BRNO UNIVERSITY OF TECHNOLOGY

VYSOKÉ UČENÍ TECHNICKÉ V BRNĚ

FACULTY OF MECHANICAL ENGINEERING

FAKULTA STROJNÍHO INŽENÝRSTVÍ

INSTITUTE OF AEROSPACE ENGINEERING

LETECKÝ ÚSTAV

TOPOLOGY OPTIMIZATION OF THE HINGE ON ELASTIC FOUNDATION

TOPOLOGICKÁ OPTIMALIZACE ZÁVĚSU NA PODDAJNÉM PODKLADU

MASTER'S THESIS

DIPLOMOVÁ PRÁCE

AUTHOR BEng. Vasili Beruashvili

AUTOR PRÁCE

SUPERVISOR Ing. František Löffelmann

VEDOUCÍ PRÁCE

BRNO 2020

Specification Master's Thesis

Department:	Institute of Aerospace Engineering
Student:	**BEng. Vasili Beruashvili**
Study programme:	Mechanical Engineering
Study branch:	Aircraft Design
Supervisor:	**Ing. František Löffelmann**
Academic year:	2019/20

Pursuant to Act no. 111/1998 concerning universities and the BUT study and examination rules, you have been assigned the following topic by the institute director Master's Thesis:

Topology Optimization of the Hinge on Elastic Foundation

Concise characteristic of the task:

Topology optimization, i.e. searching for material distribution within the design space, is used mainly in the initial design phase, usually with the aim to maximize stiffness or minimize mass by use of constraints (manufacturing, stress, displacements, ...). This approach, when the part is designed by mathematical algorithms for specific boundary conditions and loads gives high flexibility to the designer, but the result is sensitive to chosen parameters. Frequent application of the topology optimization is design of various hinges, where bolted joins are usually simplified as fixed, which might be imprecise especially if the foundation is too elastic. The diploma thesis should concern with design of the airliner galley hinge and influence of the elastic foundation consisting of sandwich panels of the galley.

Goals Master's Thesis:

1) Accomplishing topology optimization with regard to manufacturing technology (milling, 3D printing), allowable stress and buckling.
2) Finalizing geometric shape by shape optimization or size optimization of the chosen parameters especially to decrease local stress concentration.
3) Comparison of the new hinge with the original variant from the mass and stress point of view.
4) Assessing boundary condition influence (stiffness of the foundation) on the optimization results.

Recommended bibliography:

TOMLIN, Matthew a Jonathan MEYER. Topology Optimization of an Additive Layer Manufactured (ALM) Aerospace Part. The 7th Altair CAE Technology Conference 2011. Altair Engineering, 2011. Available at: https://www.additivemanufacturing.media/cdn/cms/uploadedFiles/Topology-Optimizatio--of-an-Additive-Layer-Manufactured-Aerospace-Part.pdf

MIRZENDEHDEL, Amir M a Krishnan SURESH. A hands-on introduction to topology optimization. Great Britain: [Amazon], 2017, 236 pages. ISBN 9781976480607.

MSC Nastran 2018.2: Design Sensitivity and Optimization, User's Guide [online]. Newport Beach, USA: MSC Software Corporation, 2018.

BENDSØE, Martin Philip a Ole SIGMUND. Topology optimization: theory, methods and applications. Second eddition, corrected printing. Berlin: Springer, 2004, xiv, 370 pages. ISBN 3-540-42992-1.

Deadline for submission Master's Thesis is given by the Schedule of the Academic year 2019/20

In Brno,

L. S.

doc. Ing. Jaroslav Juračka, Ph.D.
Director of the Institute

doc. Ing. Jaroslav Katolický, Ph.D.
FME dean

Abstract

Deal of this Master's thesis is to modify original part shape using topology optimization by M.S.C NASTRAN, M.S.C. PATRAN and FUSION 360 software results that will define best fitted and satisfying the operating conditions for defined load and constraint conditions. The part is mounted on an orthotropic plate (sandwich panel). The aim of this thesis is to see the effect of elastic foundation on optimization result. The part will be optimized using different design objectives and constraints. Elastic foundation theoretically will change stiffness and deformation, and this will give the ability to change stresses on the part. Original and modified shape part load capacity will be compared by M.S.C NASTRAN/PATRAN software. After shape optimization, the 3D model has to be prepared for the manufacturing process which will be the most cost-efficient.

Keywords:

Shape optimization, M.S.C NASTRAN, M.S.C. PATRAN, FUSION 360, Topology Optimization, Elastic foundation, Additive manufacturing, 3D printing, Multi-Axis milling.

