Major Project Report on

TOPOLOGY OPTIMISATION USING GENERATIVE DESIGN

Submitted by

ANUSHA NAMPALLY (201700556)

A Thesis Submitted in

Partial Fulfillment of the Requirements for the Degree of

Bachelor of Technology
in
Mechanical Engineering

(2021)

Under the Supervision of

Mr. Saurabh Sharma
Assistance Professor
SMIT

Mr. Sreedhar Babu Gollapudi
Scientist 'F'
Research Centre Imarat, DRDO

DEPARTMENT OF MECHANICAL ENGINEERING
SIKKIM MANIPAL INSTITUTE OF TECHNOLOGY
MAJHITAR, RANGPO, EAST SIKKIM-737132

DECLARATION

Thesis Title: Topology Optimization using Generative Design
The degree for which the Thesis is submitted:

I declare that the presented thesis represents largely my own ideas and work in my own words. Where others' ideas or words have been included, I adequately cite and listed in the reference materials. The thesis has been prepared without resorting to plagiarism. I have adhered to all principles of academic honesty and integrity. No falsified or fabricated data have been presented in the thesis. I understand that any violation of the above will cause for disciplinary action by the Institute, including revoking the conferred degree, if conferred, and can also evoke penal action from the sources which have not been properly cited or from whom proper permission has not been taken.

(Signature)

Anusha Nampally

Registration no.: 201700556

Date: 08/06/2021

SIKKIM MANIPAL UNIVERSITY

SIKKIM MANIPAL INSTITUTE OF TECHNOLOGY

CERTIFICATE

This is to certify that the thesis titled "Topology Optimization using Generative Design" is a bonafide record of the authentic work carried out by Anusha Nampally bearing 201700556 under the supervision and guidance of (Mr. Saurabh Sharma, Assistant Professor) during the period 2020-2021 and is submitted in partial fulfillment of the requirement for the award of the degree of Bachelor of Technology in Mechanical Engineering during the academic year 2020-2021 under Mechanical Engineering Department, Sikkim Manipal Institute of Technology.

Mr. Saurabh Sharma	Prof. (Dr.) Ruben Phipon
Assistance Professor	Head of Department
Mechanical Engineering Department	Mechanical Engineering Department
SMIT	SMIT

ACKNOWLEDGEMENT

I sincerely take this opportunity to express my thanks and deep gratitude to all who extended their wholehearted cooperation, opinion, and gracious hospitality to me in completing this work.

It was a pleasure working under the guidance of **Mr. Saurabh Sharma** as an internal guide, however I am also indebted to **Mr. Sreedhar Babu Gollapudi** who was my external guide and without his incredible support and guidance, the project work would have been insurmountable.

I am exceptionally grateful to **Dr. Ruben Phipon**, Head of Department, Mechanical Engineering, Sikkim Manipal Institute of Technology for his constant support and fortitude.

I own my greatest debt to **Prof. (Dr.) Ashis Sharma**, Director, Sikkim Manipal Institute of Technology for their kind co-operation to complete the project works.

And, lastly to my family for support my endeavours.

(Signature)

Anusha Nampally

201700556

Place : SMIT

Date : 08/06/2021

ABSTRACT

Topology Optimization reduces the unwanted material from the manufacturing part keeping the fabrication constraints in view. Objective of this paper is to perform Static and non-linear analysis of Aerospace fin. It uses FEM technique to perform analysis of stress, strain and deflection. Modeling of Aerospace fin is done in SOLIDWORKS and ANSYS, which is further analyzed by using FEM. A Force of 15kN was applied normal to the plane as aerodynamic loads. Analysis is done in SOLIDWORKS and Ansys for stress, deformation and strain. Topology Optimization is done using Ansys software. Once the results were obtained, the aerospace fin was re-modelled using Generative Design in Ansys and NETFABB. Both the static and non-linear analysis was done using SOLIDWORKS and Ansys. The results are to be made sure that the deflection and strength are constant. High strength to weight ratio with minimized deflection improving flexural rigidity without material change. The main purpose of this project is to reduce the unwanted material/mass during manufacturing which directly improves efficiency.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

NOMENCLATURE

Symbols used	Description
TO	Topology optimization
FEM	Finite Element Method
AM	Additive Manufacturing
3D	Three-Dimensional
FEA	Finite Element Analysis
STL	Stereolithography
MKS	Metre Kilogram Second
CS	Control Surface
DRDO	Defence Research and Development Organization
RCI	Research Centre Imarat

s

CHAPTER 1
INTRODUCTION

1.1 About Topology Optimization

Topology optimization (TO) is a mathematical method that optimizes material layout within a given design space, for a given set of loads and boundary conditions and constraints with the goal of maximizing the performance of the system. TO is different from shape optimization and sizing optimization in the sense that the design can attain any shape within the design space, instead of dealing with predefined configurations.

The conventional TO formulation FINITE ELEMENT METHOD (FEM) to evaluate the design performance. The design is optimized using either gradient based mathematical techniques such as the optimality criteria algorithm and the method of moving asymptotes or non-gradient-based algorithms such as genetic algorithms.

Topology Optimization has a wide range of applications in aerospace, mechanical, bio-chemical and civil engineering. Currently, engineers mostly use TO at the concept level of a design process. Due to the free forms that naturally occur, the result is often difficult to manufacture. For that reason the result emerging from TO is often fine-tuned for manufacturing. Adding constraints to the formulation in order to increase the manufacturing is an active field of research. In some cases, results from TO can be directly manufactured using Additive Manufacturing; TO is thus a key part of design for Additive Manufacturing (AM).

1.2 ADVANTAGES OF TOPOLOGY OPTIMIZATION

- REDUCES UNNECESSARY WEIGHT
- SIZE OPTIMIZATION WHICH LEADS TO FASTER DESIGN PROCESS
- INCREASES EFFICIENCY
- COST OPTIMIZATION
- SUSTAINABILITY

1.3 DISADVANTAGES OF TOPOLOGY OPTIMIZATION

- Production Limitations
- difficult designs of manufacturing

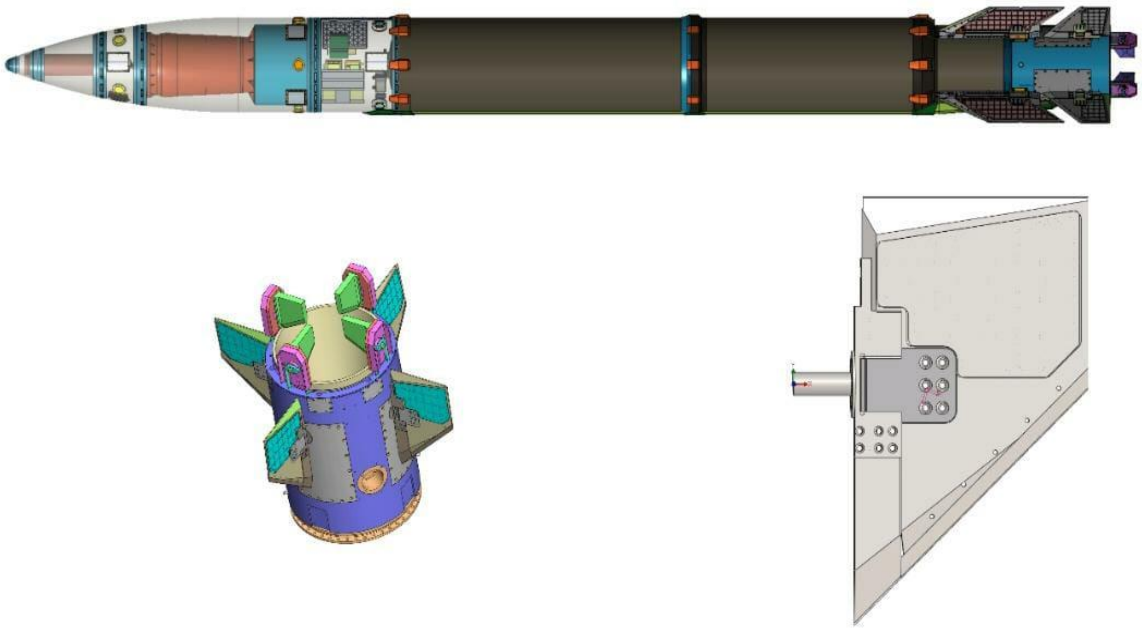


Figure 1: The Missile

1.4 SOFTWARES USED FOR MODELING, SIMULATION AND TOPOLOGY OPTIMISATION

SOLIDWORKS (2019):

- SOLIDWORKS is a 3D solid modeling software which allows users to develop full solid models in a simulated environment for both design and analysis.
- Finite element analysis (FEA) is the modeling of products and systems in a virtual environment, for the purpose of finding and solving potential (or existing) structural or performance issues. FEA is the practical application of the finite element method (FEM), which is used by engineers and scientist to mathematically model and numerically solve very complex structural, fluid, and multi-physics problems.

ANSYS :

- ANSYS is a general-purpose software, used to simulate interactions of all disciplines of physics, structural, vibration, fluid dynamics, heat transfer, topology study and electromagnetic for engineers.
- Ansys publishes engineering analysis software across a range of disciplines as follows Finite element analysis structural analysis computational fluid dynamics Heat transfer Explicit dynamic analysis (worked on topology optimization using ANSYS).

NETFABB :

- NETFABB is a software tailored for additive fabrication, rapid prototyping and 3D printing. It includes the ability of viewing, editing, generative design, repairing and analyzing STL-files or slice-based files in various formats (worked on generative designing).

1.5 DESIGNING OF AEROSPACE FIN (CONTROL SURFACE)

3D model of aerospace fin designed with the help of Solidworks modelling software with the existing measurements.

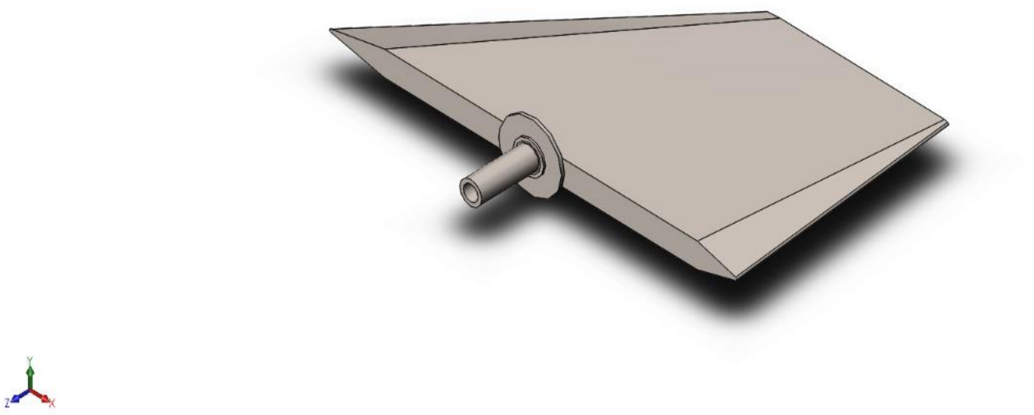
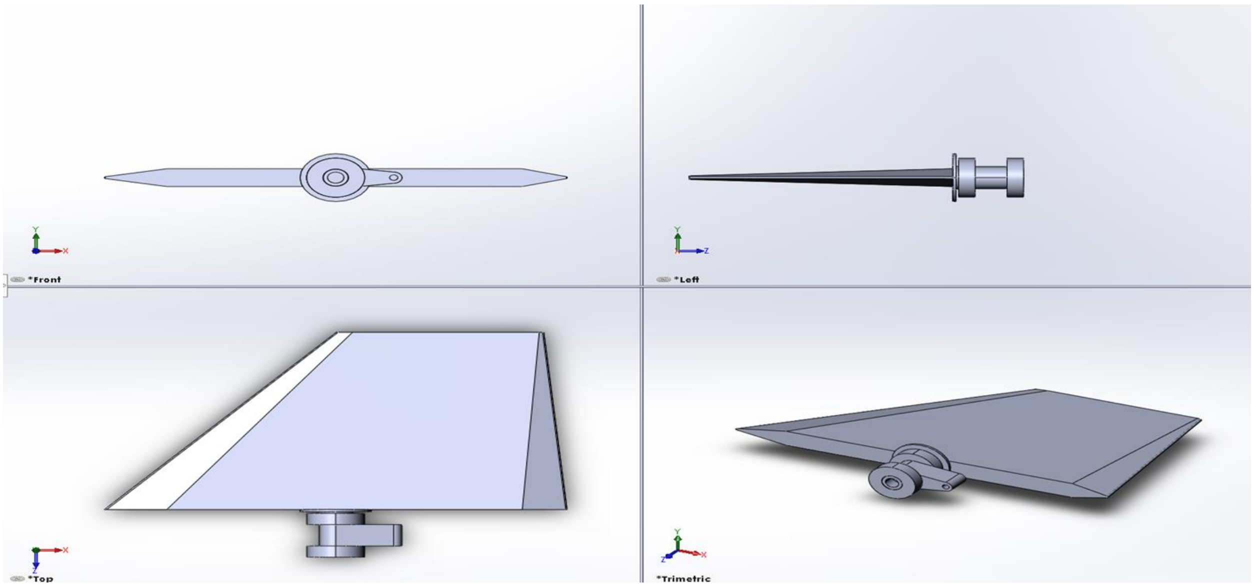


Figure 2: Modelled Aerospace fin



4-sided view of Aerospace Fin

CHAPTER 2
TOPOLOGY OPTIMIZATION USING SOLIDWORKS

2.1 STATIC ANALYSIS OF DESIGN

Static analysis allows the designer to determine the reaction forces on whole mechanical systems as well as interconnection forces transmitted to their individual joints.

