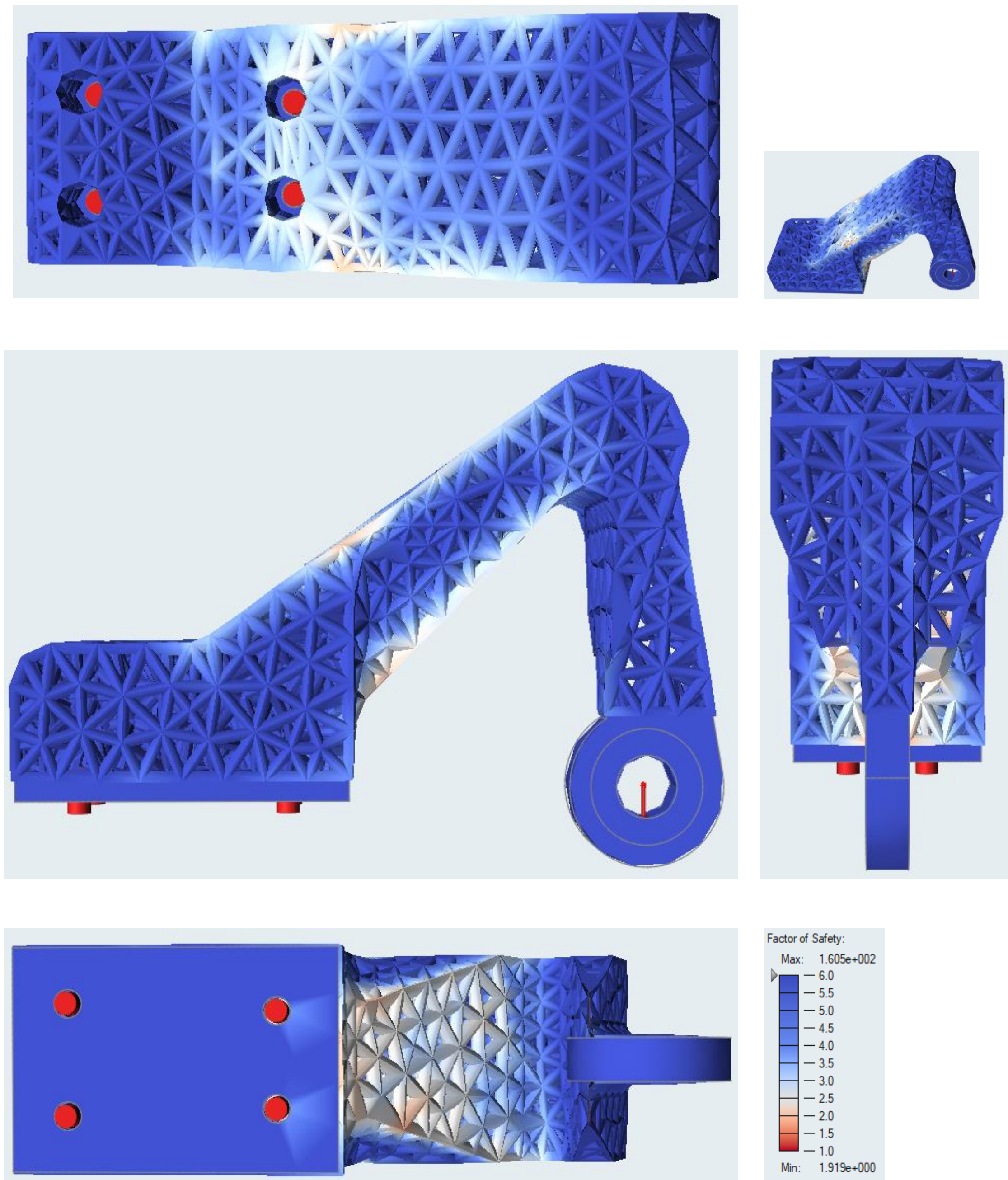


Figure 3. 27 Case F Result

CASE G

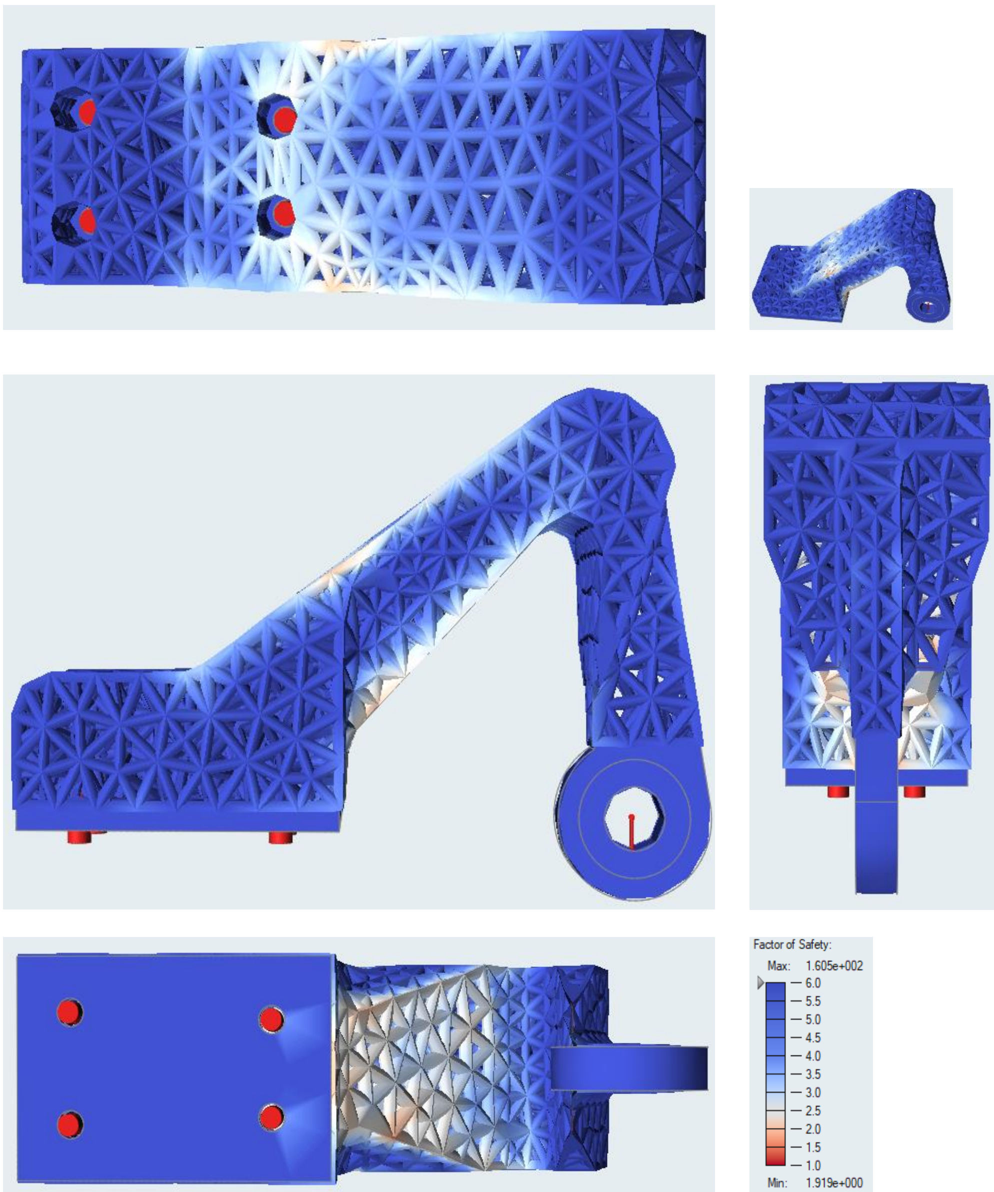


Figure 3. 28 Case G Result

The results of the Case D, Case E, Case F and Case G are summarized in table (3.4).

Table 3. 4 Lattice Optimization Results

Case	Load (N)	(FS) simulated	Inspire Mass (grams)	Mass reduction (%)
D	300	1.4	86	35.3
E	300	1.8	101	24
F	300	1.9	69	48.1
G	300	1.9	71	46.6

CHAPTER 4 – AM MANUFACTURING PROCESS

4.1 Introduction

This chapter includes the details of additive manufacturing production of the optimized components. The optimized components that will be manufactured using additive manufacturing technology are the cases of Lattice Optimization developed in Chapter number 3. The topology optimized components were already manufactured in the laboratory therefore were not manufactured again and were analyzed as-built. The previously manufactured topology optimized component and the developed lattice optimization component that will be 3D printed from now onward will be called as specimens. The details of the cases and their corresponding 3D Printed specimens are as illustrated in table (4.1).

Table 4. 1 Specimens

Case study	Description	3D printed specimens
Case A	Topology Optimization for 100 N load	Specimen A
Case B	Topology Optimization for 200 N load	Specimen B
Case C	Topology Optimization for 300 N load	Specimen C
Case D	Lattice Optimization for scenario 1 - Lattice size-(7-2-4)	Specimen D1 Specimen D2
Case E	Lattice Optimization for scenario 1 - Lattice size-(5-2-4)	Specimen E1 Specimen E2
Case F	Lattice Optimization for scenario 2 - Lattice size-(7-2-4)	Specimen F
Case G	Lattice Optimization for scenario 2 - Lattice size-(5-2-4)	Specimen G

4.2 Printer and Printing Material

The machine that will be used for printing with FDM Technology is a Stratasys F-370 printer that belongs to the group of F123 series of Stratasys Printers.