BERUASHVILI, Vasili. *Topology Optimization of the Hinge on Elastic Foundation.* Brno, 2020 Master's Thesis. Vysoké učení technické v Brně, Fakulta strojního inženýrství, Institute of Aerospace Engineering. Supervisor František Löffelmann. Dostupné také z: https://www.vutbr.cz/studenti/zav-prace/detail/125360

I declare that the thesis entitled, “Topology optimization of the hinge on elastic foundation” is my own work and that all the sources that I have used or quoted have been listed and acknowledged at the end of the thesis.

BEng. Vasili Beruashvili

In Georgia, Tbilisi on

22.06.2020

Acknowledgment

I would like to thank PhD. Teimuraz Lomtadze from the Technology Institute of Georgia, for funding my whole master's study at the Brno University of Technology therefore giving me immeasurable lifetime experience, also to express my special thanks of gratitude to my supervisor Ing. Frantisek Löffelmann. I also want to thank all the employees of the Brno University of Technology especially Ing. Tomáš Katrňák for his assistance during 2 years study at BUT. Finally, my deep and sincere gratitude to my family for their continuous and unparalleled love, help and support.

BEng. Vasili Beruashvili

.

Table of Contents

1. INTRODUCTION

Topology optimization is a special version of design optimization that finds an optimal distribution of material, given package space, loads and boundary conditions. It makes a design variable out of each finite element that can vary from 0 (remove) to 1 (keep), and the algorithm strives to force real design variables to one of these limits. This process is similar to ESO (evolutionary structural optimization).

Evolutionary Structural Optimization (ESO) is a design method based on the simple concept of gradually removing inefficient material from a structure as it is being designed. Through this method, the resulting structure will evolve towards its optimum shape.

The solution to the Topology Optimization problem is ill-posed in the sense that the design tends to a configuration with an unbounded number of microscopic holes rather than a small number of macroscopic holes. This suggests that the design will not generally converge to an optimum as the mesh is refined. There are two alternative ways for generating a well-posed Topology Optimization problem. In a procedure called relaxation Ref. [1] checkerboard designs are accommodated by extending the design space to include materials with periodic, perforated microstructures and then using homogenization theory to compute effective material properties. Alternatively, in a procedure called restriction, the design space is restricted to exclude checkerboard designs by imposing perimeter constraint Ref. [2].

The most popular mathematical method for topology optimization is the Solid Isotropic Material with Penalization method (SIMP). Bendsoe and Kikuchi (1988) and Rozvany and Zhou (1992). initially proposed the SIMP method. The SIMP method predicts an optimal material distribution within a given design space, for given load cases, boundary conditions, manufacturing constraints, and performance requirements.

According to *Bendsoe (1989)*: "shape optimization in its most general setting should consist of a determination for every point in space whether there is material in that point or not." The traditional approach to topology optimization is the discretization of a domain into a grid of finite elements called isotropic solid microstructures. Each element is either filled with material for regions that require material or emptied of material for regions where you can remove material (representing voids). The density distribution of material within a design domain, ρ, is discrete, and each element is assigned a binary value:

- $\rho_{(e)} = 1$ where material is required (black)
- $\rho_{(e)} = 0$ where material is removed (white)

For example, the image shows an optimized material layout of a loaded beam. The solid elements with densities $\rho_{(e)} = 1$ is black, whereas the void elements with $\rho_{(e)} = 0$ are removed.

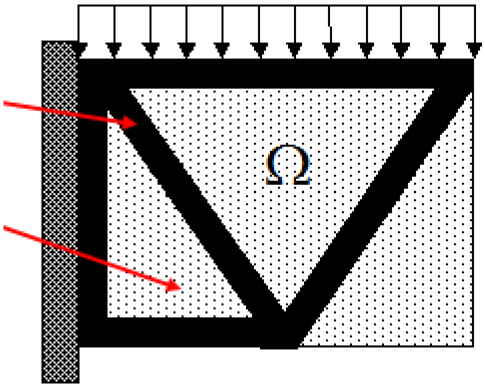

Figure 1. Optimized material layout of a loaded beam Ref. [3]

Since the material relative density can vary continuously, the material Young modulus at each element can also vary continuously. For each element e the relation between the material relative density factor ρ_e and the Young modulus of elasticity of the assigned isotropic material model E_0 is computed by the power law:

$$E(\rho_e) = E_0 * \rho_e^p$$

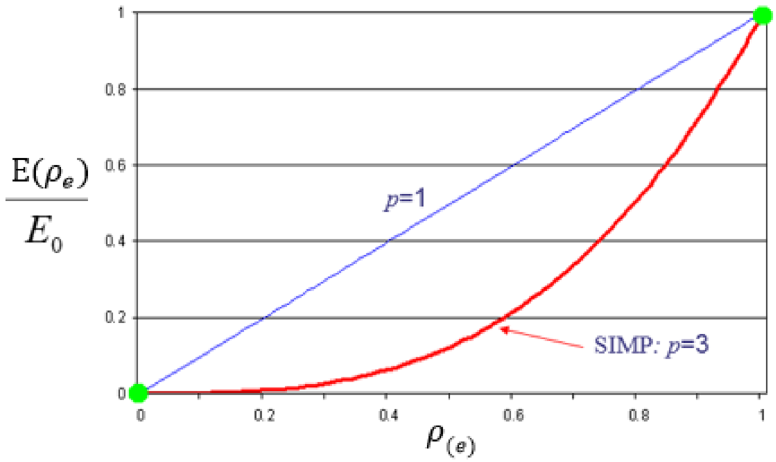

Figure 2. penalty factor (p) diagram Ref. [3]