Here Static analysis is performed on the Aerospace Fin in Solidworks. The analysis results are as follows:

- Static Stress Analysis
- Static Deformation analysis
- Static Strain Analysis

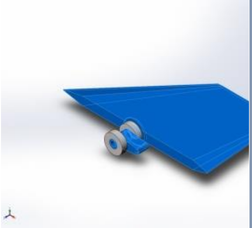
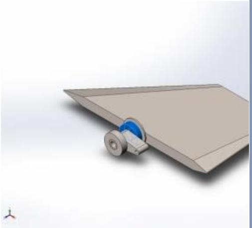
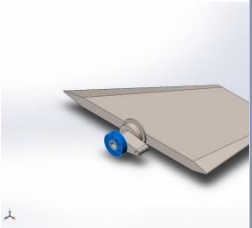
	Treated As	Volumetric Properties
Boss-Extrude11 	Solid Body	Mass:20.0136 kg Volume:0.00259917 m^3 Density:7,700 kg/m^3 Weight:196.134 N
Boss-Extrude1 	Solid Body	Mass:0.476427 kg Volume:6.18737e-05 m^3 Density:7,700 kg/m^3 Weight:4.66899 N
Boss-Extrude1 	Solid Body	Mass:0.476427 kg Volume:6.18737e-05 m^3 Density:7,700 kg/m^3 Weight:4.66899 N
Unit system:		SI (MKS)
Length/Displacement		Mm
Temperature		Kelvin
Angular velocity		Rad/sec
Pressure/Stress		N/m^2

Table 1: Static analysis of design

Table 2: Study Properties

Study name	Static 1
Analysis type	Static
Mesh type	Solid Mesh
Thermal Effect:	On
Thermal option	Include temperature loads
Zero strain temperature	298 Kelvin
Include fluid pressure effects from SOLIDWORKS Flow Simulation	Off
Solver type	FFE Plus
In plane Effect:	Off

Soft Spring:	Off
Inertial Relief:	Off
Incompatible bonding options	Automatic
Large displacement	Off
Compute free body forces	On
Friction	Off

Table 3: Material Properties

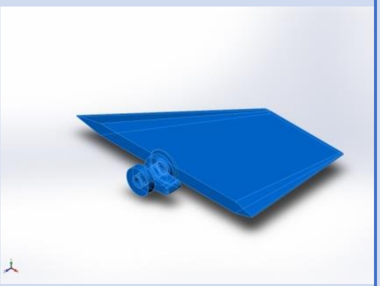
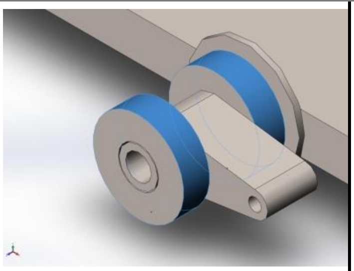
Model Reference	Properties	Components
	Name: Alloy Steel Model type: Linear Elastic Isotropic Default failure criterion: Max von Mises Stress Yield strength: 6.20422e+08 N/m^2 Tensile strength: 7.23826e+08 N/m^2 Elastic modulus: 2.1e+11 N/m^2 Poisson's ratio: 0.28 Mass density: 7,700 kg/m^3 Shear modulus: 7.9e+10 N/m^2 Thermal expansion coefficient: 1.3e-05 /Kelvin	Solid Body 1(Boss-Extrude11) (aerospace fin new-1), Solid Body 1(Boss-Extrude1) (bearing-3), Solid Body 1(Boss-Extrude1) (bearing-4)

Table 4: Loads and Fixtures

Fixture name	Fixture Image	Fixture Details
Fixed-1		Entities: 2 face(s) Type: Fixed Geometry
Load name	Load Image	Load Details

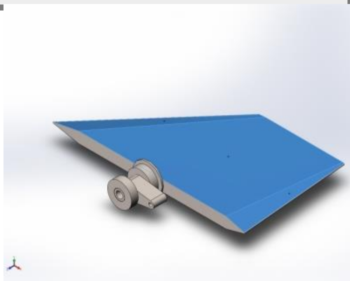
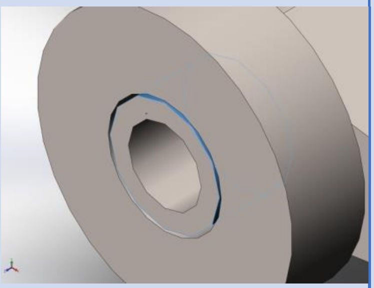
Fixture name	Fixture Image	Fixture Details
Force-1		Entities: 3 face(s) Type: Apply normal force Value: 15,000 N

Table 5: Connector Definitions

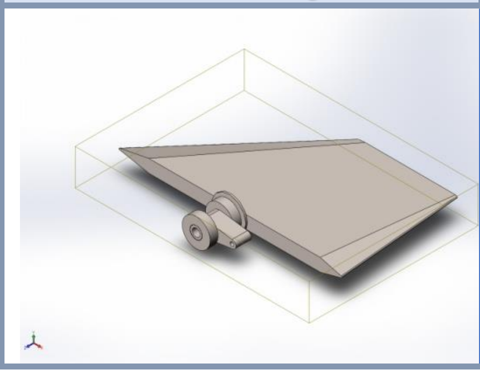
Pin/Bolt/Bearing Connector

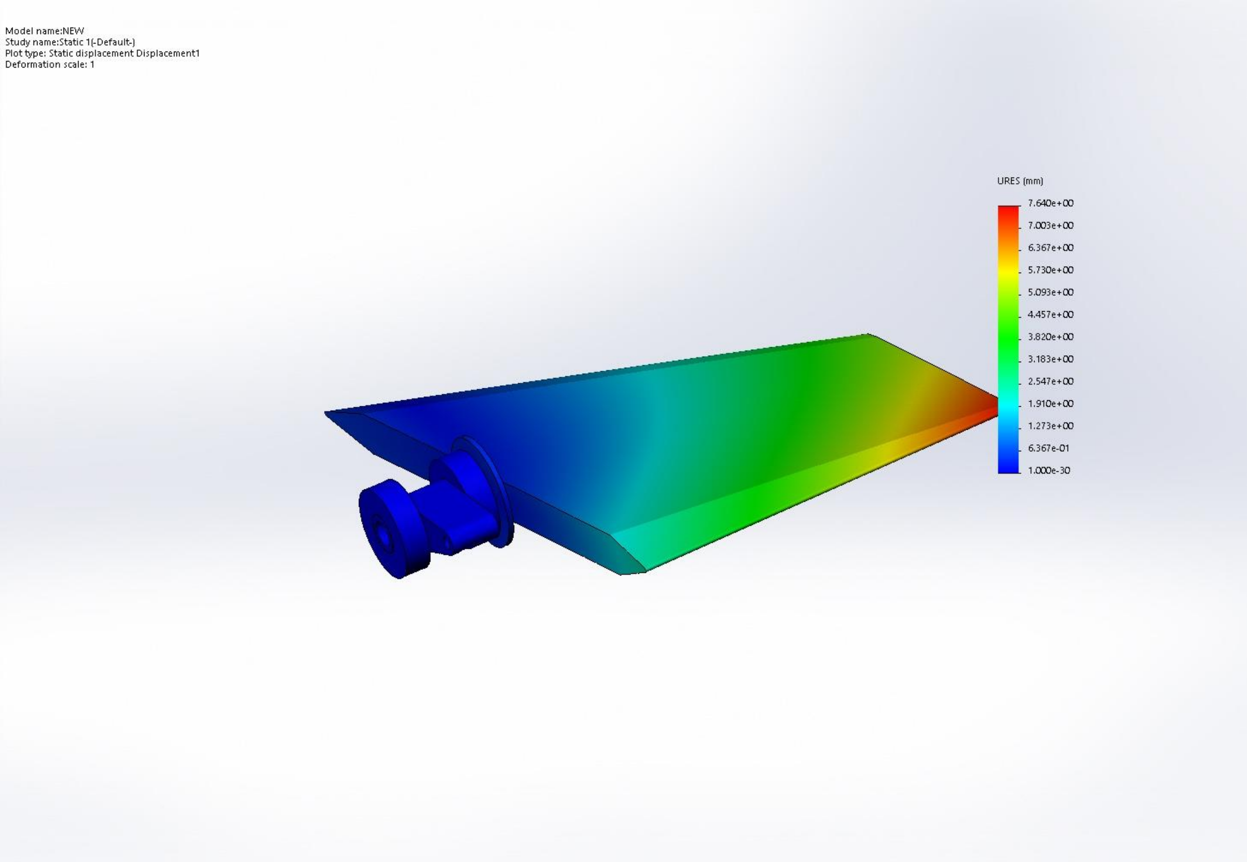
Model Reference	Connector Details	Strength Details
 Bearing Support-1	Entities: 1 face(s) Type: Bearing	No Data

Connector Forces

Type	X-Component	Y-Component	Z-Component	Resultant
Axial Force (N)	-0	-0	-2,321.7	-2,321.7
Shear Force (N)	-923.35	-18,412	0	18,435
Bending moment (N.m)	0	0	0	0

Table 6: Contact Information

Contact	Contact Image	Contact Properties
Global Contact		Type: Bonded Components: 1 component(s) Options: Compatible mesh



1

Figure 3: Static-deformation analysis

Table 7: Mesh information

Mesh type	Solid Mesh
Mesher Used:	Standard mesh
Automatic Transition:	Off
Include Mesh Auto Loops:	Off
Jacobian points	4 Points
Element Size	10.6495 mm
Tolerance	0.532476 mm
Mesh Quality Plot	High

Table 8: Mesh information – Details

Total Nodes	32031
Total Elements	18814
Maximum Aspect Ratio	16.861
% of elements with Aspect Ratio < 3	93.9
% of elements with Aspect Ratio > 10	0.0159
% of distorted elements(Jacobian)	0
Time to complete mesh(hh;mm;ss):	00:00:03