The printing process involves a series of steps that build the workpiece by depositing material layer upon layer. The printer has two movable extruders and the thickness of the layers in accordance with the material used varies between 0,127 mm and 0,330 mm with an accuracy of +/- 0.200 mm. Figure (4.1) shows the technical specification of the machine.

PRODUCT SPECIFICATIONS					
System Size and Weight	1626 x 864 x 711 mm (64 x 34 x 28 in.) 227 kg (500 lbs) with consumables				
Noise Specification	46 dB maximum during build, 35 dB when idle				
Model Capabilities		Stratasys F170	Stratasys F270	Stratasys F370	
	Maximum Build Size (XYZ)	254 x 254 x 254 mm (10 x 10 x 10 in.)	305 x 254 x 305 mm (12 x 10 x 12 in.)	355 x 254 x 355 mm (14 x 10 x 14 in.)	
	Model Materials	PLA*, ABS-M30™, ASA, QSR support material	PLA*, ABS-M30, ASA, QSR support material	PLA*, ABS-M30, ASA, PC-ABS, QSR support material	
Layer Thickness		0.013 in. (0.330 mm)	0.010 in. (0.254 mm)	0.007 in. (0.178 mm)	0.005 in. (0.127 mm)
	PLA		X		
	ABS	X	X	X	X
	ASA	X	X	X	X
	PC-ABS	X	X	X	X
Accuracy	Parts are produced within an accuracy of +/- .200 mm (.008 in), or +/- .002 mm/mm (.002 in/in), whichever is greater.				
Material Delivery Options	Stratasys F170 = 2 material spool bays, 1 for model, 1 for support located in a drawer on the front of the unit Stratasys F270/F370 = 4 material spool bays, 2 for model, 2 for support located in a drawer on the front of the unit				
Network Connectivity	Wired: TCP/IP protocols at 100 Mbps minimum 100 base T, Ethernet protocol, RJ45 connector Wireless: IEEE 802.11n, g, or b; Authentication: WPA2-PSK, 802.1x EAP; Encryption: CCMP, TKIP				
Software	GrabCAD Print (download): Stratasys F170, F270 and F370 Insight software license: Stratasys F370 only				
System Requirements	Windows 7, 8, 8.1 and 10 (64bit only) with a minimum of 4GB RAM (8GB or more recommended)				
Operating Environment	Operating: Temperature: 59-86°F (15-30°C), Humidity: 30-70% RH Storage: Temperature: 32-95°F (0-35°C), Humidity: 20-90% RH				
Power Requirements	100–132V/15A or 200–240V/7A. 50/60 Hz				
Regulatory Compliance	CE, FCC, EAC, EMC (low-voltage directive), TUV, FCC, RC, RCM, RoHs, WEEE, Reach				

Figure 4. 1 Stratasys F-370 printer Technical specification

The Lattice Optimized bracket selected for the testing will be 3D Printed using material by Stratasys named as ABS PLUS - P430. The technical datasheet of the material is as shown in figure (4.2).

ABSplus-P430

PRODUCTION-GRADE THERMOPLASTIC
FOR DESIGN SERIES 3D PRINTERS

MECHANICAL PROPERTIES	TEST METHOD	ENGLISH	METRIC
		XZ AXIS	XZ AXIS
Tensile Strength, Ultimate (Type 1, 0.125", 0.2"/min)	ASTM D638	4,700 psi	33 MPa
Tensile Strength, Yield (Type 1, 0.125", 0.2"/min)	ASTM D638	4,550 psi	31 MPa
Tensile Modulus (Type 1, 0.125", 0.2"/min)	ASTM D638	320,000 psi	2,200 MPa
Tensile Elongation at Break (Type 1, 0.125", 0.2"/min)	ASTM D638	6%	6%
Tensile Elongation at Yield (Type 1, 0.125", 0.2"/min)	ASTM D638	2%	2%
IZOD Impact, notched (Method A, 23°C)	ASTM D256	2.0 ft-lb/in	106 J/m

MECHANICAL PROPERTIES	TEST METHOD	ENGLISH		METRIC	
		XZ AXIS	ZX AXIS	XZ AXIS	ZX AXIS
Flexural Strength (Method 1, 0.05"/min)	ASTM D790	8,450 psi	5,050 psi	58 MPa	35 MPa
Flexural Modulus (Method 1, 0.05"/min)	ASTM D790	300,000 psi	240,000 psi	2,100 MPa	1,650 MPa
Flexural Strain at Break (Method 1, 0.05"/min)	ASTM D790	4%	4%	2%	2%

OTHER[2]	TEST METHOD	VALUE
Specific Gravity	ASTM D792	1.04

Figure 4. 2 ABSplus-P430 data sheet

4.3 Additive Manufacturing Production

The procedure for 3D printing the Topology and Lattice Optimized is as stated below:

1. Error correction on the STL File
2. Printing Parameter Setup
3. Printing and Post Processing of the Workpiece

Error correction on the STL File

Once the Topology and Lattice Optimized Design are finalized, the finalized files are exported in STL Format. The exportation of files in STL format introduces errors in the geometry of the workpiece, therefore to prepare the file for 3D printing, a software named Magics was used. The feature of Magics software called as Fix Wizard automatically detects the problem in STL file, thus allowing us to repair and correct defects of our STL file. The repair includes attachment of inverted triangles, bad edges, holes within the CAD and other defects related to STL file. Figure (4.3) shows Magics software feature fix wizard used to repair and correct our STL file.

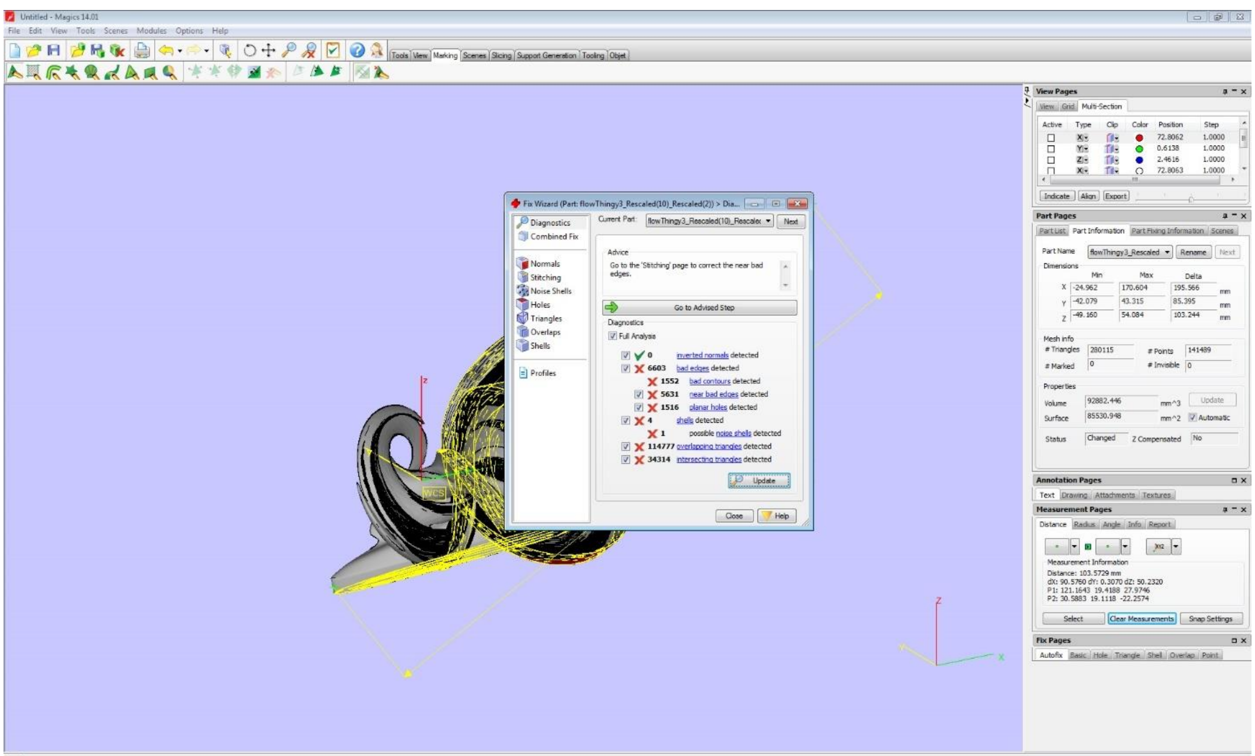


Figure 4. 3 Fix Wizard Feature

Once the file is free from errors and defects then we can proceed to the next step.

Printing Parameter Setup

After removing the defects from the STL file using the Magics software the next step is to use the application Grabcad Print. Grabcad print application is provided by the manufacturer of the printer Stratasys that is must to use in order to 3D Print through Stratasys Printers.

The application allows us to decide printing parameters such as

- Material fill in or fill in density of the material for the print
- Layer thickness
- Setting the printing orientation of the part
- Enter media
- Support generation

The application also estimates the printing time required to print. Figure (4.4) and figure (4.5) shows the Grabcad application set up of one of our specimens that will be 3D Printed.