The penalty factor p diminishes the contribution of elements with intermediate densities (gray elements) to the total stiffness. The penalty factor steers the optimization solution to elements that are either solid black (ρ_e = 1) or void white (ρ_e=ρ_{min}).

Structural topology optimization helps the designer in defining the type of shape, which is best fitted to satisfying the operating conditions for defined load and constraint conditions. It can be seen as a procedure of optimizing the arrangement of the available material in the design space and removing the material that isn't needed. Topology optimization is usually used to achieve an acceptable initial layout of the structure, which is then refined with a shape optimization tool. The topology optimization procedure proceeds step by step with a gradual cut-out of small portions of low-stressed material, which aren't loaded.

In topology optimization of Aerospace structures, parts and units, parametrization of shape is often performed by a color-scale gradient interpolation function. In this Master's thesis, will be analyzed and compared the various approaches to this concept in light of using different software and approaches to design and manufacture of complex shape structures. with using modern technologies in additive manufacturing (AM), for example, metal 3D printing

Topology optimization is one of the ways of structural optimization like Generative design, Fully Stressed Design, Geometry parameterization, and others.

In the last 30 years, the use of FEM based software for topology and shape optimization in the industry has been used widely and has shown to be an effective way of to partially or fully solve different types of problems. Aim of the idea behind is that aircraft designers and the stress engineer both are involved in the initial stage when the first design concept is generated. Topology optimization is used to generate an optimum concept.

In our case part is mounted on the galley's ceiling which is made from a sandwich panel with composite face sheets. Firstly, hinge and brackets will be analyzed on a stiff foundation and obtained stress results will be compared to the case where the hinge is mounted on an elastic foundation.

Types of manufacturing that will be used will be multi-axis Machining and 3D printing with different categories: powder bed fusion, binder jetting, direct energy deposition, and material extrusion. with a few exceptions that types can cover all types of metals and other materials

Additive manufacturing (AM) is the industrial production name for 3D printing, a computer-controlled process that creates three-dimensional objects by depositing materials, usually in layers

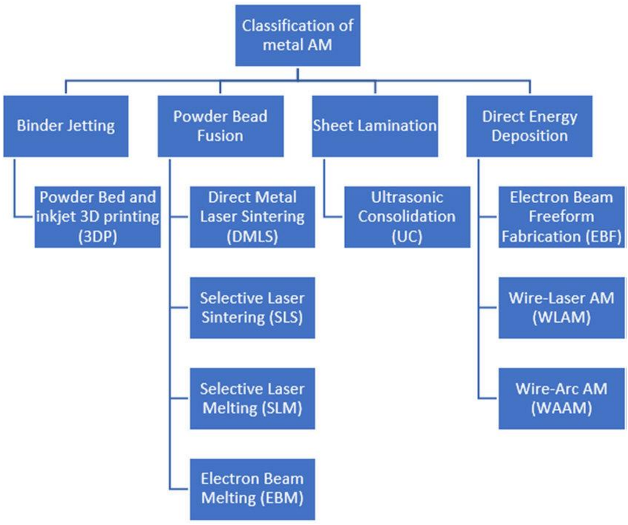

Figure 3. classification of metal AM Ref. [4]

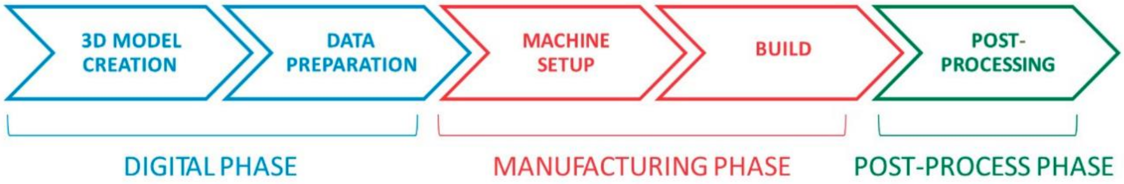

Figure 4. AM manufacturing process setup Ref. [5]

Metal Powder Bed Fusion (PBF)

This category includes DMLS (direct metal laser sintering), SLM (selective laser melting), and EBM (electron beam melting) machines.