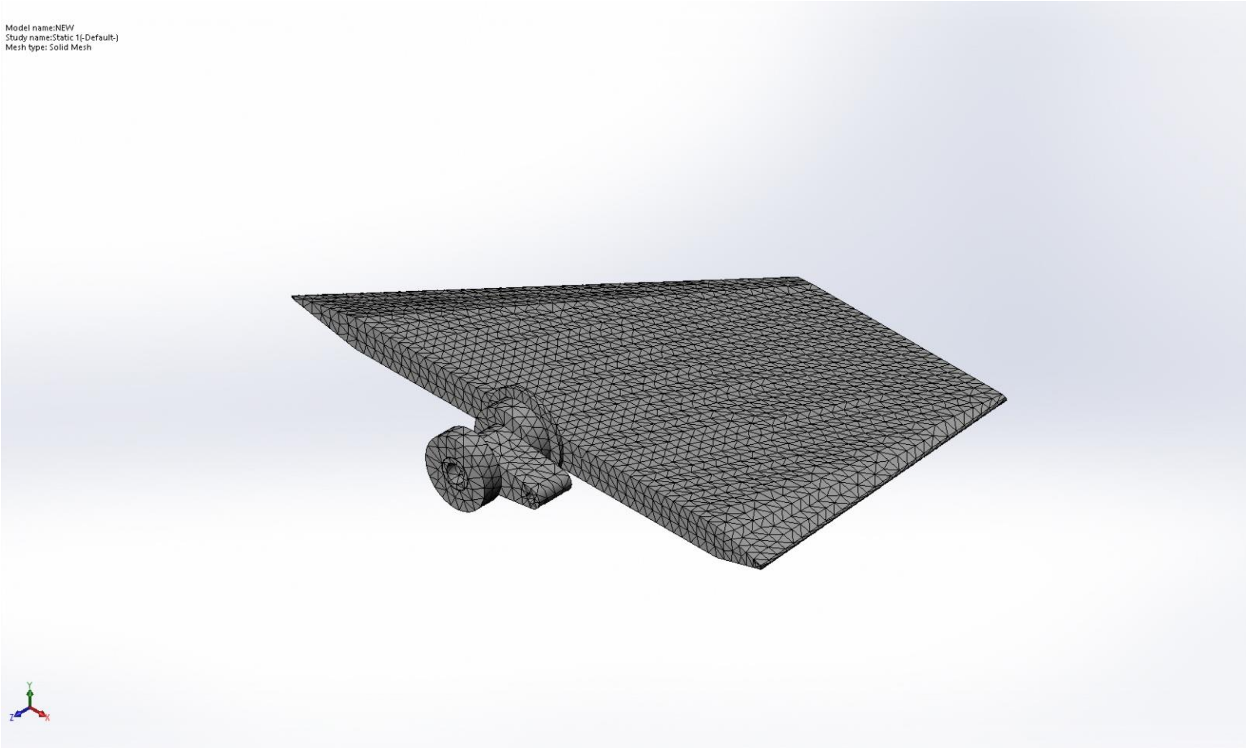


Figure 4a: Meshed Aerospace fin

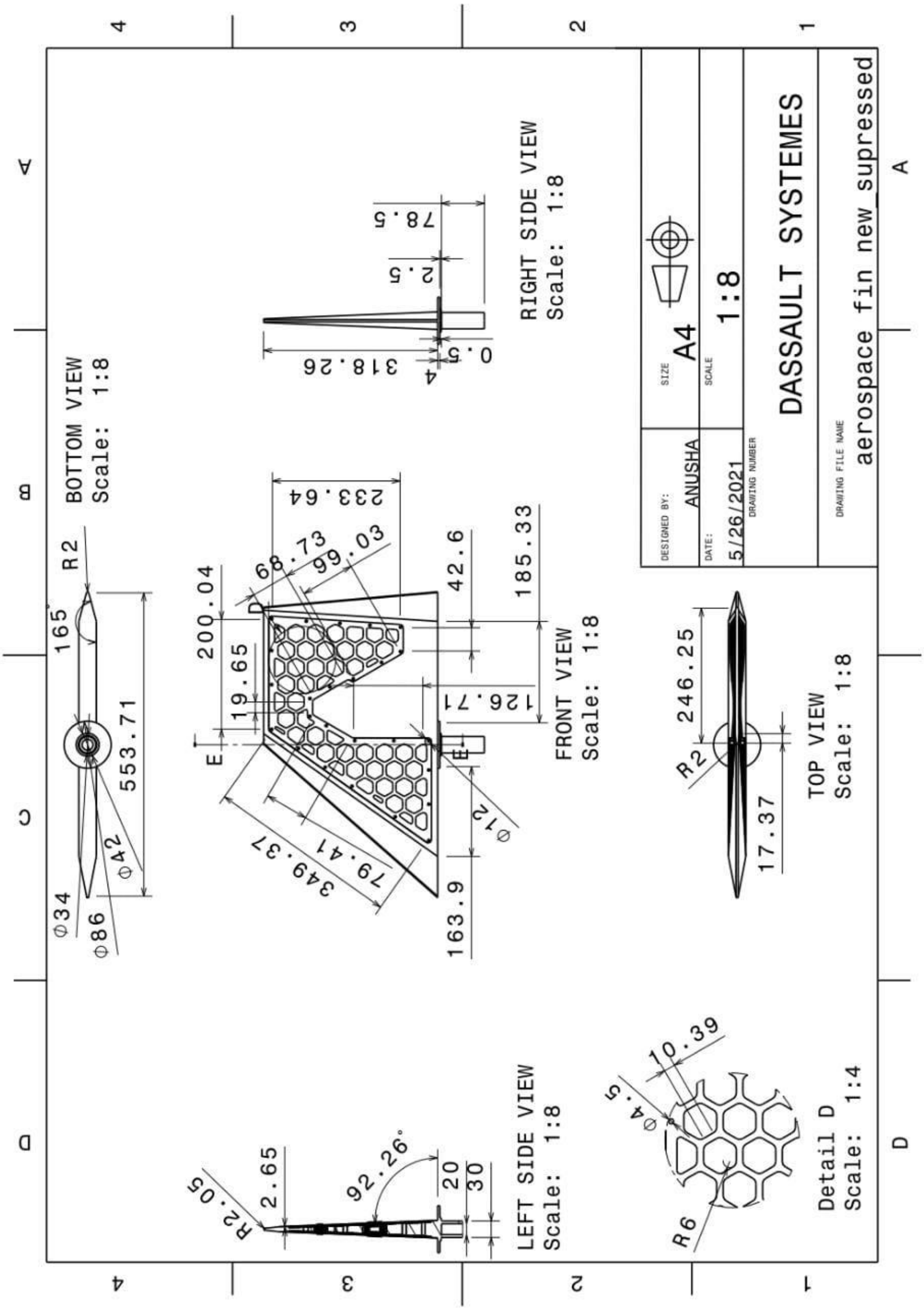


Figure 4b: DRAFTING FILE OF AEROSPACE FIN

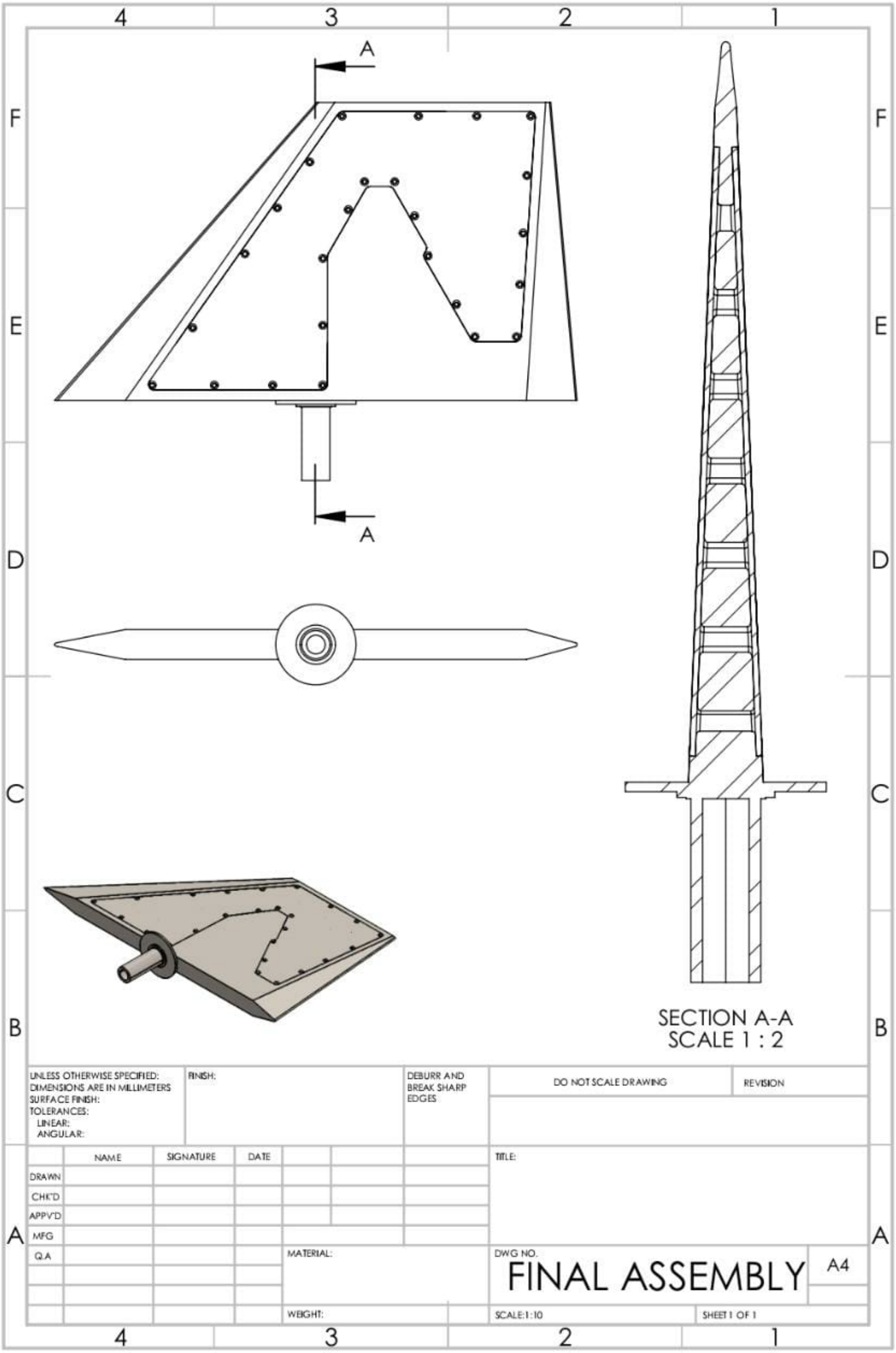
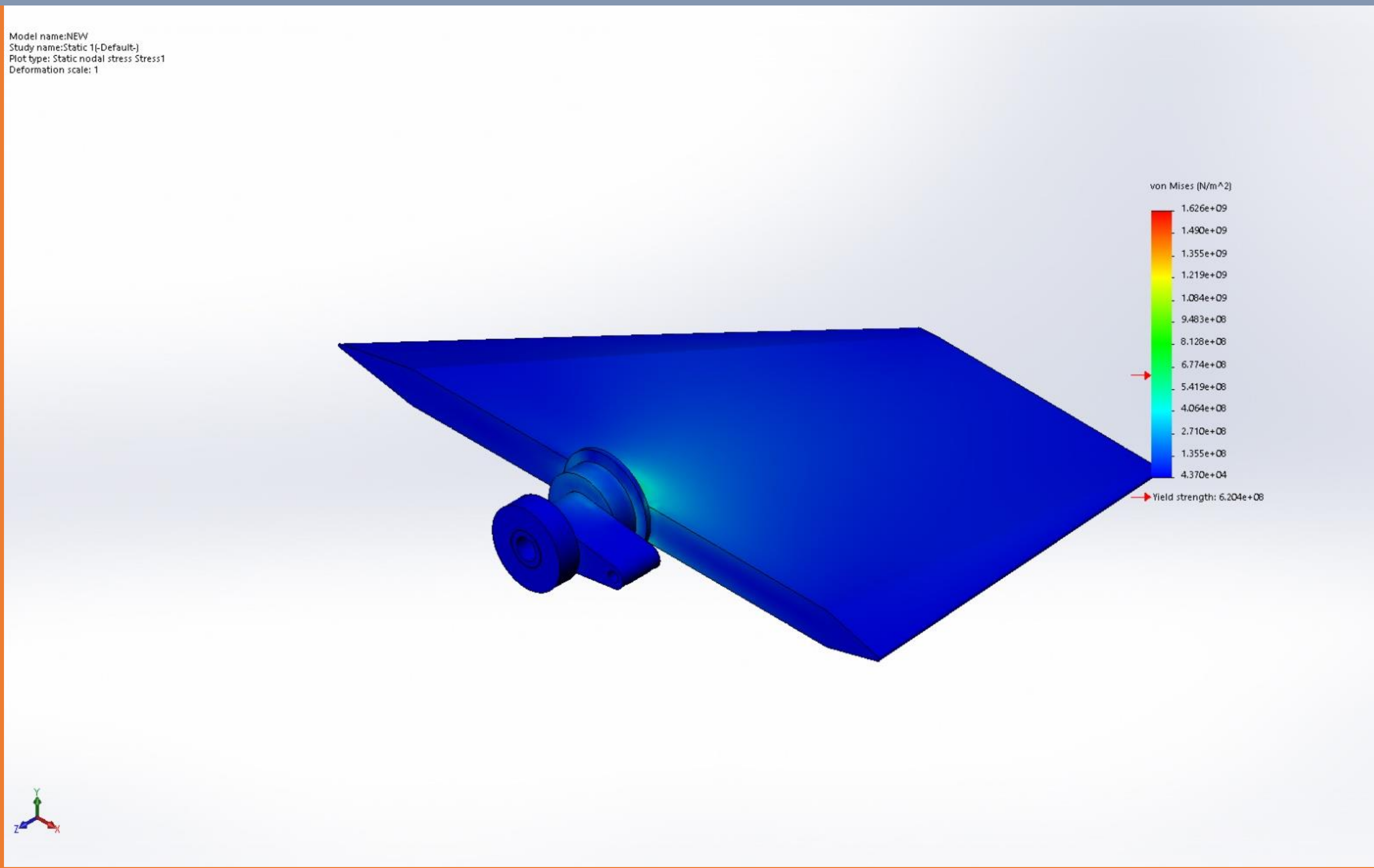


Figure 4c: Drafting file of Final Assembly

Table 9: Stress analysis in solidworks

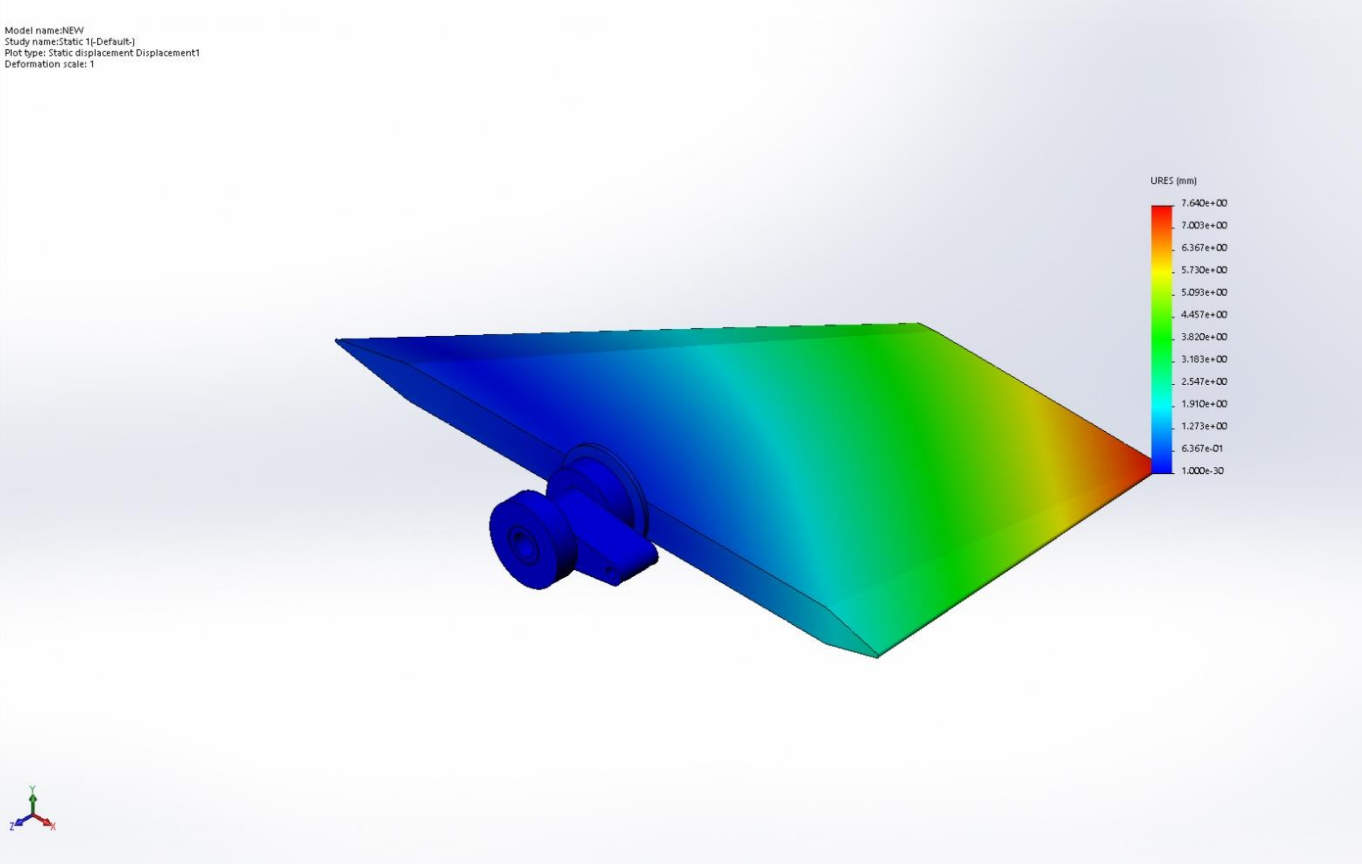
Name	Type	Min	Max
Stress1	VON: von Mises Stress	4.370e+04 N/m^2 Node: 29777	1.626e+09 N/m^2 Node: 29953