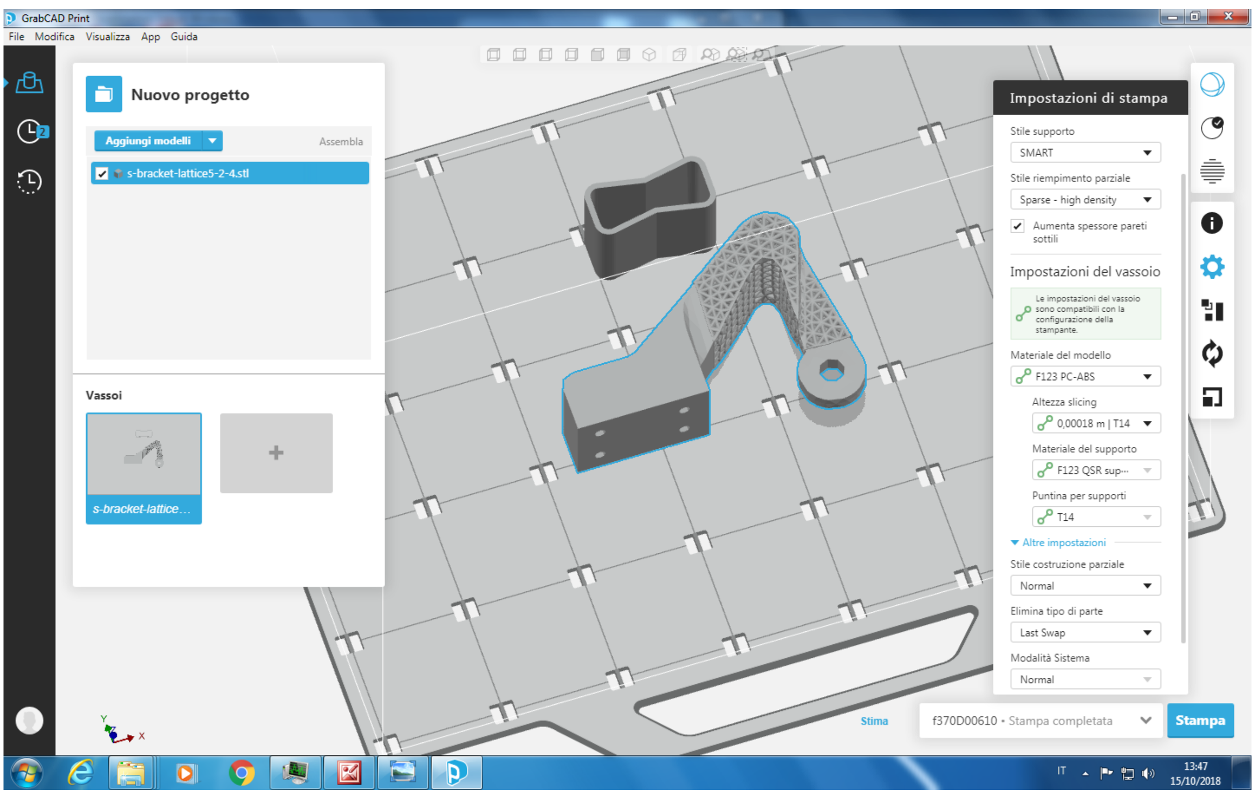


Figure 4. 4 Grabcad application setup

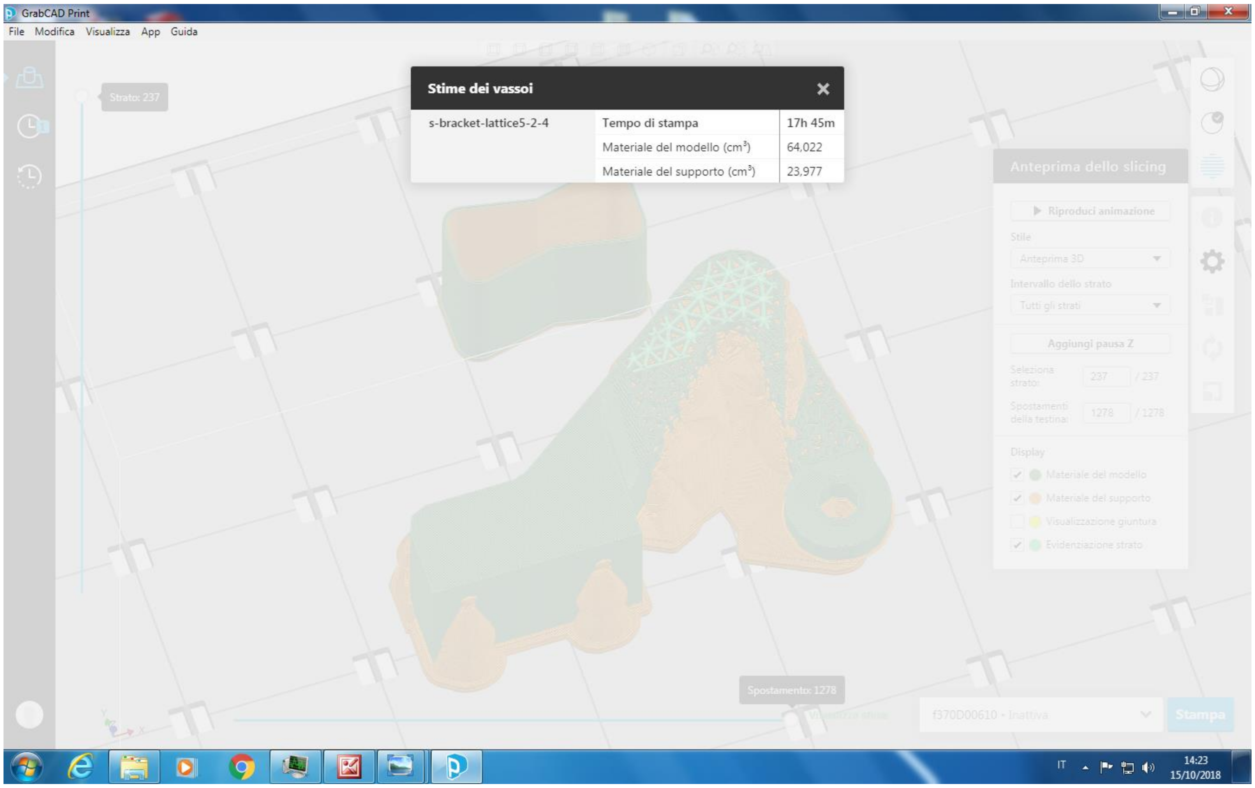


Figure 4. 5 Grabcad application setup

Once all the printing parameters are set the file is ready for printing and we can proceed with the machine to begin the printing.

Printing and Post Processing of the Workpiece

The brackets will be 3D printed using Stratasys F-370 printer. The printer is equipped with two extruders, one for printing material and the other for support material. The printing material will be ABS plus P-430 and the support material that will be used is QSR Support. Both the materials are from Stratasys. Figure (4.6) shows the specimen on the printer bed of the printer after the completion of the print job.

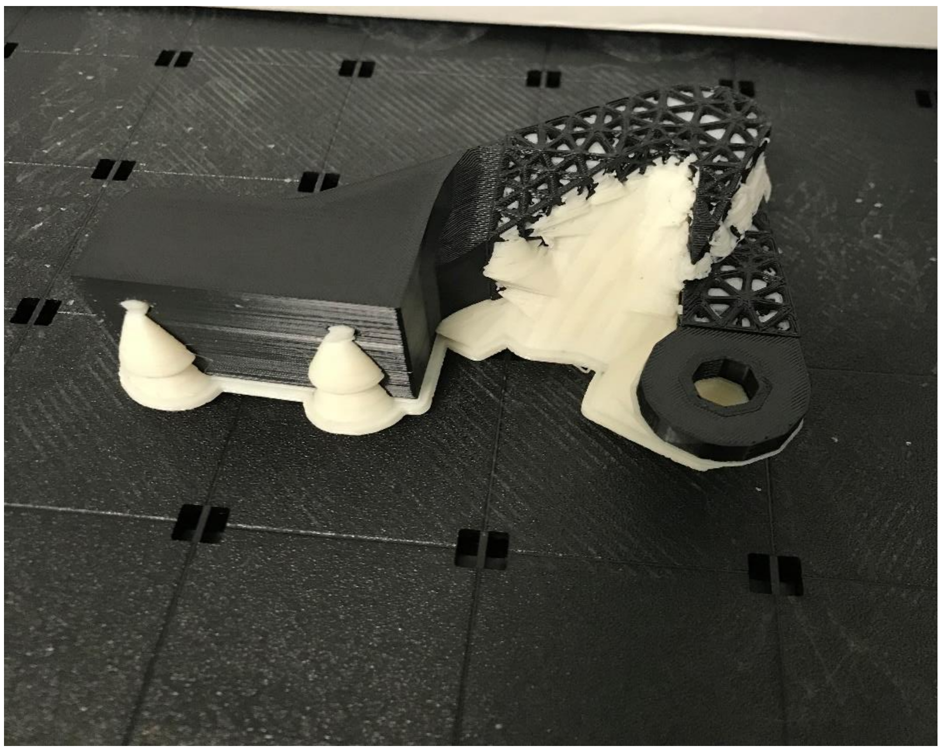


Figure 4. 6 Printed component

Once the printing is complete the samples are extracted from the printer bed and initially manually cleaned. The next step is clean the component and remove all supporting media from the specimen. As the support material has a property of solubilizing in the solution of water and sodium hydroxide, therefore in order to remove the support material the component will be left in a bath of water and NaOH, at a temperature of 70 degree Celsius up to 8 hours. Once support material is removed the specimen is ready for the testing phase.

4.4 Cost Analysis

As shown in table (4.2) the Manufacturing time and consumption of material was evaluated. Based on data the cost associated with the 3D printing of the specimens was evaluated. The cost associated to specimens is as shown in table (4.3).

Table 4. 2 3D Printing time and Material

3D Printed Specimen	Printing time (hours)	Printing Material (cm^3)	Support Material (cm^3)
Specimen D1	17 hours - 56 min	61.963	23.722
Specimen D2	17 hours - 56 min	61.963	23.722
Specimen E1	17 hours - 45 min	64.022	23.977
Specimen E2	17 hours - 45 min	64.022	23.977
Specimen F	33 hours - 2 min	52.58	24.01
Specimen G	33 hours – 1 min	52.39	23.96

Table 4. 3 Costing

3D Printed Specimen	AM Machine and post cleaning cost (€)	Printing Material cost (€)	Support Material cost (€)	Total cost Of specimen (€)
Specimen D1	60.33	10.53	4.03	74.89
Specimen D2	60.33	10.53	4.03	74.89
Specimen E1	59.87	10.88	4.07	74.83
Specimen E2	59.87	10.88	4.07	74.83
Specimen F	98.125	8.93	4.0	111.14
Specimen G	98.04	8.90	4.0	111.02
Total cost				521.6

CHAPTER 5 – Experimentation Process

5.1 Background

The tensile test is a case of deformation of a material and is performed to study the behavior of the material under loading. In relevance to our thesis we will perform tensile test on 3D printed component – (Topology Optimized and Lattice Optimized) to obtaining factor of safety value of real 3D printed component that will be compared with the factor of safety value obtained from software simulation.

The Machine that will be used to perform the test is AURA 10T 2018 a product of Easydur Italiana as shown in figure (5.1).

Figure 5. 1 AURA 10T 2018

5.2 Testing procedure

The work pieces that will be tested are (Topology Optimized and Lattice Optimized) therefore have a complicated geometry. As illustrated in figure (5.2) the testing procedure adopted keeping in consideration the Force Application and constraints applied during the Design is that the workpiece will be pulled along Y axis and is constrained thought four holes of the base.

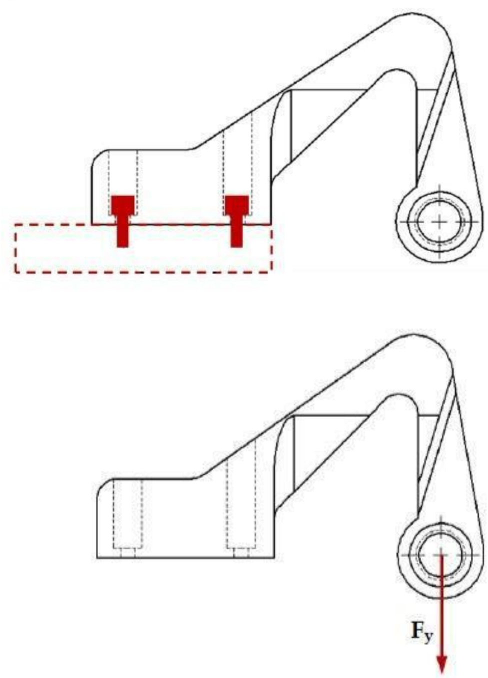


Figure 5. 2 Testing procedure

In order to perform the tensile test keeping in consideration the specification used during the simulation, two additional parts were used:

1. C Shape Bracket - to constraint the base of the work piece and align the axis of the hole on the arm with axis on which the machine applies the force.
2. cylindrical shape locking pin – to establish connection between the upper jaw of the machine and the Hole on the Bracket arm where Force in Y- axis direction will be applied.

The experimental setup with C Shape bracket and Locking pin is illustrated as in figure (5.3).

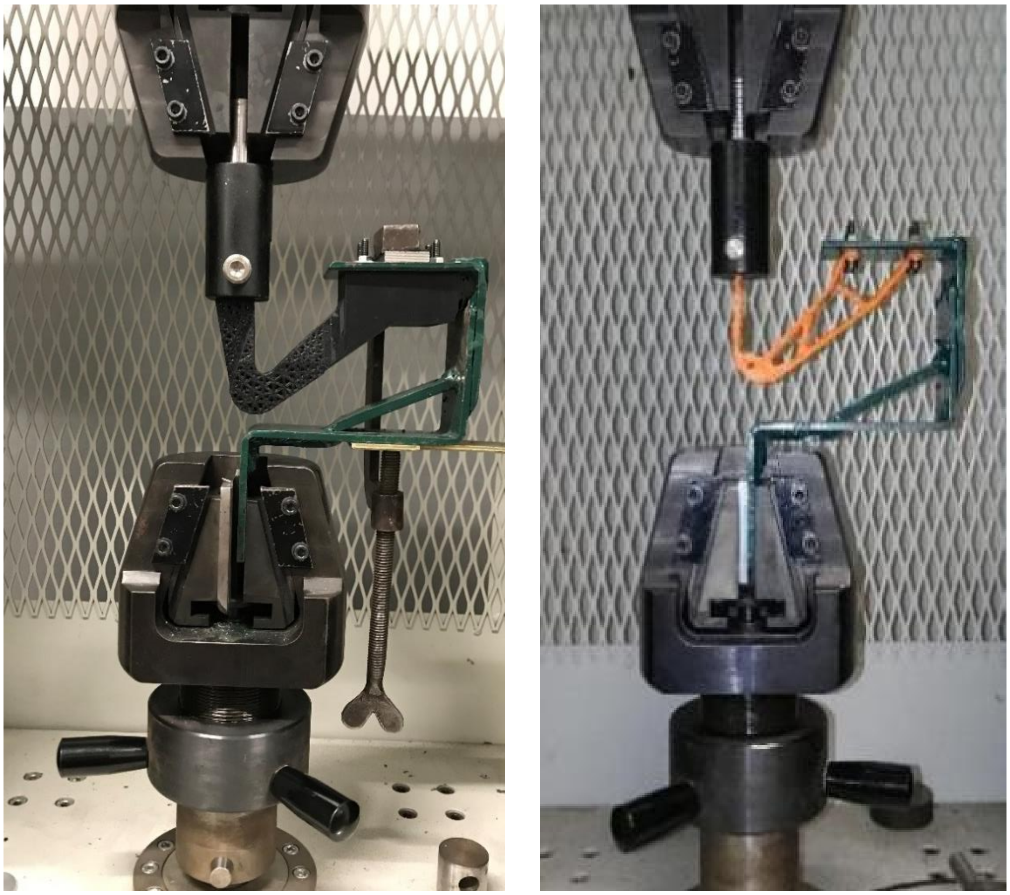


Figure 5. 3 Experimental Setup

5.3 Tensile testing of topology optimized specimen

The 3D printed Topology optimized components that are analyzed in this section are the one already developed in the laboratory. Therefore, it was decided not to reproduce the component and to do interpretation of the previously obtained data. The components were topology optimized with FS = 1.5 for load cases of 100 N, 200 N and 300 N. The specimen and their description are as stated in table (5.1).

Table 5. 1 Testing – Topology optimized specimen

3D Printed Specimens	Case study	Description
Specimen A	Case A	Topology Optimization for 100 N load
Specimen B	Case B	Topology Optimization for 200 N load
Specimen C	Case C	Topology Optimization for 300 N load

Case Study A

As illustrated in figure (5.4) the component is Topology Optimized with the design objective of reducing Mass with applied load of 100N. The results of the simulation illustrate a minimum Factor of safety of 1.658 in the region of upper surface of four holes where constraint is applied.

It can be theoretically approximated from the simulation result that the specimen will resist up to a load of 150 N.

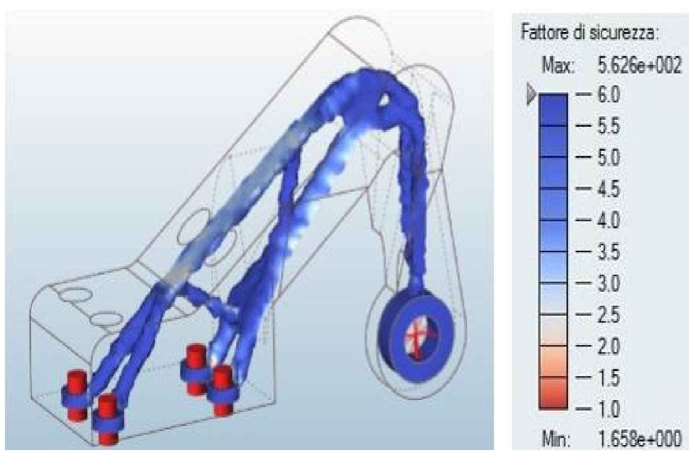


Figure 5. 4 Case A - FEM Result

Specimen A

The specimen tested broke in the beginning of the test without even satisfying minimum factor of safety of 1. The possible reason of failure could be that the optimized specimen had very small thin cross section due to which there was not proper adhesion in between the layers of deposition of the molten material therefore the result is not conformed. Figure 5.5 shows the broken specimen.

Figure 5. 5 Specimen A - Broken

Figure (5.6) shows the graph of the results obtained from the experiment. It can be noted that the specimen broke at:

• maximum load 37.26 N corresponding to a FS 0.37

• at maximum load deformation 3.25 mm

The result outcome was not as expected by the simulation and the component showed the structural weakness related to its manufacturing that the software did not indicated during the analysis.