NEW-Static 1-Stress-Stress1

Table 10: Displacement Analysis

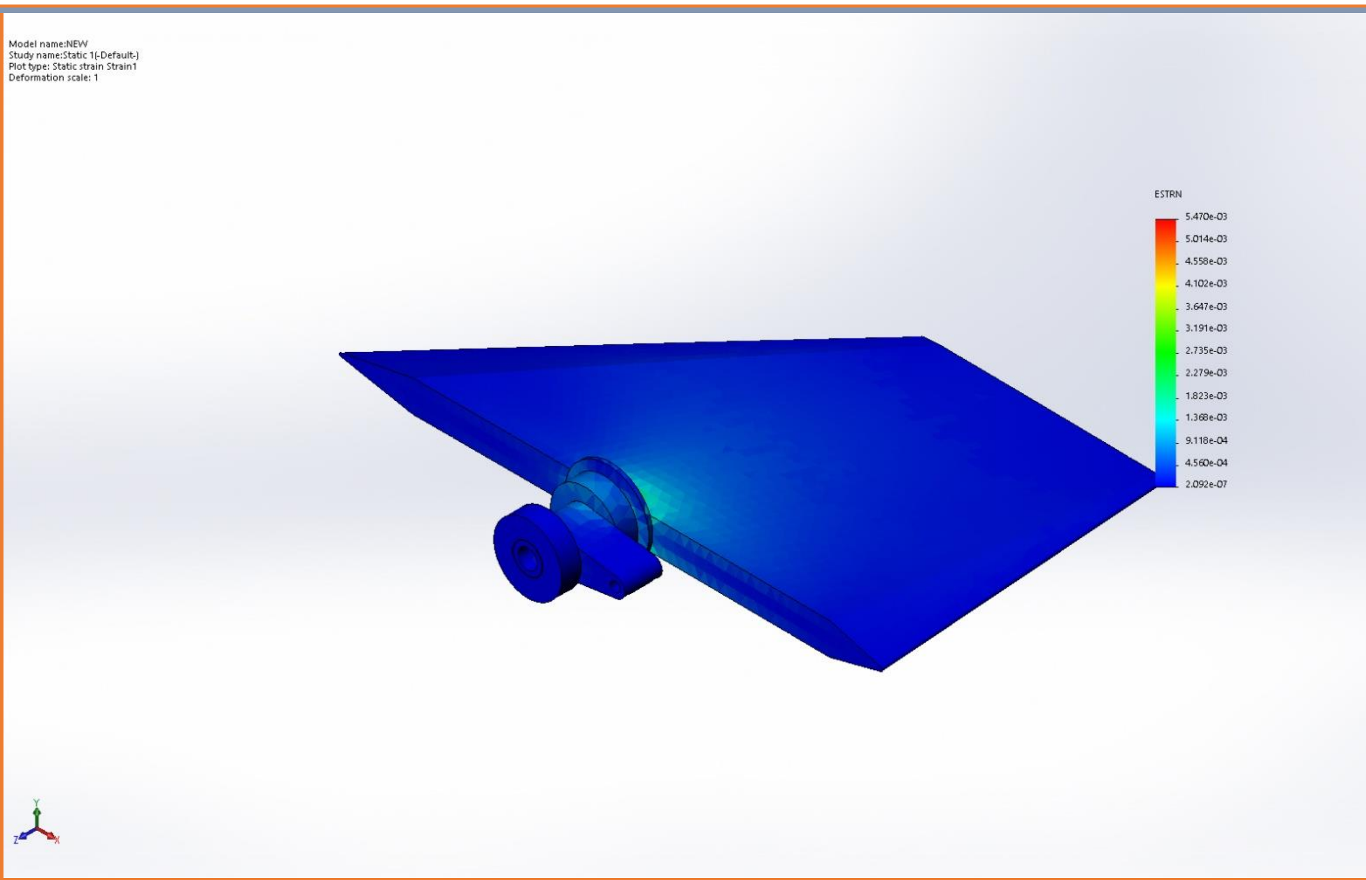
Name	Type	Min	Max
Displacement1	URES: Resultant Displacement	0.000e+00 mm Node: 25	7.640e+00 mm Node: 165



NEW-Static 1-Displacement-Displacement1

Table 11: Equivalent Strain Analysis

Name	Type	Min	Max
Strain1	ESTRN: Equivalent Strain	2.092e-07 Element: 10141	5.470e-03 Element: 7844



NEW-Static 1-Strain-Strain1

Table 12: Strain Analysis in Solidworks

Name	Type	Min	Max
Strain1	ESTRN: Equivalent Strain	2.092e-07 Element: 10141	5.470e-03 Element: 7844

Fin after topology optimization

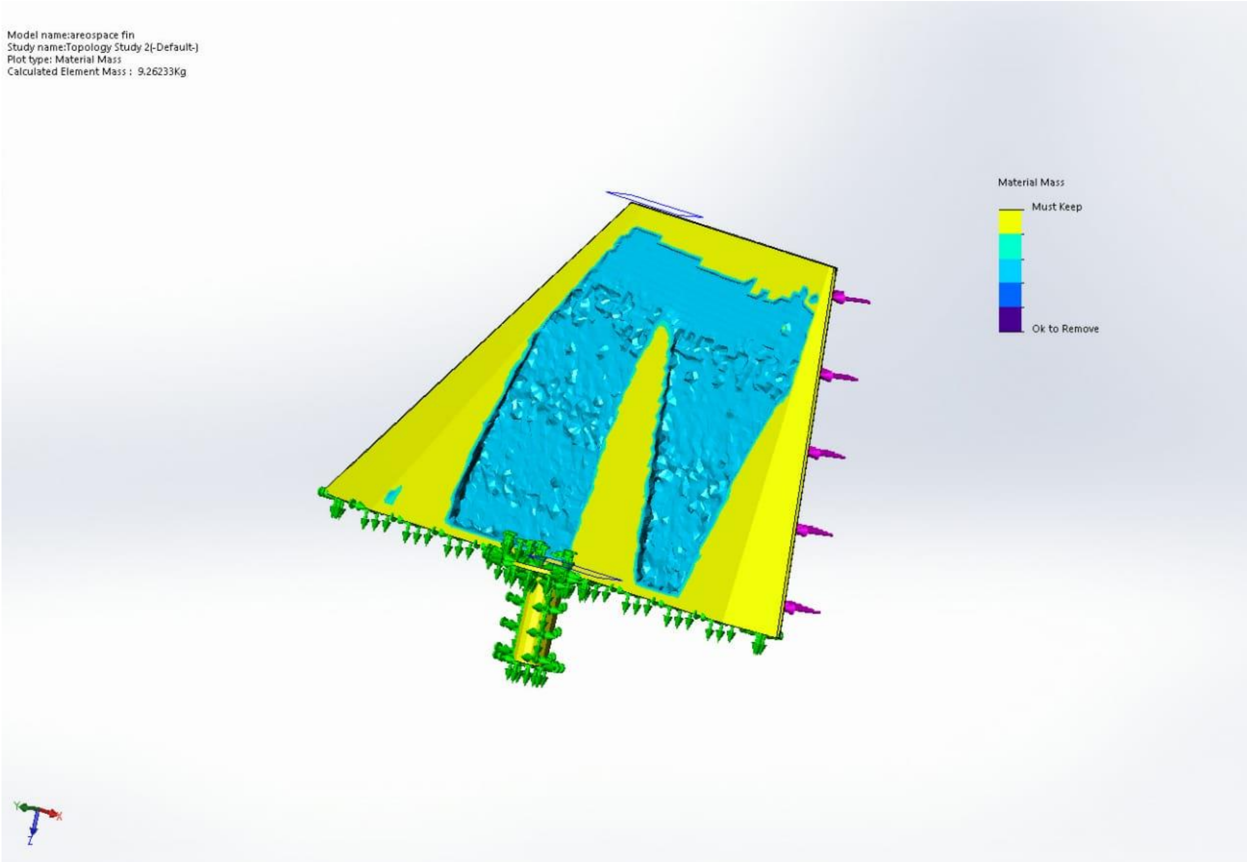


Figure 5: Optimized fin in Solidworks

CHAPTER 3
TOPOLOGY OPTIMIZATION USING ANSYS

3.1 GENERATIVE DESIGN

Generative design is a design process that involves a program that will generate a certain number of outputs that meet certain constraints, and a designer that will fine tune the feasible region by selecting specific output or changing input values, ranges and distribution. The designer doesn't need to be a human, it can be a test program in a testing environment or an artificial intelligence. Here NETFABB is used to define the lattice commander and lattice structure.

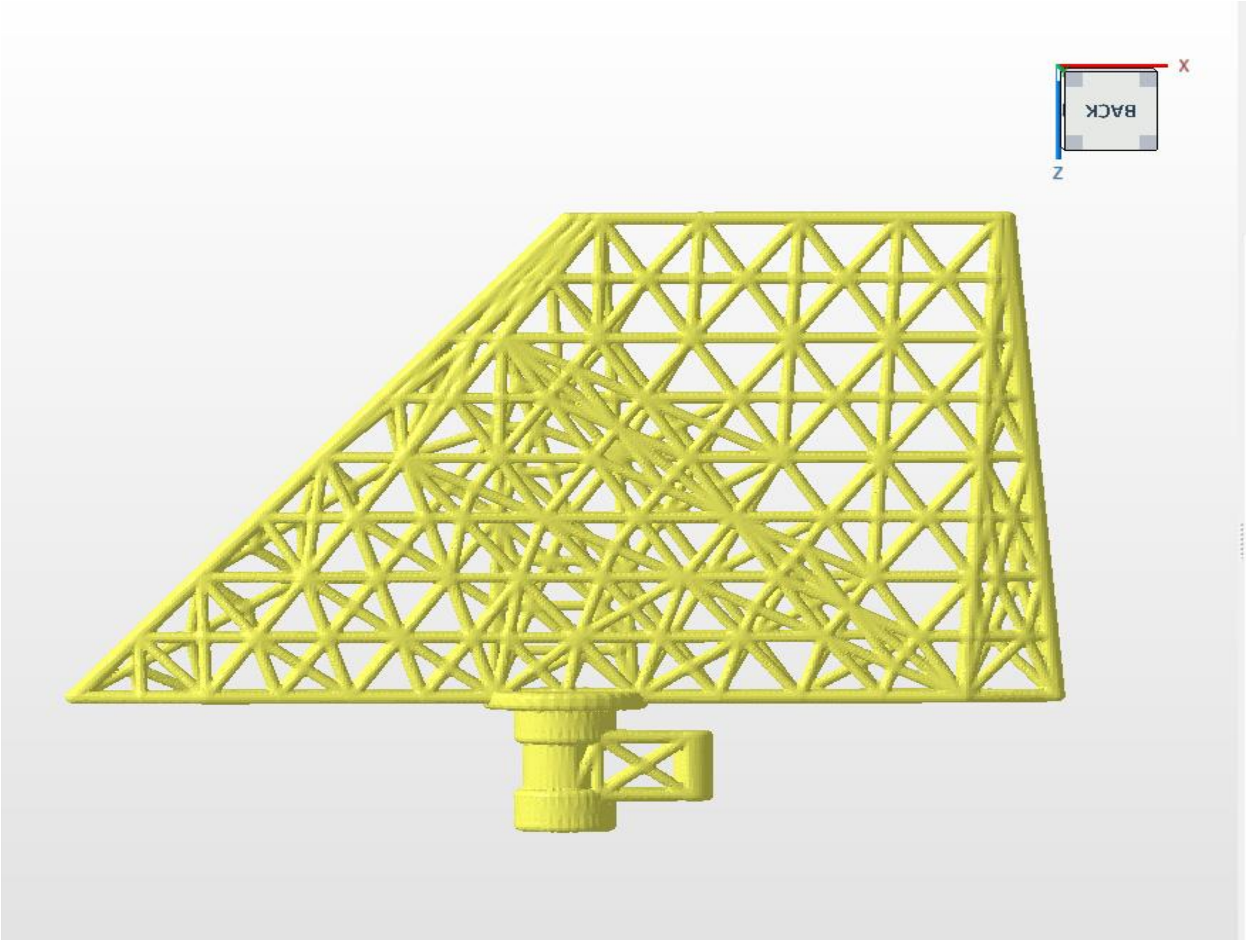


Figure 6: Lattice structure of Aerospace Fin (NETFABB)

3.2 STATIC ANALYSIS

Units

Unit System	Metric (m, kg, N, s, V, A) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

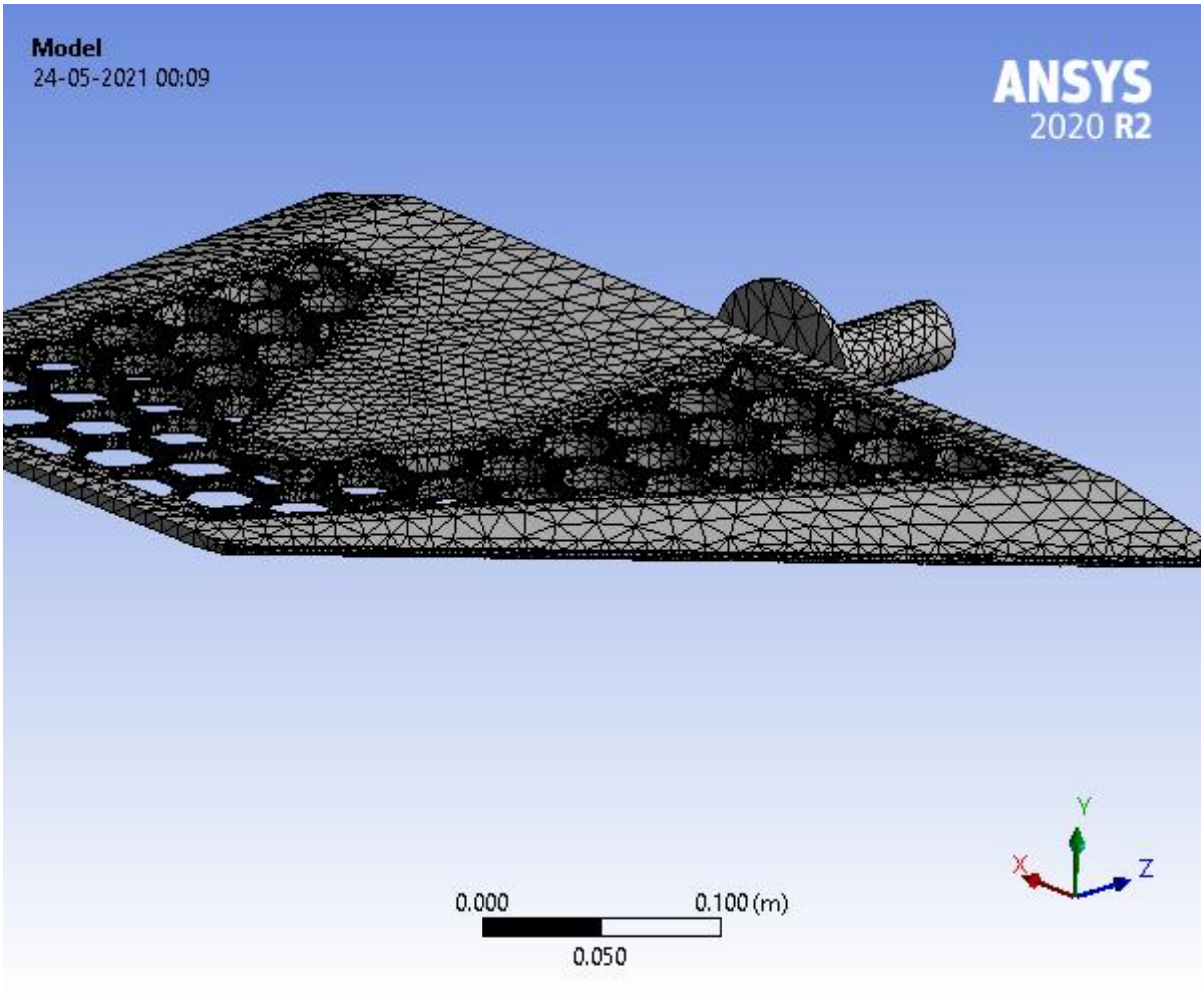


Figure 7: Optimized fin in Ansys

Table 13: Geometry (ANSYS)