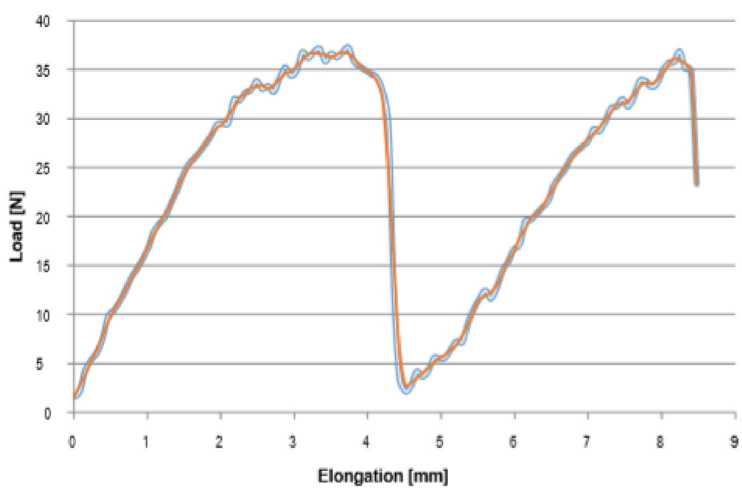


Figure 5. 6 Specimen A - Load/Elongation Graph

Case Study B

The figure (5.7) illustrates the component that is Topology Optimized with the design objective of reducing Mass with applied load of 200N. The results of the simulation illustrate a minimum Factor of safety of 1.571 in the region where it has a narrow area and upper surface of holes where constraint is applied.

It can be theoretically approximated from the simulation result that the specimen will resist up to a load of 300 N.

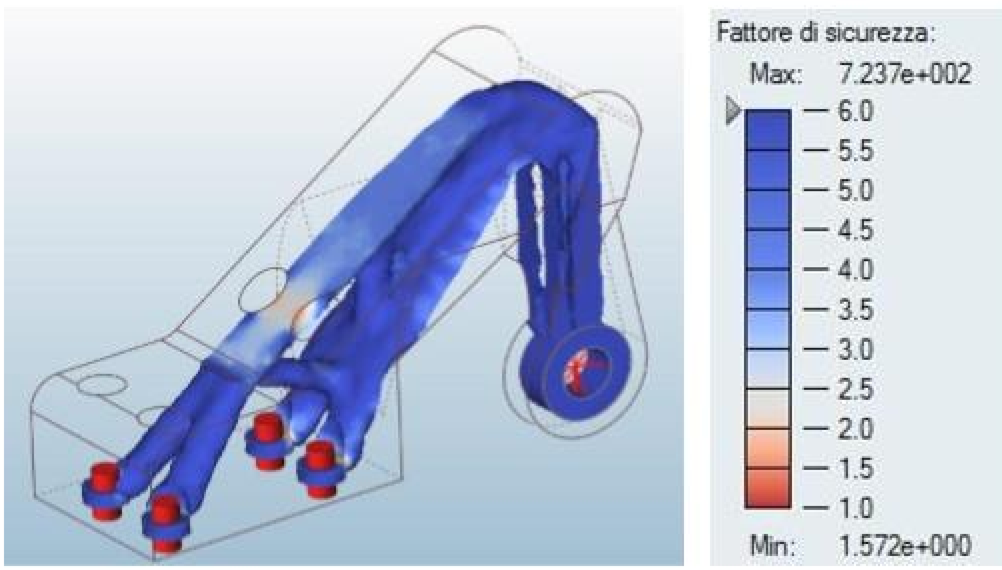


Figure 5. 7 Case B - FEM Result

Specimen B

Difference between the factor of safety predicted by the simulation and the factor of safety attained from experiment was noted as the specimen tested broke after satisfying a factor of safety greater than 1. The possible reason of failure is due to notch effect and anisotropy of the material. Figure (5.8) shows the broken specimen.

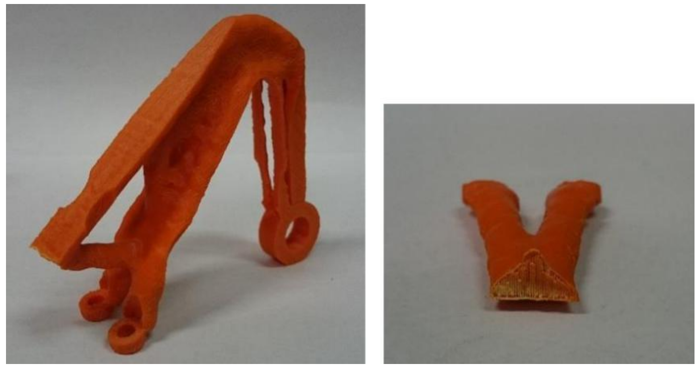


Figure 5. 8 Specimen B - Broken

Figure (5.9) shows the graph of the results obtained from the experiment. It can be noted that the specimen broke at

- maximum load 235.95 N corresponding to a FS 1.18
- at maximum load deformation 12.76 mm

The result outcome was not the same as expected by the simulation, but the component exhibited good resistance to the load by satisfying factor of safety greater than 1.

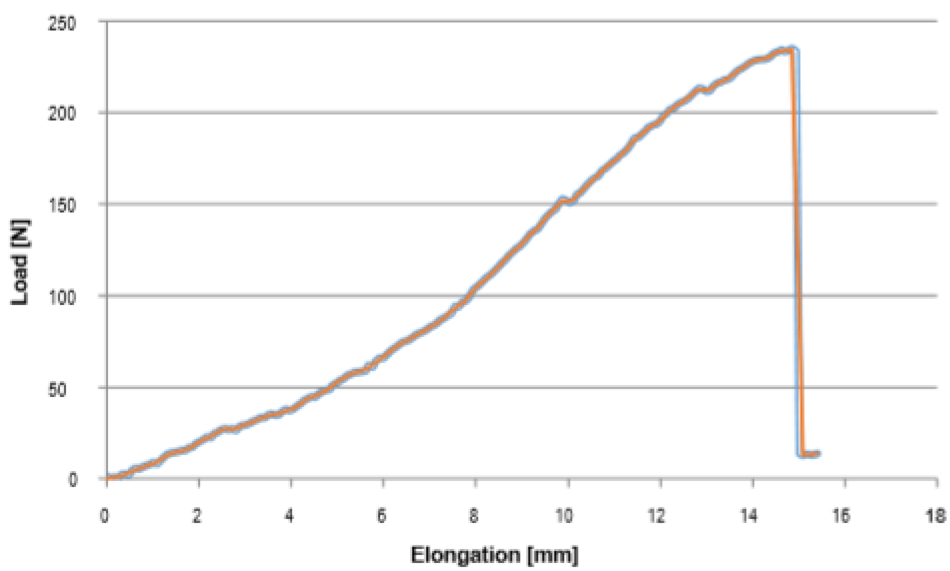


Figure 5. 9 Specimen B - Load/Elongation Graph

Case Study C

Figure (5.10) illustrates the component that is Topology Optimized with the design objective of reducing Mass with applied Load of 300N. The results of the simulation illustrate a minimum Factor of safety of 1.580 and it can be noted that the specimen has more material if compared to other specimens. The specimen is more solid as compare to other specimens.

It can be theoretically approximated from the simulation result that the specimen will resist up to a load of 450 N.

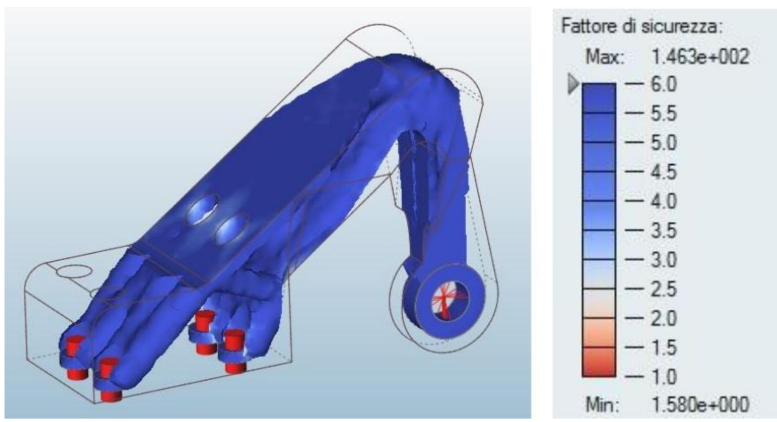


Figure 5. 10 Case C - FEM Result

Specimen C

This time the factor of safety obtained from the experiment was very near to the one predicted by the simulation. It was noted that the specimen broke at 412.92 N. Figure (5.11) shows the broken specimen from which the direction of deposition of the molten material can be observed.

Figure 5. 11 Specimen C - Broken

Figure (5.12) shows the graph of the results obtained from the experiment. It can be noted that the specimen broke at

• maximum load 412.92 N corresponding to a FS 1.37

• at maximum load deformation 14.18 mm

The result outcome was very near to the one expected by the simulation, and the component exhibited good resistance to the load by satisfying a factor of safety of 1.37

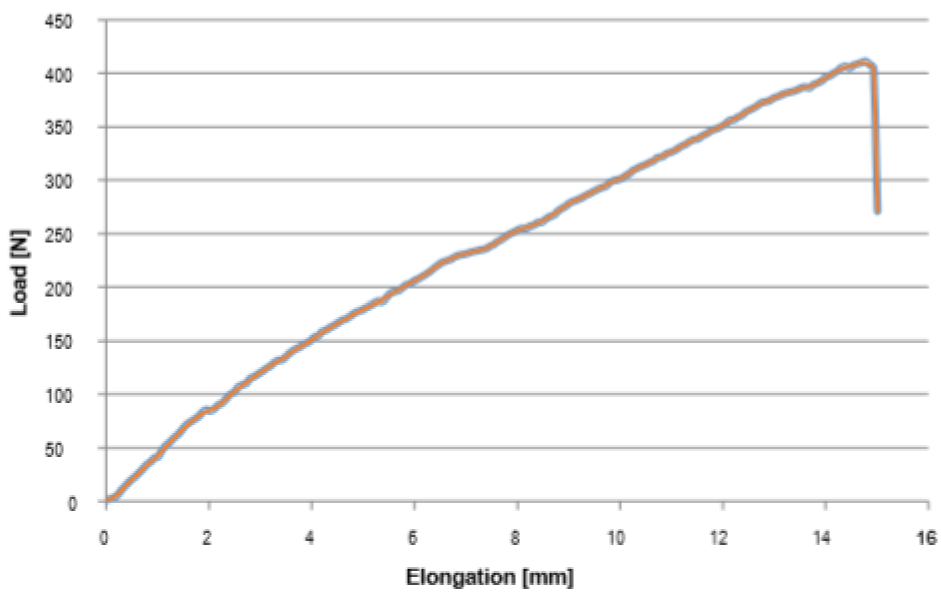


Figure 5. 12 Specimen C - Load/Elongation Graph

Tensile test Results

Table 5. 2 Testing Result – Topology optimization

Specimen	Force (N)	Theoretical FS	Theoretical Strength (N)	Real FS	Experimental Strength (N)	Percentage change
A	100	1.658	165	0.37	37.26	-77.68%
B	200	1.571	314	1.180	235.95	-24.89%
C	300	1.516	454	1.376	412.92	-9.23%

5.4 Tensile testing of lattice optimized specimen

The 3D printed Lattice optimized components that were tested on the tensile testing machine are detailed in table (5.3).

Table 5. 3 Testing – Lattice optimization

3D printed specimens	Case study	Description
Specimen D1 Specimen D2	Case D	Lattice Optimization for scenario 1 - Lattice size-(7-2-4)
Specimen E1 Specimen E2	Case E	Lattice Optimization for scenario 1 - Lattice size-(5-2-4)
Specimen F	Case F	Lattice Optimization for scenario 2 - Lattice size-(7-2-4)
Specimen G	Case G	Lattice Optimization for scenario 2 - Lattice size-(5-2-4)

Case Study D

As evident from the static analysis the design space of the chosen component was that of scenario 1 and was lattice optimized.

As illustrated in figure (5.13) the bracket is Lattice Optimized with lattice size (length – min/max diameter) as (7-2-4) with the design objective of reducing the Mass with applied force of 300N. The results of the simulation illustrate a minimum Factor of safety of 1.4 in the region where the lattice is attached to the full solid part.

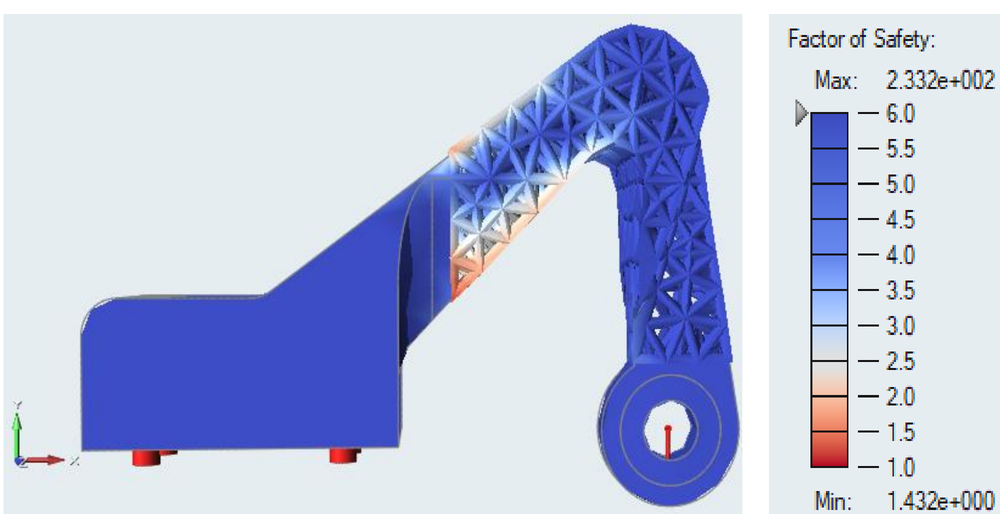


Figure 5. 13 Case D - FEM Result

It can be theoretically approximated from the simulation result that the specimen will resist up to a load of 420 N. For the purpose of testing Two copies, specimen D1 and D2 of lattice (7-2-4) were printed and tested.

Specimen D1

As evident from figure (5.14) the Specimen broke at the location predicted by the simulation. The Specimen did not reach the factor of safety of 1.4 as predicted by the simulation but achieved a Factor of safety greater than 1 by breaking at 337N. Despite of the difference the specimen exhibited satisfactory resistance to loading condition.

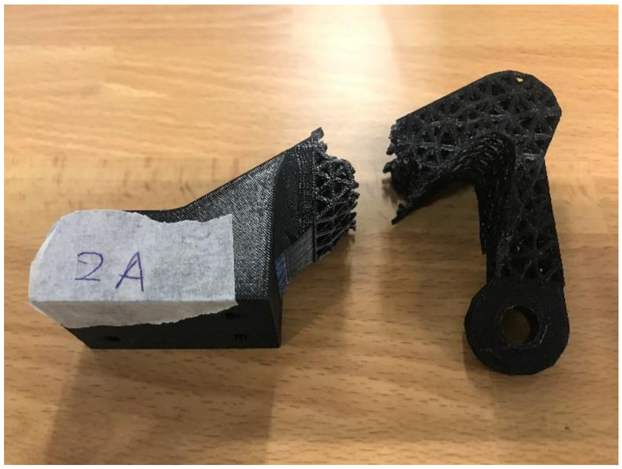


Figure 5. 14 Specimen D1 - Broken

Figure (5.15) shows the graph of the results obtained from the experiment. It can be noted that the specimen broke at

- maximum load 337 N corresponding to a Factor of Safety of 1.12
- maximum load deformation 7.8 mm

The result outcome was near to the result predicted by the simulation, and the component exhibited satisfactory resistance to the load by satisfying factor of safety of 1.12

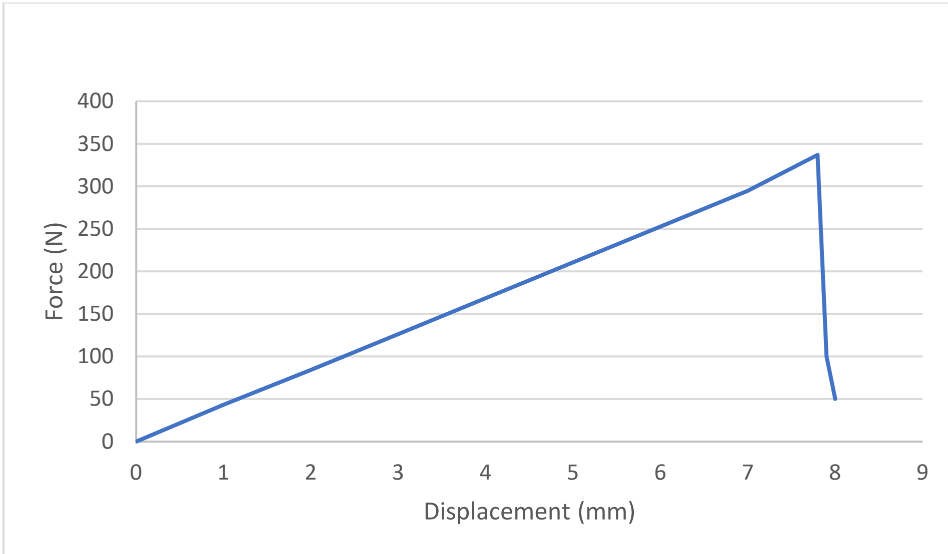


Figure 5. 15 Specimen D1 – Load/Displacement Graph

Specimen D2

The Specimen broke at the location of minimum cross sectional area, but cracks were also observed at the place where the minimum factor of safety was predicted by the simulation. The Specimen reached the factor of safety of 1.35 which is very close to the value predicted by the simulation. The broken specimen is shown in figure (5.16).

Figure 5. 16 Specimen D2 - Broken

Figure (5.17) shows the graph of the results obtained from the experiment. It can be noted that the specimen broke at

• maximum load 406 N corresponding to a Factor of Safety of 1.35

• maximum load deformation 7.7 mm

The result outcome is approximately equal to the one predicted by the simulation, and the component exhibited good resistance to the load by satisfying factor of safety of 1.35.

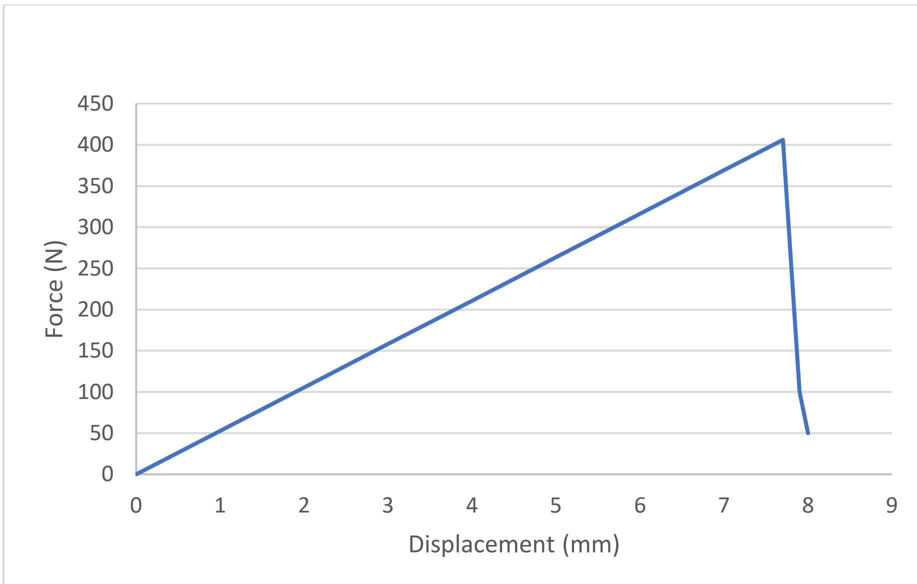


Figure 5. 17 Specimen D2 – Load/Displacement Graph

Case study E

As illustrated in figure (5.18) the bracket is Lattice Optimized with lattice size (length – minimum Diameter – maximum Diameter) as (5-2-4) with the design objective of reducing the Mass with an applied force of 300N. The results of the simulation illustrate a minimum Factor of safety of 1.8 in the region where the lattice is attached to the full solid part.