Object Name	AEROSPACE FIN
State	Fully Defined
Definition	
Type	Step
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Length X	0.55861 m
Length Y	8.6e-002 m
Length Z	0.40376 m
Properties	
Volume	1.7484e-003 m^3

Mass	13.681 kg
Scale Factor Value	1.
Statistics	
Bodies	1
Active Bodies	1
Nodes	170984
Elements	93612
Mesh Metric	None
Update Options	
Basic Geometry Options	
Solid Bodies	Yes
Surface Bodies	Yes
Line Bodies	No
Parameters	Independent
Parameter Key	ANS;DS

Table 14: Mesh, Material and Properties information (ANSYS)

Object Name	*aerospace fin*
State	Meshed
Graphics Properties	
Visible	Yes
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Treatment	None
Material	
Assignment	Low alloy steel, 4130, cast, hardened & tempered
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	0.55861 m
Length Y	8.6e-002 m
Length Z	0.40376 m
Properties	
Volume	1.7484e-003 m^3
Mass	13.681 kg
Centroid X	4.6985e-002 m
Centroid Y	-1.2695e-008 m
Centroid Z	-0.13544 m

Moment of Inertia Ip1	7.8341e-002 kg·m²
Moment of Inertia Ip2	0.30689 kg·m²
Moment of Inertia Ip3	0.22993 kg·m²
Statistics	
Nodes	170984
Elements	93612
Mesh Metric	None

Table 15: Material assignment

Object Name	*Low alloy steel, 4130, cast, hardened & tempered Assignment*
State	Fully Defined
General	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Material Name	Low alloy steel, 4130, cast, hardened & tempered
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Reference Temperature	By Environment
Suppressed	No

3.3 Coordinate Systems

Table 16: Coordinate System

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0.
Origin	
Origin X	0. m
Origin Y	0. m
Origin Z	0. m
Directional Vectors	
X Axis Data	[1. 0. 0.]
Y Axis Data	[0. 1. 0.]
Z Axis Data	[0. 0. 1.]

Table 17: Mesh sizing

Object Name	*Mesh*
State	Solved
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	1.e-002 m
Sizing	
Use Adaptive Sizing	Yes
Resolution	Default (2)
Mesh Defeaturing	Yes
Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	0.6946 m
Average Surface Area	1.5654e-004 m²
Minimum Edge Length	3.004e-004 m
Quality	
Error Limits	Aggressive Mechanical
Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Straight Sided Elements	No
Rigid Body Behavior	Dimensionally Reduced
Triangle Surface Mesher	Program Controlled
Topology Checking	Yes
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No

Statistics	
Nodes	170984
Elements	93612

3.4 Static Structural Analysis

A static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects.

Table 18: Static Structural Analysis

Object Name	*Static Structural (A5)*
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No

Table 19: Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Quasi-Static Solution	Off
Rotordynamics Controls	
Coriolis Effect	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	No
Combine Restart Files	Program Controlled

Nonlinear Controls	
Newton-Raphson Option	Program Controlled
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Program Controlled

Table 20: Accelerations

Object Name	*Standard Earth Gravity*
State	Fully Defined
Scope	
Geometry	All Bodies
Definition	
Coordinate System	Global Coordinate System
X Component	0. m/s² (ramped)
Y Component	-9.8066 m/s² (ramped)
Z Component	0. m/s² (ramped)
Suppressed	No
Direction	-Y Direction

Figure 8: Standard Earth Gravity

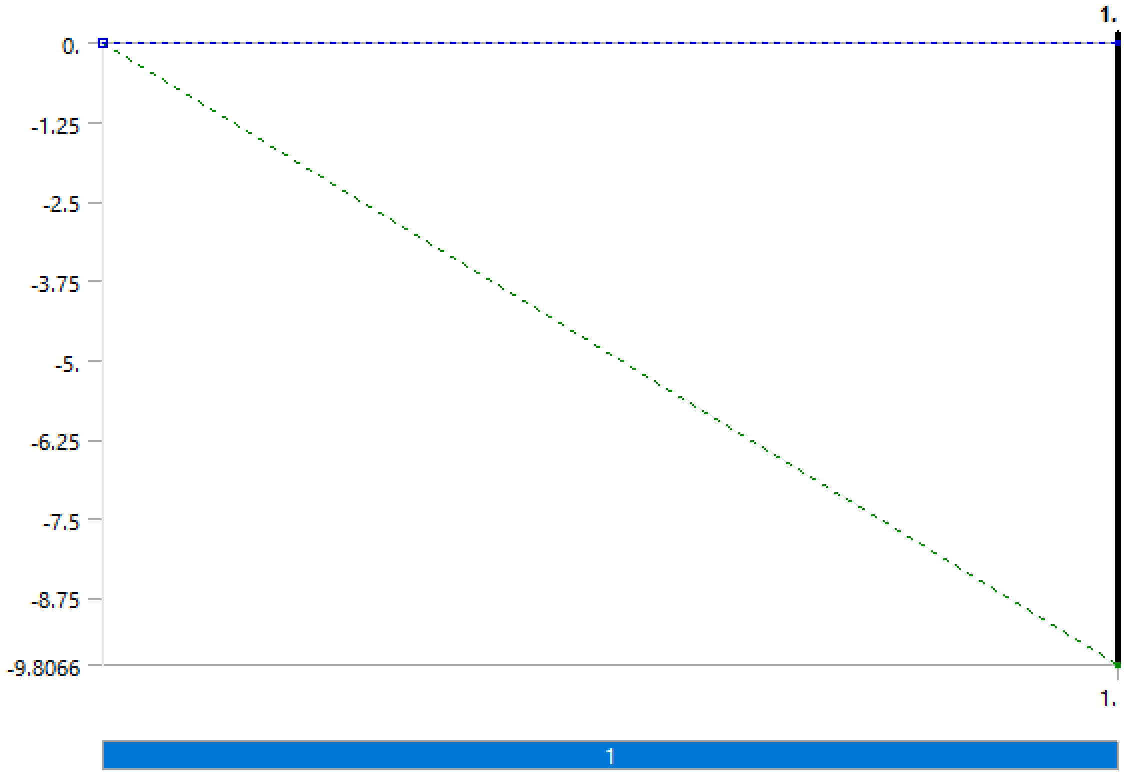


Table 21: Loads

Object Name	*Fixed Support*	*Force*
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Geometry	19 Faces	2 Faces
Definition		
Type	Fixed Support	Force
Suppressed	No	
Define By		Components
Applied By		Surface Effect
Coordinate System		Global Coordinate System
X Component		0. N (ramped)
Y Component		-15000 N (ramped)
Z Component		0. N (ramped)

Figure 9: Force

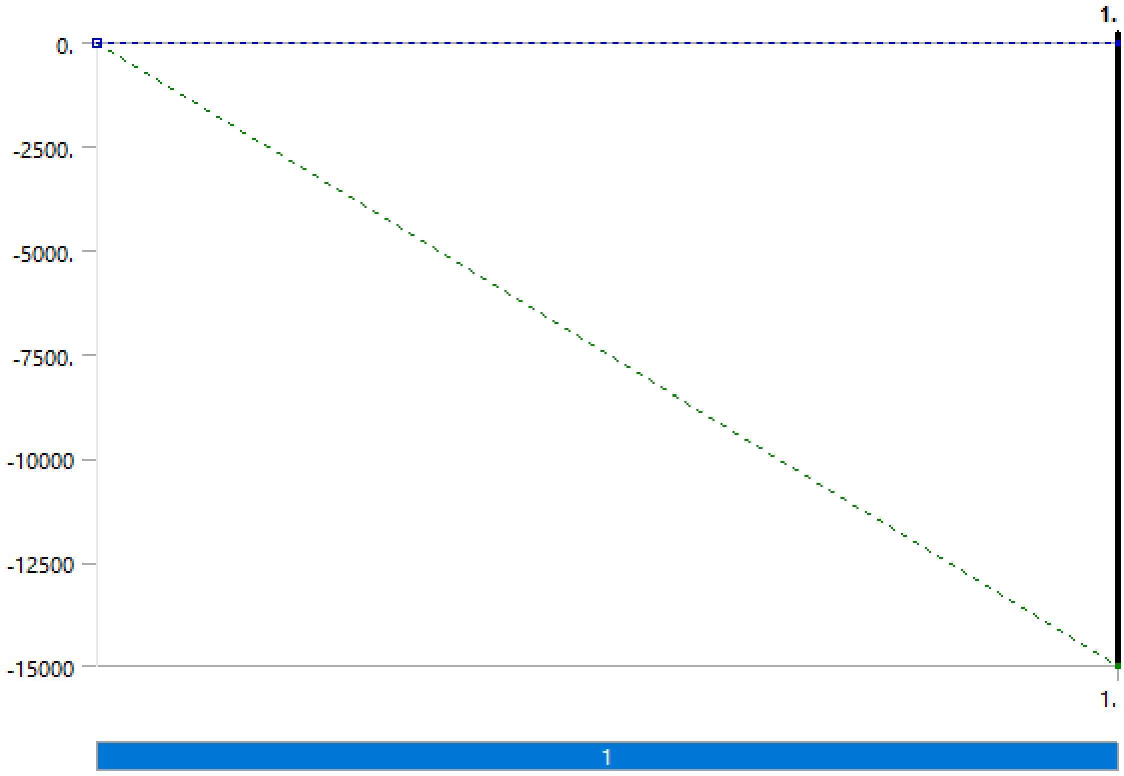


Table 22: Solution

Object Name	*Solution (A6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done
MAPDL Elapsed Time	2 m 26 s
MAPDL Memory Used	964. MB
MAPDL Result File Size	171.94 MB
Post Processing	
Beam Section Results	No
On Demand Stress/Strain	No

CHAPTER 3
NON-LINEAR ANALYSIS

Units

Unit System	Metric (m, kg, N, s, V, A) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

Geometry

Table 27: Geometry

Object Name	Geometry
State	Fully Defined
Definition	
Source	Topology Optimization\aerospace fin
Type	Step
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Length X	0.55845 m
Length Y	8.6e-002 m
Length Z	0.40376 m
Properties	
Volume	2.5225e-003 m^3
Mass	19.739 kg
Scale Factor Value	1.
Statistics	
Bodies	1
Active Bodies	1
Nodes	10424
Elements	5236
Mesh Metric	None
Update Options	
Assign Default Material	No
Basic Geometry Options	
Solid Bodies	Yes
Surface Bodies	Yes
Line Bodies	No

Parameters	Independent
Parameter Key	ANS; DS
Attributes	No
Named Selections	No
Material Properties	No
Advanced Geometry Options	
Use Associativity	Yes
Coordinate Systems	No
Reader Mode Saves Updated File	No
Use Instances	Yes
Smart CAD Update	Yes
Compare Parts On Update	No
Analysis Type	3-D
Mixed Import Resolution	None
Clean Bodies On Import	No
Stitch Surfaces On Import	Program Tolerance
Decompose Disjoint Geometry	Yes
Enclosure and Symmetry Processing	Yes

Table 28: Parts

Object Name	aerospace fin
State	Meshed
Graphics Properties	
Visible	Yes
Transparency	1
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Treatment	None
Material	
Assignment	Low alloy steel, 4130, cast, hardened & tempered
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	0.55845 m
Length Y	8.6e-002 m
Length Z	0.40376 m
Properties	
Volume	2.5225e-003 m^3
Mass	19.739 kg
Centroid X	4.5266e-002 m
Centroid Y	-4.0672e-009 m

Centroid Z	-0.15243 m
Moment of Inertia Ip1	0.11984 kg·m²
Moment of Inertia Ip2	0.43545 kg·m²
Moment of Inertia Ip3	0.31757 kg·m²
Statistics	
Nodes	10424
Elements	5236
Mesh Metric	None

Coordinate Systems

Table 29: Coordinate System

Object Name	Global Coordinate System
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0.
Origin	
Origin X	0. m
Origin Y	0. m
Origin Z	0. m
Directional Vectors	
X Axis Data	[1. 0. 0.]
Y Axis Data	[0. 1. 0.]
Z Axis Data	[0. 0. 1.]