It can be theoretically approximated from the simulation result that the specimen will resist up to a load of 540 N. For the purpose of testing Two copies, specimen E1 and specimen E2 of lattice (5-2-4) were printed and tested.

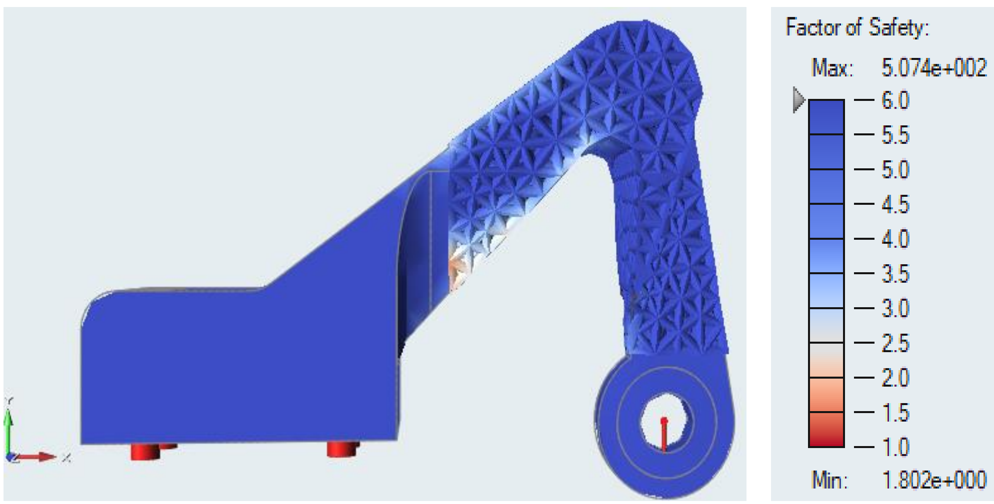


Figure 5. 18 Case E – FEM Result

SPECIMEN E1

The Specimen broke at the location of minimum cross sectional area and not in the region of minimum factor of safety as predicted by the simulation. The Specimen exhibited a very good resistance to the loading condition and reached a factor of safety of 2.3 before breaking, which is more than the minimum factor of safety calculated in simulation. Figure (5.19) shows the broken specimen after the test.

Figure 5. 19 Specimen E1 - Broken

Figure (5.20) shows the graph of the results obtained from the experiment. It can be noted that the specimen broke at

- maximum load 709 N corresponding to a Factor of Safety of 2.36
- maximum load deformation 10.4 mm

The result in terms of factor of safety is more than that predicted by the simulation, and the component exhibited good resistance to the load by breaking at a factor of safety of 2.3

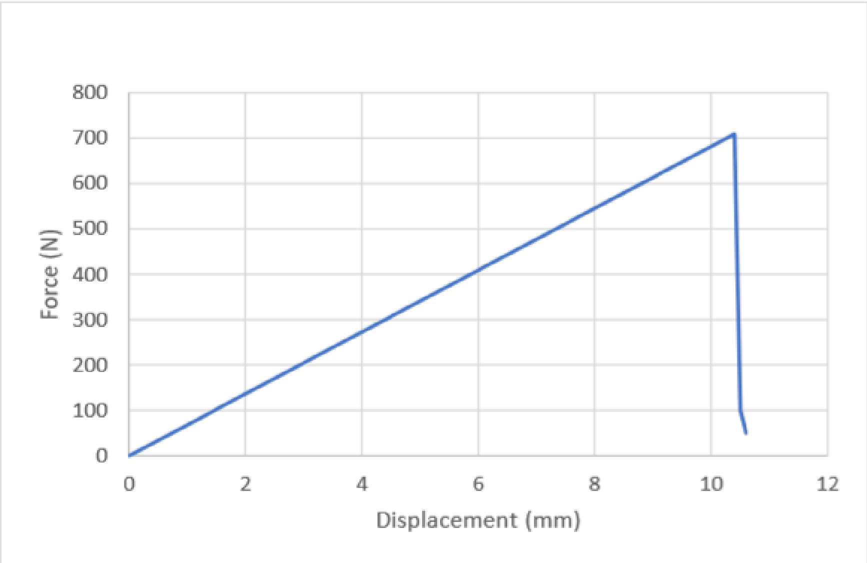


Figure 5. 20 Specimen E1 – Load/Displacement Graph

Specimen E2

As evident from the figure (5.21) the Specimen broke in the region near the region of minimum factor of safety predicted by the simulation. The important thing to observe is that it broke in the region where material density was full and not in the Lattice optimized region. Possible reasons for the failure are

- adhesion in between the layers of deposition of the molten material
- notch effect and anisotropy of the material

The Specimen exhibited a very good resistance to the loading condition and reached the factor of safety of 2.2 before breaking, which is more than the minimum factor of safety calculated in simulation.

Figure 5. 21 Specimen E2 – Broken specimen

Figure (5.22) shows the graph of the results obtained from the experiment. It can be noted that the specimen broke at

• maximum load 662 N corresponding to a Factor of Safety of 2.2

• at maximum load deformation of 10 mm

The result in terms of factor of safety is approximately equal to that of the Specimen E1, and the component exhibited good resistance to the load by breaking at a factor of safety of 2.2

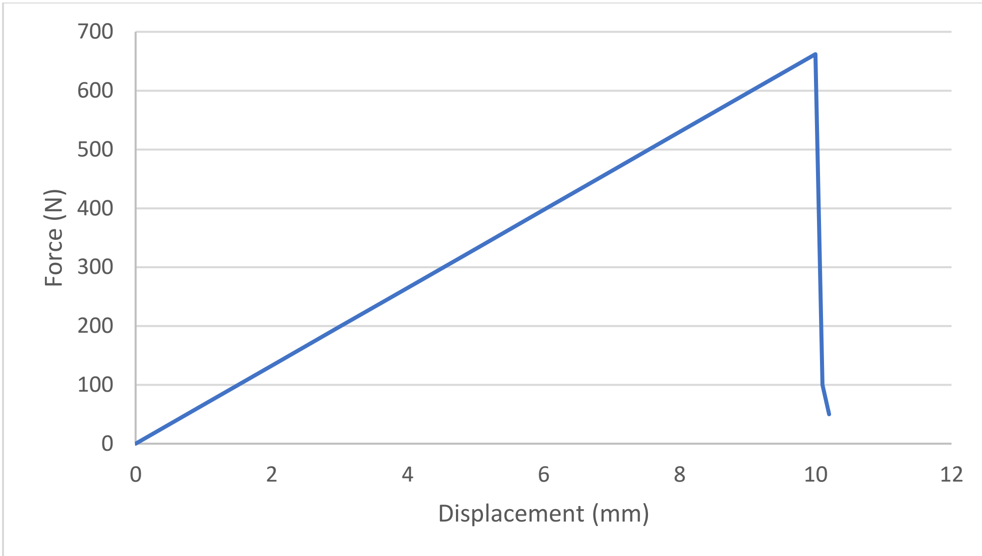


Figure 5. 22 Specimen E2 – Load/Displacement Graph

Case study F

As illustrated in figure (5.23) the bracket with solid base is fully Lattice Optimized with lattice size (length – minimum Diameter – maximum Diameter) as (7-2-4) with the design objective of reducing the Mass with an applied force of 300N. The results of the simulation illustrate a minimum Factor of safety of 1.9 .

It can be theoretically approximated from the simulation result that the specimen will resist up to a load of 570 N. For the purpose of testing, specimen F of lattice size (7-2-4) was printed and tested.

Figure 5. 23 Case F – FEM Result

Specimen F

The Specimen broke at the location of minimum cross sectional area and not in the region of minimum factor of safety as predicted by the simulation. The Specimen exhibited a satisfactory response to the loading condition and reached a factor of safety of 1.14 before breaking, which is less than the minimum factor of safety calculated in simulation. Possible reasons for the failure are

- adhesion in between the layers of deposition of the molten material
- notch effect

Figure (5.24) shows the broken specimen after the test.

Figure 5. 24 Specimen F - Broken

Figure (5.25) shows the graph of the results obtained from the experiment. It can be noted that the specimen broke at

• maximum load 344 N corresponding to a Factor of Safety of 1.14

• maximum load deformation of 6.2 mm

The result outcome was not the same as expected by the simulation, and the component exhibited a satisfactory response to the loading condition by satisfying a factor of safety greater than 1.

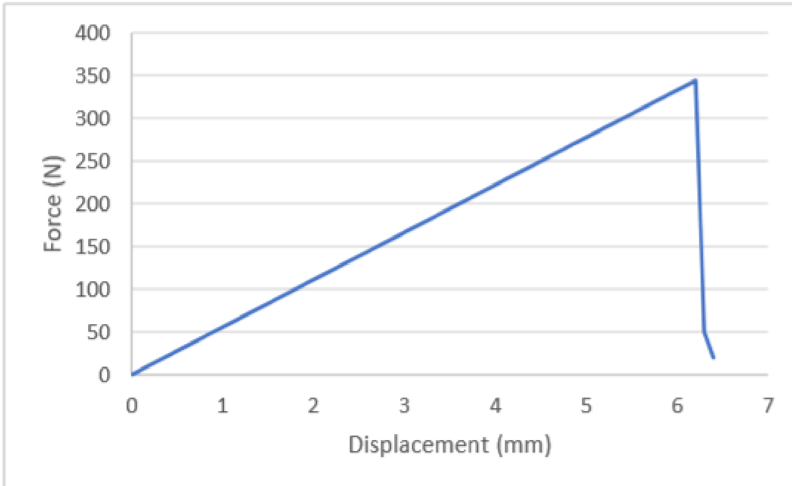


Figure 5. 25 Specimen F – Load/ Displacement Graph

Case study G

As illustrated in figure (5.26) the bracket with solid base is fully Lattice Optimized with lattice size (length – minimum Diameter – maximum Diameter) as (5-2-4) with the design objective of reducing the Mass with an applied force of 300N. The results of the simulation illustrate a minimum Factor of safety of 1.9.

It can be theoretically approximated from the simulation result that the specimen will resist up to a load of 570 N. For the purpose of testing, specimen G of lattice size (5-2-4) was printed and tested.

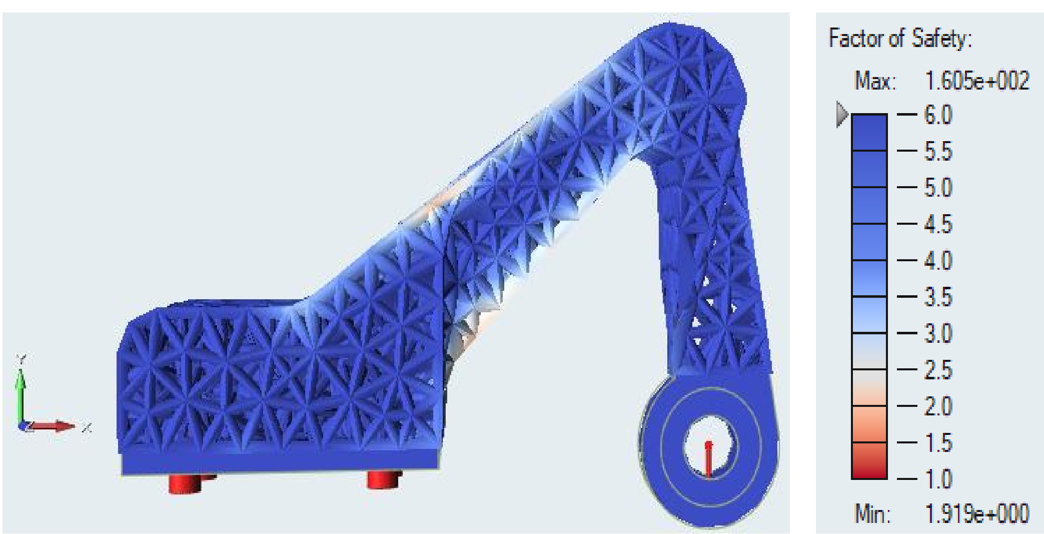


Figure 5. 26 Case G – FEM Result

Specimen G

The Specimen broke at the location of minimum cross sectional area and not in the region of minimum factor of safety as predicted by the software. The Specimen exhibited a very good resistance to the loading condition and reached the factor of safety of 1.4 before breaking, which is less than the minimum factor of safety calculated in simulation. Possible reason for the failure in the region is

- adhesion in between the layers of deposition of the molten material
- notch effect

Figure (5.27) shows the broken specimen after the test.

Figure 5. 27 Specimen G - Broken

Figure (5.28) shows the graph of the results obtained from the experiment. It can be noted that the specimen broke at

- maximum load 434 N corresponding to a Factor of Safety of 1.44
- maximum load deformation 8 mm

The result in terms of minimum factor of safety is less than that calculated in simulation, but the component exhibited a good resistance to the load by breaking at a factor of safety of 1.4

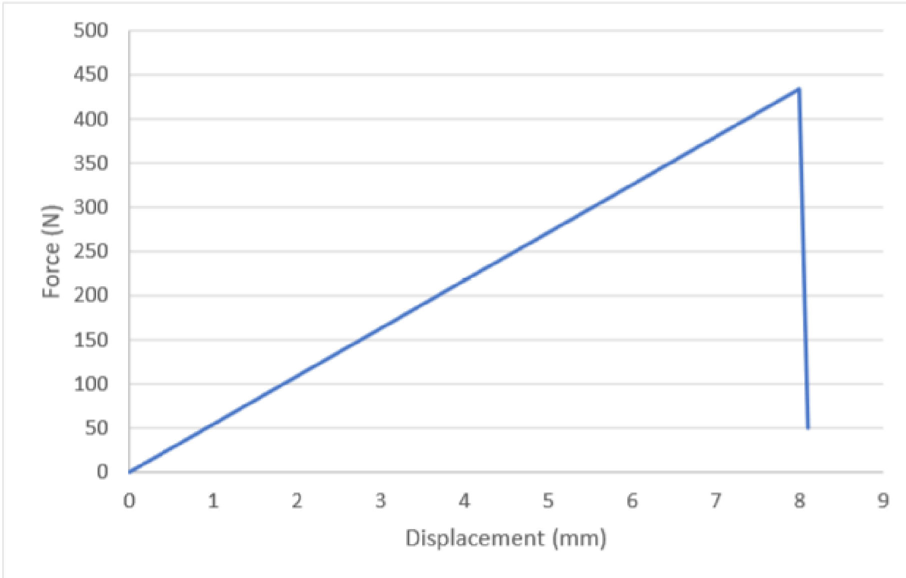


Figure 5. 28 Specimen G – Load/ Displacement Graph

5.5 Result Comparison and Conclusion

In consideration of the Results obtained from the Tensile tests for Topology and lattice optimized specimen as shown in table (5.4), it was observed that the variation in results was due to a change in the distribution of the load. Distribution of load changes after initial crack initiation on specimen whereas the software is not simulating the presence of cracks and manufacturing defects. Another possible reason of the change in distribution of the load is that the specimen is undergoing displacement during the test.

It was also observed that specimen E1 and specimen E2 which were optimized with lattice size (5-2-4) performed better than expected by the simulation as compared to specimen D1 and specimen D2 which were optimized with lattice size (7-2-4). Therefore, it can be concluded that a lattice with short trusses works better in comparison to the lattice with long trusses.

Among Topology optimized specimens the specimen C performed better by exhibiting less percentage change in terms of strength. Specimen C have high strength to weight ratio and a low cost to strength ratio equal to 0.14 as shown in table (5.5).

Table 5. 4 Testing Result – Specimen

Specimen	Force (N)	Theoretical FS	Theoretical Strength (N)	Real FS	Experimental Strength (N)	Percentage change
A	100	1.658	165	0.37	37.26	-77.68 %
B	200	1.571	314	1.180	235.95	-24.89 %
C	300	1.516	454	1.376	412.92	-9.23 %
D1	300	1.4	420	1.12	337	- 19.7 %
D2	300	1.4	420	1.35	406	- 3.3 %
E1	300	1.8	540	2.36	709	+ 31.3 %
E2	300	1.8	540	2.2	662	+ 22.6 %
F	300	1.9	570	1.14	344	- 39.6 %
G	300	1.9	570	1.44	434	- 23.9 %

Table (5.5) shows the Strength to weight ratio (N/Kg) and cost to load ratio of the specimens.

Table 5.5 Specimens – (N/Kg) - (€/N)

Specimen	Experimental strength (N)	Weight (Kg)	3D printing cost (€)	Strength to weight ratio (N/Kg)	Cost-to-strength Ratio (€/N)
C	412	0.0364	59.89	11318.7	0.14
D1	337	0.086	74.89	3918.6	0.22
D2	406	0.086	74.89	4720.9	0.18
E1	709	0.101	74.83	7019.8	0.10
E2	662	0.101	74.83	6554.5	0.11
F	344	0.069	111.14	4985.5	0.32
G	434	0.071	111.02	6112.7	0.25

The values in tables 5.4 and 5.5 illustrates that the best Lattice optimized specimen is the E1 of case study E, as it exhibits highest Strength to weight ratio (N/Kg) ratio with low cost and lowest Cost-to-strength Ratio (€/N) among all. Whereas the best Topology optimized specimen is specimen C as it exhibits highest Strength to weight ratio (N/Kg) ratio with low cost and lowest Cost-to-strength Ratio (€/N) among all topology optimized specimens.

Due to the presence of manufacturing defect, it was realized that in order to improve reliability of results more number of replicas of specimens should by 3D printed and tested in future work.

References

- *Gibson, D. W. Rosen, and B. Stucker, Additive Manufacturing Technologies, Second Edition Springer, 2010.*
- *D. Brackett, I. Ashcroft, and R. Hague, “Topology optimization for additive manufacturing,” Solid Freeform Fabrication Symposium.*
- *W. Dias and D. Anand, “Design and Optimization of Lattice Structure for 3D Printing using Altair OptiStruct,” 2015. [Online]. Available: http://insider.altairhyperworks.com/design-and-optimization-of-lattice-structuresfor-3d-printing-using-altair-optistruct/. [Accessed: 23-May-2016].*
- *https://forum.altair.com/index.php?/topic/1525-inspire-20173/*
- *https://solidthinking.com/inspire2018.html*
- *A.LOCATELLI, Msc Thesis in Mechanical Engineering, Politecnico Di Torino, October 2016.*