Mesh

Table 30: Mesh

Object Name	Mesh
State	Solved
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	1.e-002 m
Sizing	
Use Adaptive Sizing	Yes
Resolution	Default (2)
Mesh Defeaturing	Yes
Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	0.69447 m

Average Surface Area	1.0649e-002 m²
Minimum Edge Length	5.e-004 m
Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	None
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Straight Sided Elements	No
Rigid Body Behavior	Dimensionally Reduced
Triangle Surface Mesher	Program Controlled
Topology Checking	Yes
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Statistics	
Nodes	10424
Elements	5236

Static Structural Analysis

Table 31: Analysis

Object Name	Static Structural
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL
Options	
Environment Temperature	22. °C
Generate Input Only	No

Table 32: Analysis Settings

Object Name	*Analysis Settings*
State	Fully Defined
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Substeps
Initial Substeps	10.
Minimum Substeps	5.
Maximum Substeps	10.
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Quasi-Static Solution	On
Rotordynamics Controls	
Coriolis Effect	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	No
Combine Restart Files	Program Controlled
Nonlinear Controls	
Newton-Raphson Option	Program Controlled
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Program Controlled
Advanced	
Inverse Option	No
Contact Split (DMP)	Off
Output Controls	
Stress	Yes
Surface Stress	No
Back Stress	No
Strain	Yes
Contact Data	Yes
Nonlinear Data	No
Nodal Forces	No
Volume and Energy	Yes
Euler Angles	Yes

General Miscellaneous	No
Contact Miscellaneous	No
Store Results At	All Time Points
Result File Compression	Program Controlled
Analysis Data Management	
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No
Contact Summary	Program Controlled
Delete Unneeded Files	Yes
Nonlinear Solution	Yes
Solver Units	Active System
Solver Unit System	MKS

Table 33: Accelerations

Object Name	*Standard Earth Gravity*
State	Fully Defined
Scope	
Geometry	All Bodies
Definition	
Coordinate System	Global Coordinate System
X Component	0. m/s² (ramped)
Y Component	-9.8066 m/s² (ramped)
Z Component	0. m/s² (ramped)
Suppressed	No
Direction	-Y Direction

Table 34: Loads

Object Name	*Fixed Support*	*Force*
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Geometry	19 Faces	1 Face
Definition		
Type	Fixed Support	Force
Suppressed	No	
Define By		Components
Applied By		Surface Effect

Coordinate System		Global Coordinate System
X Component		0. N (ramped)
Y Component		-15000 N (ramped)
Z Component		0. N (ramped)

Solution

Table 35: Solution

Object Name	*Solution (A6)*
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done
MAPDL Elapsed Time	13. s
MAPDL Memory Used	305. MB
MAPDL Result File Size	20.75 MB
Post Processing	
Beam Section Results	No
On Demand Stress/Strain	No

Table 36: Results

Object Name	*Total Deformation*	*Equivalent Elastic Strain*	*Equivalent Stress*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Deformation	Equivalent Elastic Strain	Equivalent (von-Mises) Stress
By	Time		
Display Time	Last		
Calculate Time History	Yes		
Identifier			
Suppressed	No		
Results			
Minimum	0. m	2.5278e-011 m/m	4.2507 Pa
Maximum	6.2241e-004 m	1.8719e-004 m/m	3.7532e+007 Pa
Average	1.096e-004 m	5.7518e-005 m/m	9.6972e+006 Pa
Minimum Occurs On	aerospace fin		
Maximum Occurs On	aerospace fin new_supressed-FreeParts|Loft1		

Minimum Value Over Time			
Minimum	0. m	2.5298e-012 m/m	0.42507 Pa
Maximum	0. m	2.5278e-011 m/m	4.2507 Pa
Maximum Value Over Time			
Minimum	6.2229e-005 m	1.872e-005 m/m	3.7534e+006 Pa
Maximum	6.2241e-004 m	1.8719e-004 m/m	3.7532e+007 Pa
Information			
Time	1. s		
Load Step	1		
Substep	7		
Iteration Number	8		
Integration Point Results			
Display Option		Averaged	
Average Across Bodies		No	

Table 37: Total Deformation

Time [s]	Minimum [m]	Maximum [m]	Average [m]
0.1	0.	6.2229e-005	1.0959e-005
0.2		1.2446e-004	2.1918e-005
0.35		2.1782e-004	3.8358e-005
0.55		3.423e-004	6.0278e-005
0.75		4.6679e-004	8.22e-005
0.875		5.446e-004	9.5902e-005
1.		6.2241e-004	1.096e-004

Table 38: Equivalent Elastic Strain

Time [s]	Minimum [m/m]	Maximum [m/m]	Average [m/m]
0.1	2.5298e-012	1.872e-005	5.7508e-006
0.2	5.0593e-012	3.744e-005	1.1502e-005
0.35	8.8526e-012	6.552e-005	2.0129e-005
0.55	1.3909e-011	1.0296e-004	3.1632e-005
0.75	1.8963e-011	1.4039e-004	4.3137e-005
0.875	2.2121e-011	1.6379e-004	5.0327e-005
1.	2.5278e-011	1.8719e-004	5.7518e-005

Table 39: Equivalent Stress

Time [s]	Minimum [Pa]	Maximum [Pa]	Average [Pa]
0.1	0.42507	3.7534e+006	9.6955e+005
0.2	0.85014	7.5069e+006	1.9392e+006
0.35	1.4877	1.3137e+007	3.3936e+006
0.55	2.3379	2.0643e+007	5.333e+006
0.75	3.188	2.815e+007	7.2726e+006
0.875	3.7194	3.2841e+007	8.4849e+006
1.	4.2507	3.7532e+007	9.6972e+006

Figure 19: Total Deformation

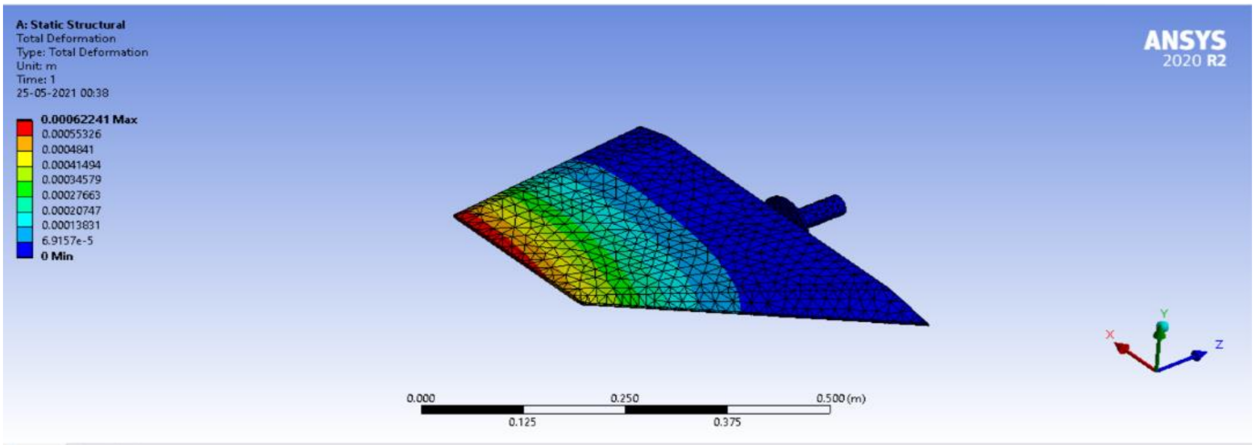


Figure 20: Elastic Strain

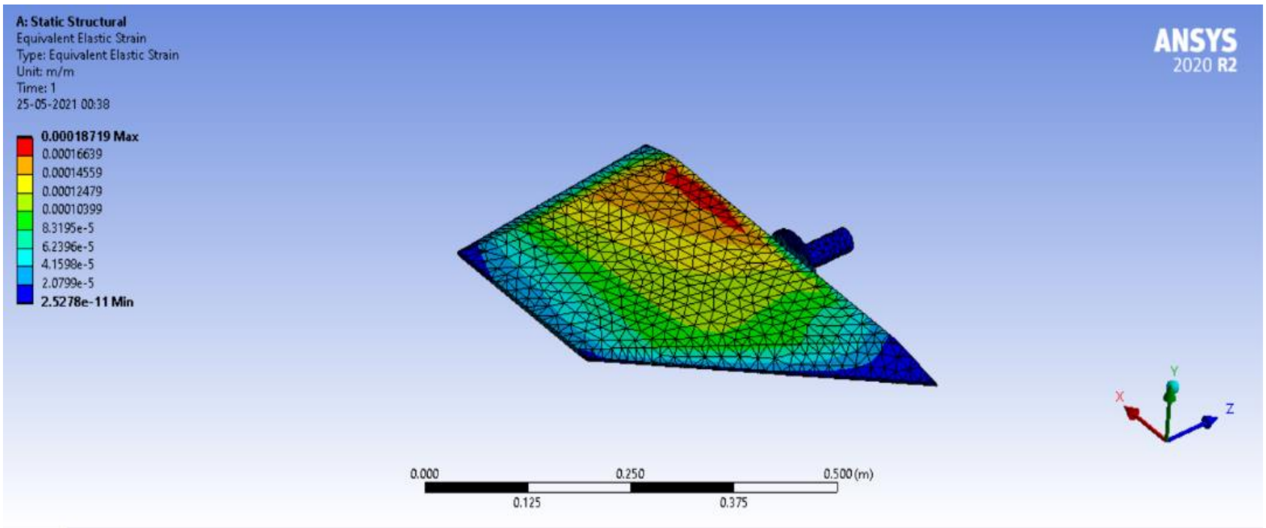
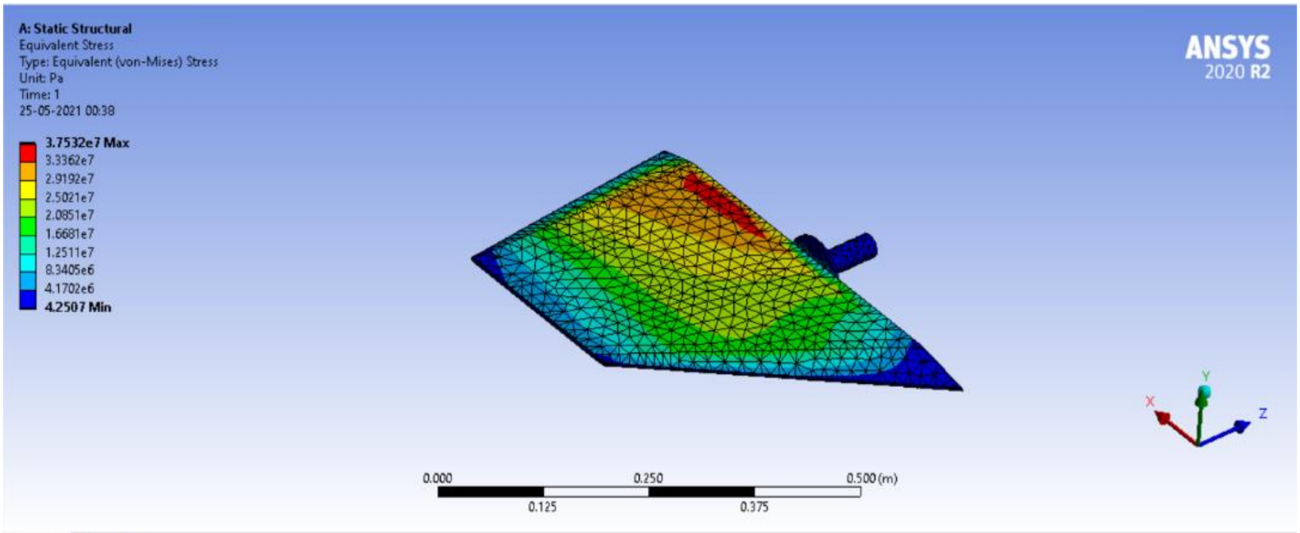


Figure 21: Elastic stress



Total Steps – 10
Min – 5
Max – 10

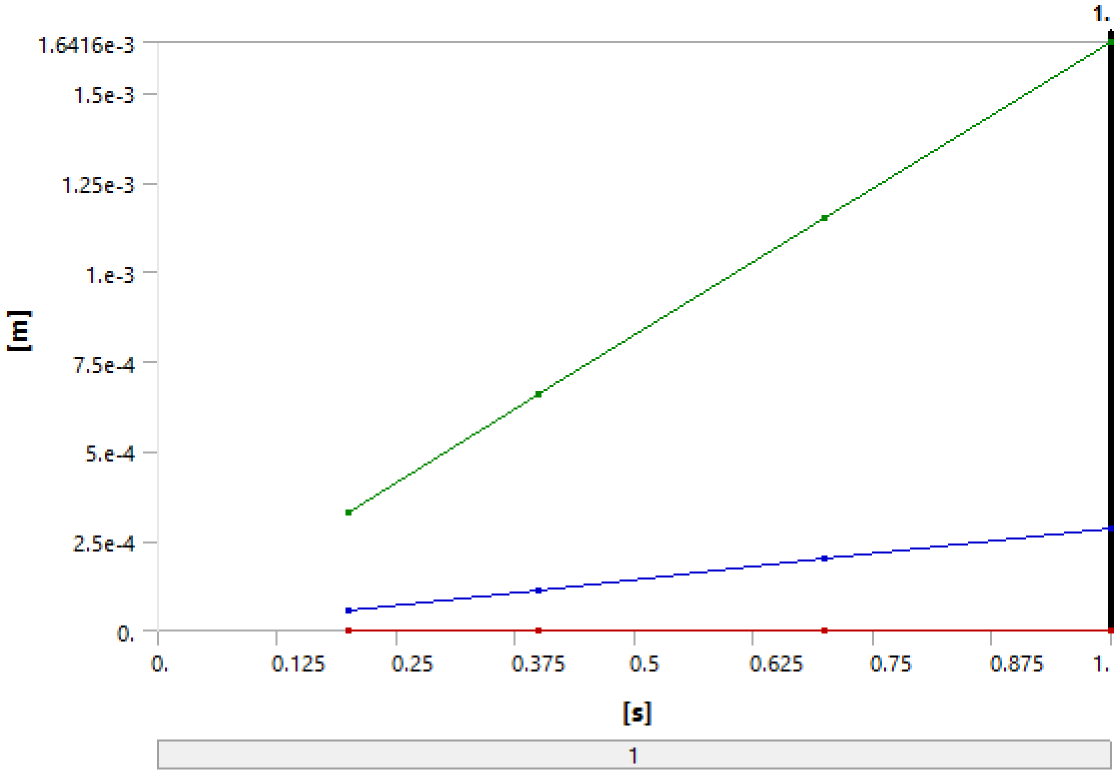
5. **RESULTS**

Table 40: Results

Object Name	*Total Deformation*	*Equivalent Elastic Strain*	*Equivalent Stress*
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		
Definition			
Type	Total Deformation	Equivalent Elastic Strain	Equivalent (von-Mises) Stress
By	Time		
Display Time	Last		
Calculate Time History	Yes		
Identifier			
Suppressed	No		
Results			
Minimum	0. m	2.2269e-011 m/m	2.8444 Pa
Maximum	1.6416e-003 m	1.2302e-003 m/m	2.4714e+008 Pa
Average	2.8617e-004 m	1.4759e-004 m/m	2.7916e+007 Pa
Minimum Occurs On	aerospace fin new_supressed_Hex_optimised-FreeParts|Loft1		
Maximum Occurs On	aerospace fin new_supressed_Hex_optimised-FreeParts|Loft1		
Minimum Value Over Time			
Minimum	0. m	4.4539e-012 m/m	0.56888 Pa
Maximum	0. m	2.2269e-011 m/m	2.8444 Pa
Maximum Value Over Time			
Minimum	3.2833e-004 m	2.4605e-004 m/m	4.9428e+007 Pa
Maximum	1.6416e-003 m	1.2302e-003 m/m	2.4714e+008 Pa
Information			
Time	1. s		
Load Step	1		
Substep	4		
Iteration Number	5		
Integration Point Results			

Display Option		Averaged
Average Across Bodies		No

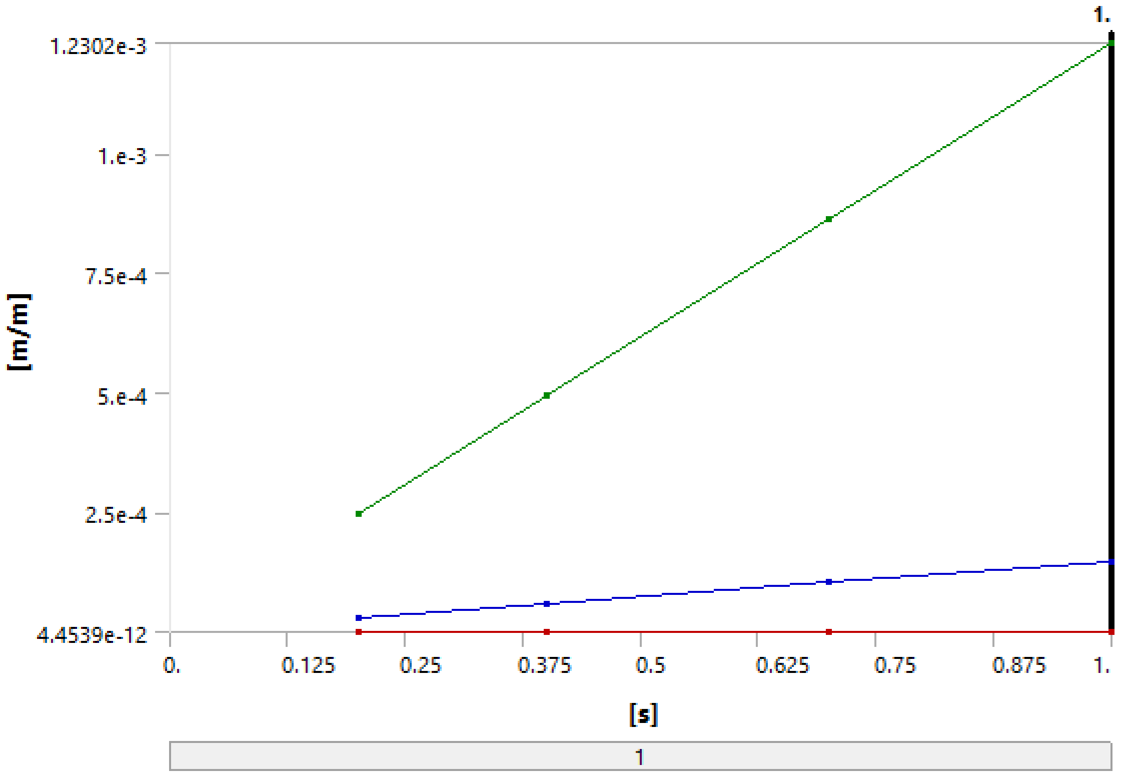
Figure 22: Graph of Total Deformation



Total Deformation

Time [s]	Minimum [m]	Maximum [m]	Average [m]
0.2	0.	3.2833e-004	5.7234e-005
0.4		6.5666e-004	1.1447e-004
0.7		1.1492e-003	2.0032e-004
1.		1.6416e-003	2.8617e-004

Figure 23: Graph of Equivalent Elastic Strain



Equivalent Elastic Strain

Time [s]	Minimum [m/m]	Maximum [m/m]	Average [m/m]
0.2	4.4539e-012	2.4605e-004	2.9519e-005
0.4	8.9078e-012	4.921e-004	5.9037e-005
0.7	1.5589e-011	8.6117e-004	1.0332e-004
1.	2.2269e-011	1.2302e-003	1.4759e-004

Figure 24: Graph of Equivalent Stress

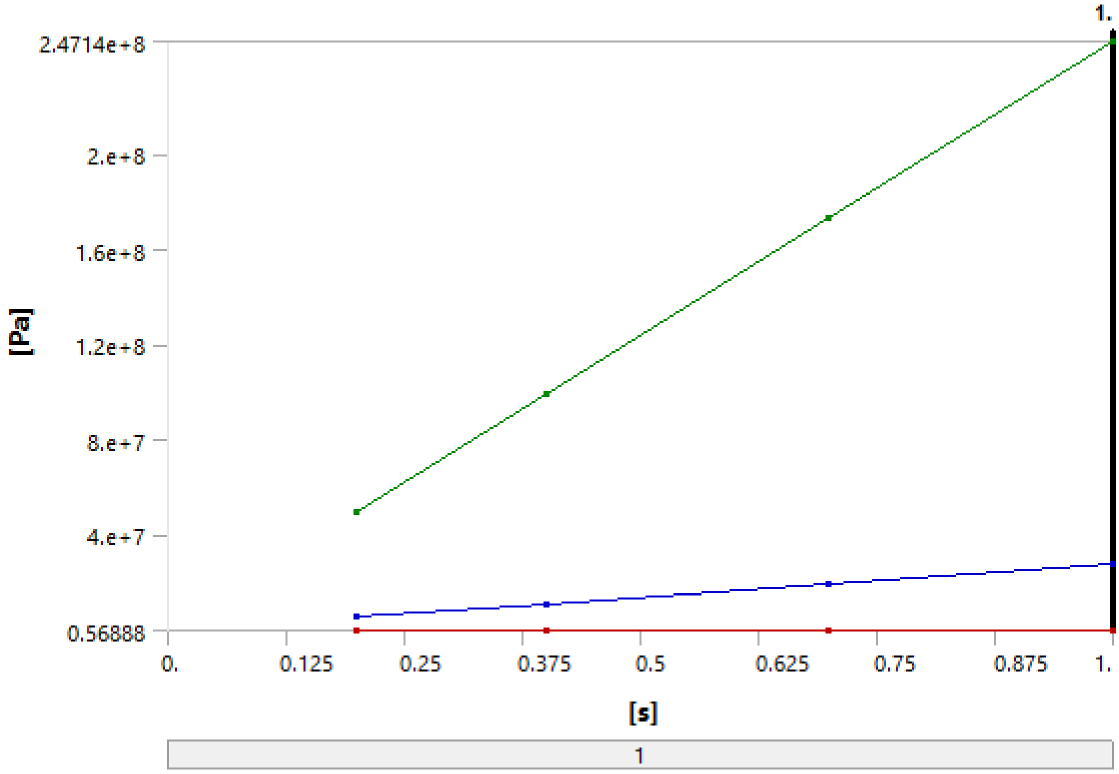


Table 25: Equivalent Stress

Time [s]	Minimum [Pa]	Maximum [Pa]	Average [Pa]
0.2	0.56888	4.9428e+007	5.5831e+006
0.4	1.1378	9.8856e+007	1.1166e+007
0.7	1.9911	1.73e+008	1.9541e+007
1.	2.8444	2.4714e+008	2.7916e+007

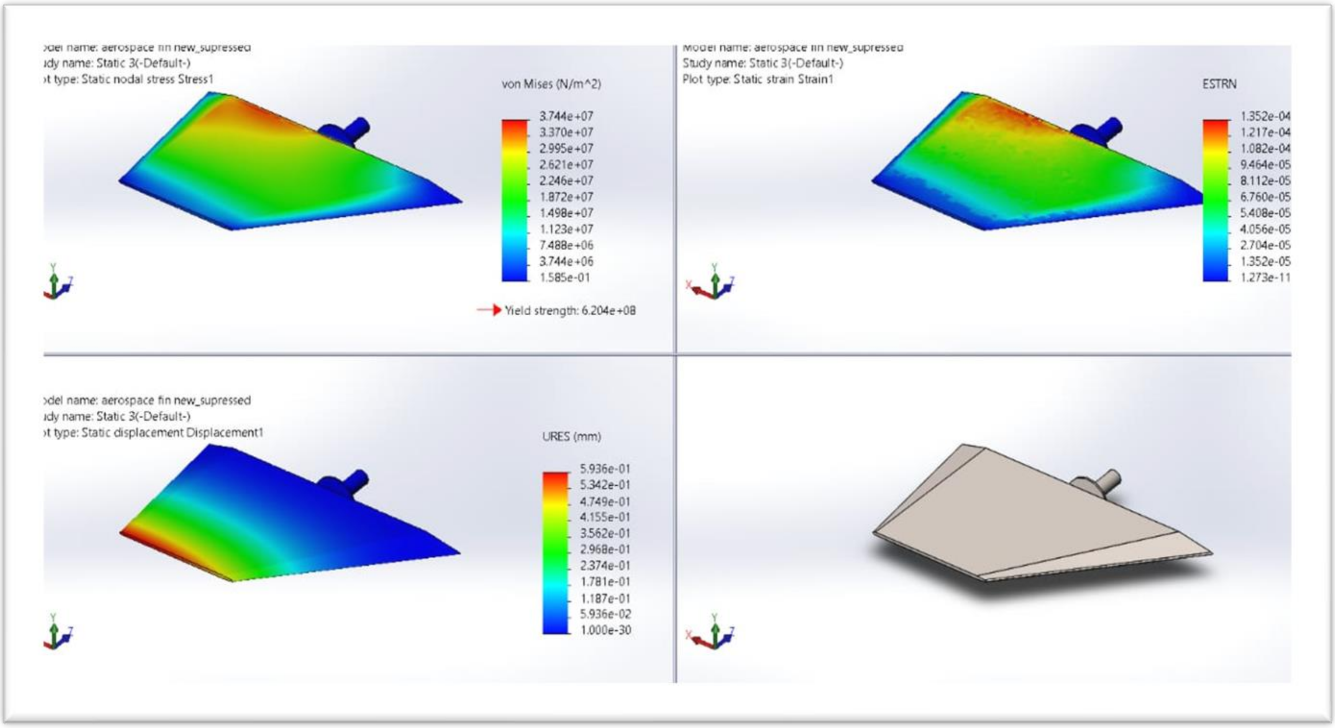


Figure 25: FEM Analysis (Non-linear)

Figure 26: Total deformation (Analysis)

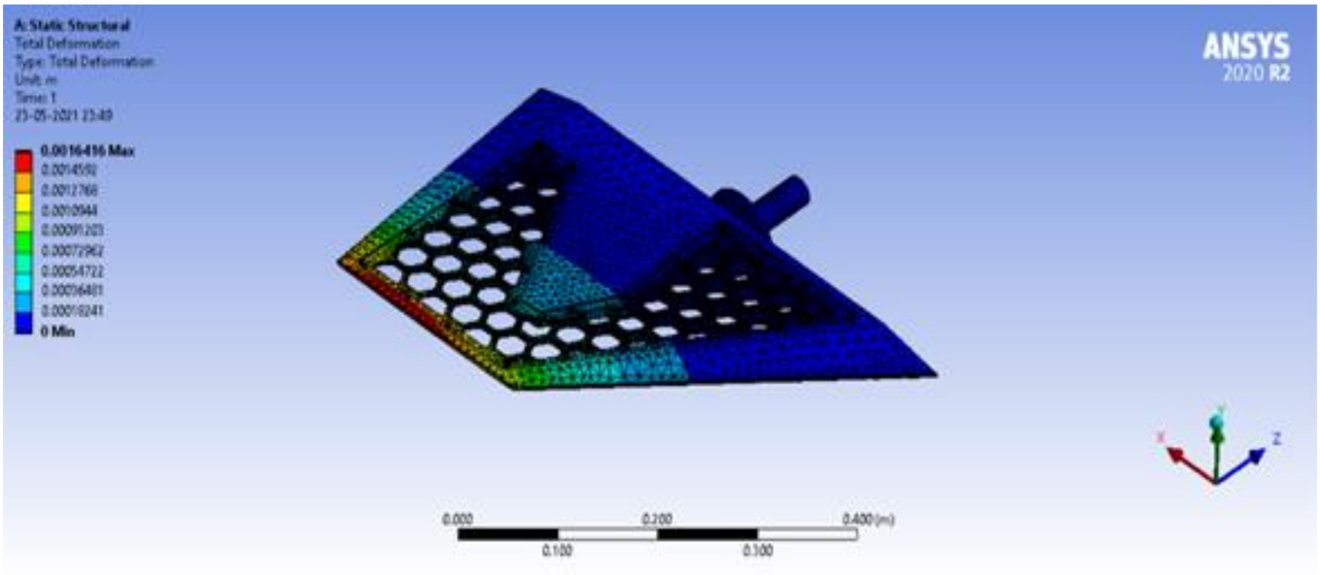


Figure 27: Total Elastic Strain

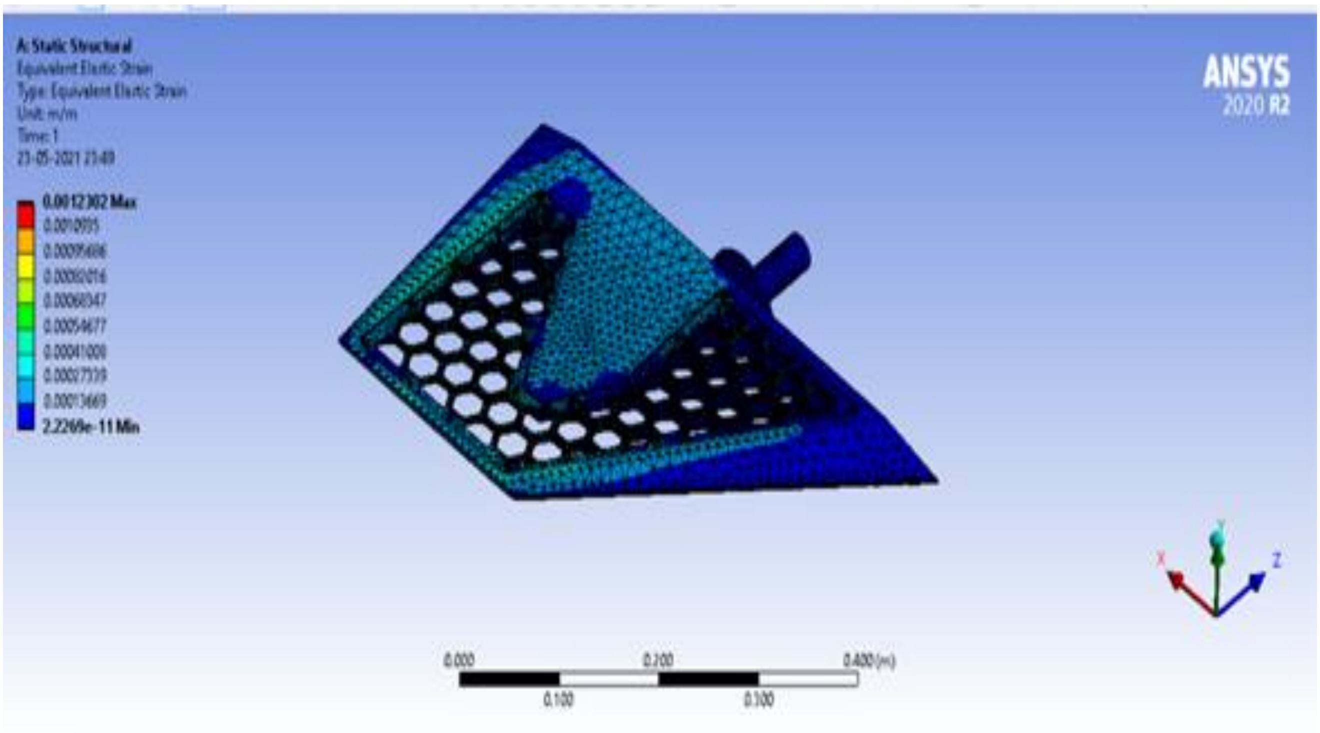
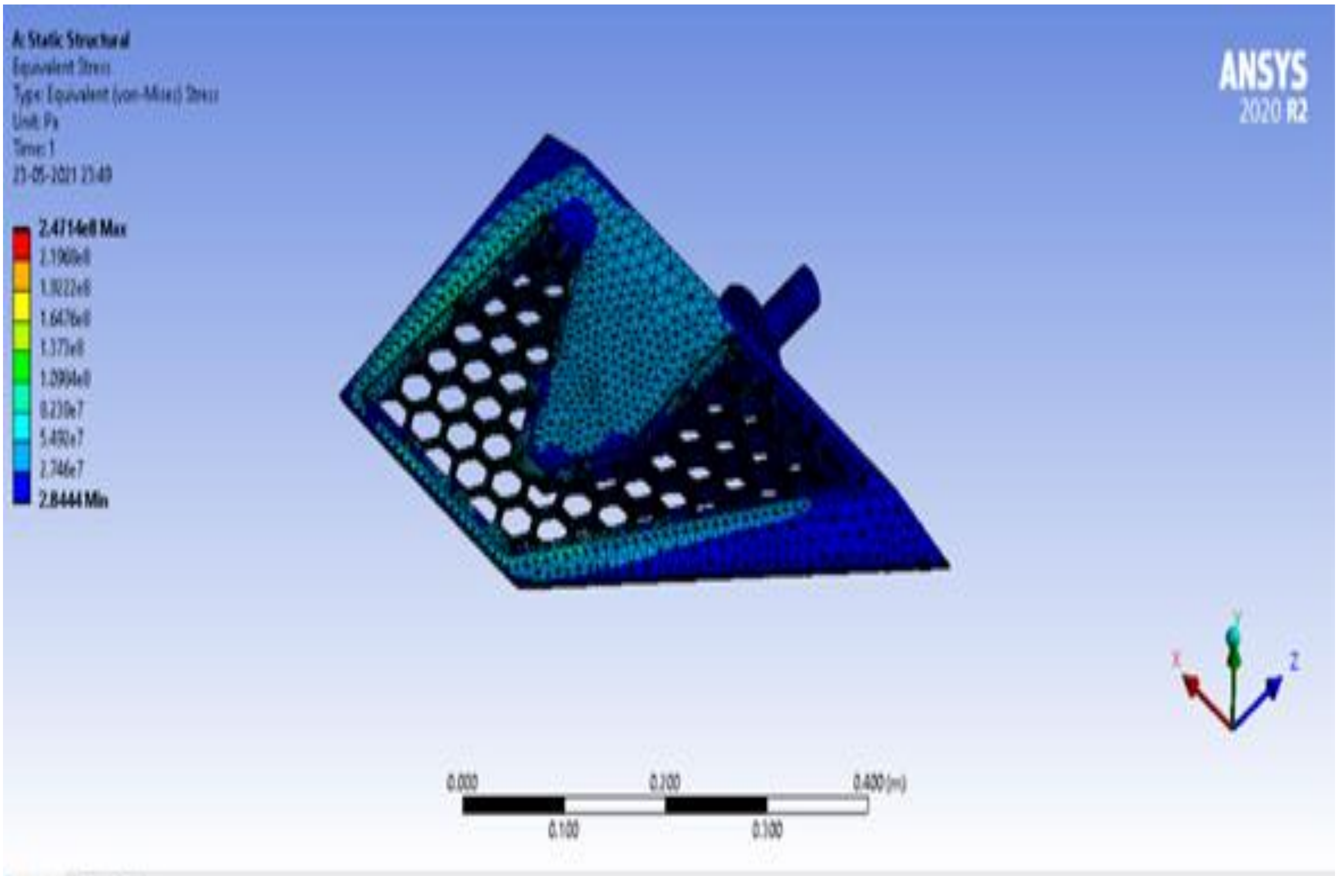


Figure 28: Total Elastic Stress (Analysis)



Volume reduction after Topology Optimization = 22 % (data obtained from Analysis)

Cost reduction per fin:
Actual cost of Alloy Steel per kg = Rs 50/-
Weight of fin (actual) = 20 kg
Total cost of fin = 20*50 = Rs 1000/-
Reduction in weight = (22/100) *20 = 4.4 kg reduction
The weight of the fin after optimization = (20-4.4) = 15.6 kgs
Cost of the fin after optimization = (15.6*50) = Rs 780/-

Reduction in cost after Optimization = (Total cost of fin-Cost of fin after Optimization)

= (1000-780) = **Rs 220/- for each fin.**
% Reduction in cost = (Reduction in cost*100)/Total cost of fin = (220*100)/1000 = **22%**

6. CONCLUSION (OVERALL):

The aerospace fin after Topology Optimization are as follows:

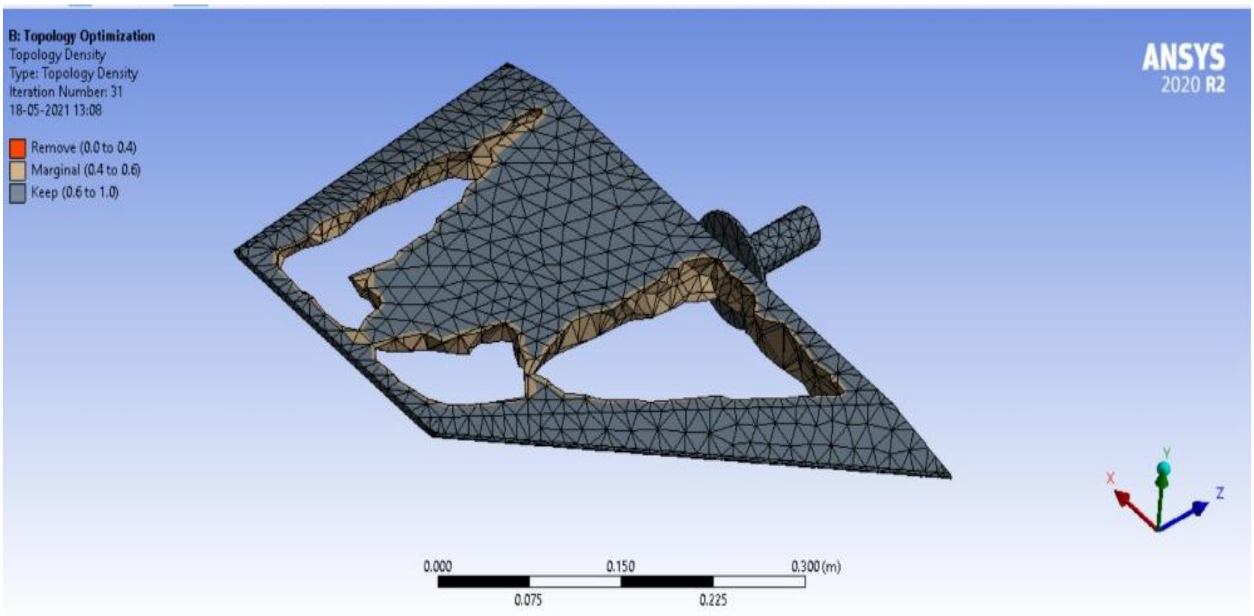


Figure 17: Topology Optimization (ANSYS)

OUTCOMES:
- **Cost optimisation**:
- As the extra weight is removed, the extra material is removed which is directly proportional to minimisation of cost.
- Ease of fabrication will lead to manufacturing cost reduction.

- Range of the system is increased as four such items are used per system resulted in considerable weight saving.
- Extra/unnecessary weight/mass is minimised (size optimisation).
- A shortened design process.
- No change in interface dimensions hence, same assembly procedure is valid with no change in documentation .
- Centre of gravity is brought closer to mounting location resulting in reduced control effort to steer the vehicle.
- Lesser current drawn by actuator due to reduced inertia resulting in lowering of battery power requirement.

Aerospace fin after Generative Design in Ansys:

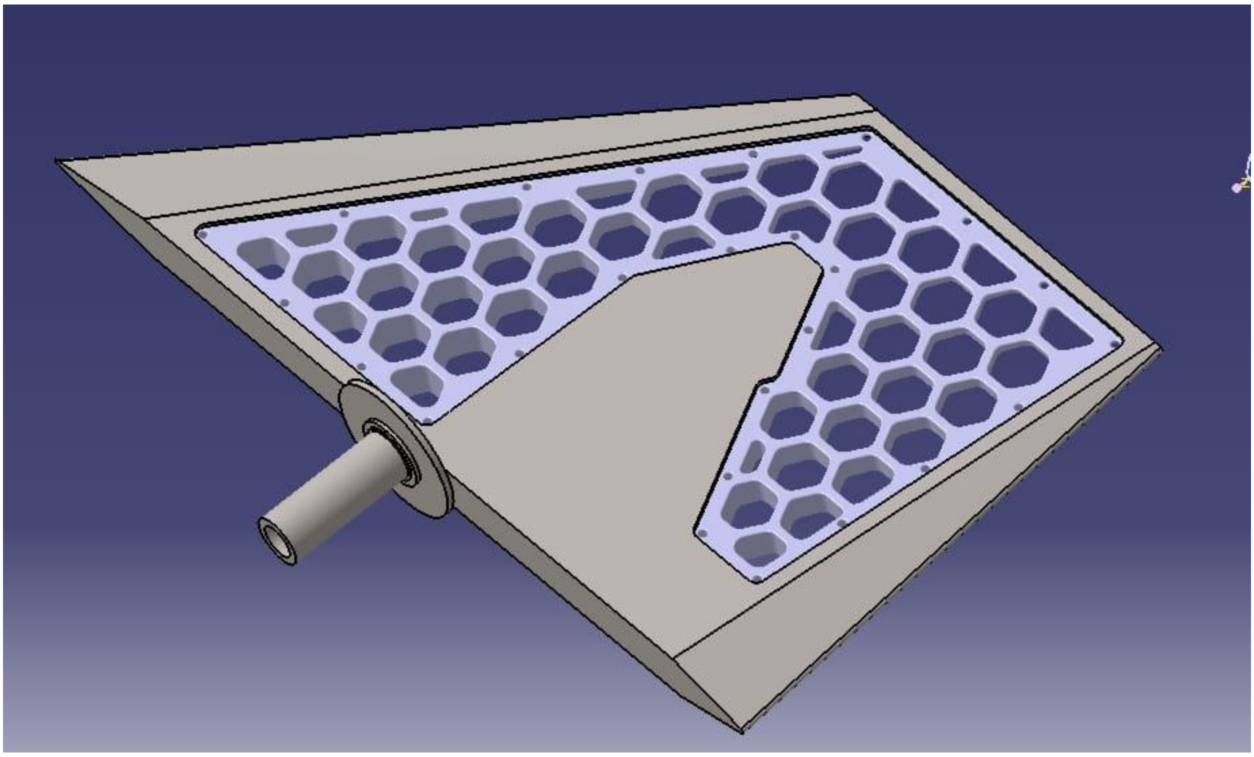


Figure 18: Fin with Generative designing in ANSYS

7. REFERENCES:

1) An existing Control surface, DRDO RCI.

2)Bendsoe, MP., Kikuchi N., Generating optimal topologies in structural design using a homogenization method, Computer Methods in Applied Mechanics and Engineering, vol. 71, no. 2, pp. 197-224, 1988.

3)Bendsøe, MP. and Sigmund O., Topology Optimization: Theory, Methods, and Applications, Springer-Verlag, Berlin Heidelberg New York, 2003.

4)Bendsoe, MP., Optimal shape design as a material distribution problem, Structural Optimization, vol. 1, pp. 193-202, 1989.

5)Cazacu, R. and Grama L., Overview of Structural Topology Optimization Methods for Plane and Solid Structures, Annals of the University of Oradea, Issue No.3, December 2014.

6)Pedersen, C.B.W. and Allinger, P., Industrial implementation and applications of topology optimization and future needs, IUTUAM Symposium on Topological Design Optimization of Structures, Machines, and Materials: Status and Perspectives (eds. M.P. Bendsøe, N. Olhoff and O. Sigmund), Springer, pp. 147-156, 2006.

7)Rozvany, G.I.N, Zhou M., and Birker T., Generalized shape optimization without homogenization, Structural optimization, vol. 4, no. 3-4, pp. 250-252, 1992.