Metal parts produced using PBF melting contains residual stresses. Residual stress can be can be decreased by heat treatment (annealing)

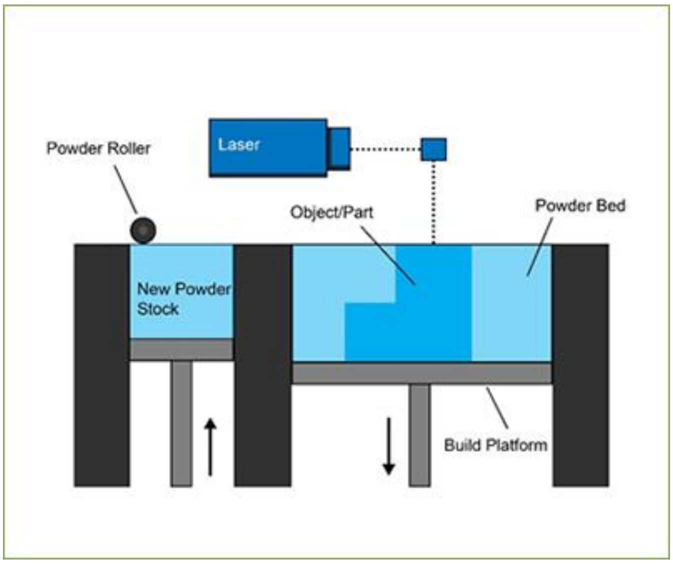

Figure 5. Metal Powder Bed Fusion Ref. [6]

The history and categorization of PBF metal 3D printers get a bit messy and it mostly relates to the difference between sintering and melting.

Binder jetting, like laser sintering, can handle more than metal materials. Sand, ceramic, and full-color objects are also possible with the technology. Because metal binder jetting machines operate at room temperature, warping does not occur and supports are not necessary. “As such, binder jetting machines can be much larger than powder bed fusion machines and objects can be stacked to use the entirety of the build chamber, so it's a popular choice for small batch production runs and on-demand replacement parts”. This sentence cites one Ref. [7]

The final shape of Hinge and brackets after will be also optimized for classical types of multi-axis milling (X, Y, Z; X, Y, Z, A, B)

Multi-Axis milling can be classified by the amount of CNC axis's 3-4-5 because topology optimization output is complex shape part 5 Axis milling will be the most sufficient technology to use.

5 axis milling machine can move on 3 axes and rotate on two, that to the axis are named A and be so 5 axis milling machine contains X Y Z an A B axis's, for visualization A and B we can describe its orientation. As it turns, in the same way, plane banks. Its role is described by the fourth axis, A: the rotational axis around X for axis B rotation axis will be Y

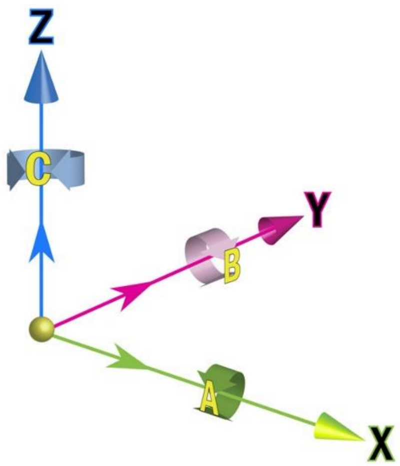

Figure 6. Axis's and their orientations in the coordinate system

In the simplest terms, 5-axis machining involves using a CNC to move a part or cutting tool along five different axes simultaneously. This enables the machining of very complex parts, which is why 5-axis is especially popular for aerospace applications.

However, several factors have contributed to the wider adoption of 5-axis machining. These include:

- A push toward single-setup machining (sometimes referred to as “Done-in-One”) to reduce lead time and increase efficiency
- The ability to avoid collision with the tool holder by tilting the cutting tool or the table, which also allows better access to part geometry
- Improved tool life and cycle time as a result of tilting the tool/table to maintain optimum cutting position and constant chip load

.

This Master's thesis task is to modify the shape of the galley mount.t is an assembly of one hinge and two brackets which are mounted on galleys celling.

steps of part optimization are divided into:

- Stress analysis of hinge and bracket assembly with appropriate boundary conditions and loading results of stresses will be compared to the results of the optimized part.
- Stress analysis of the hinge and brackets assembly.
- Stress analysis of the hinge and brackets assembly on elastic foundation.
- Topology optimization of the hinge on elastic foundation by classical method, minimizing compliance, with mass constraints.
- Topology optimization of the hinge on the elastic foundation for buckling
- Topology optimization of the modified hinge (increased design domain) on elastic foundation
- Topology optimization of the modified hinge without elastic foundation
- Topology optimization (generative design) in Fusion360

When these steps will be don. The final shape of the modified part will be redesigned for3D printing and Multi-Axis milling.

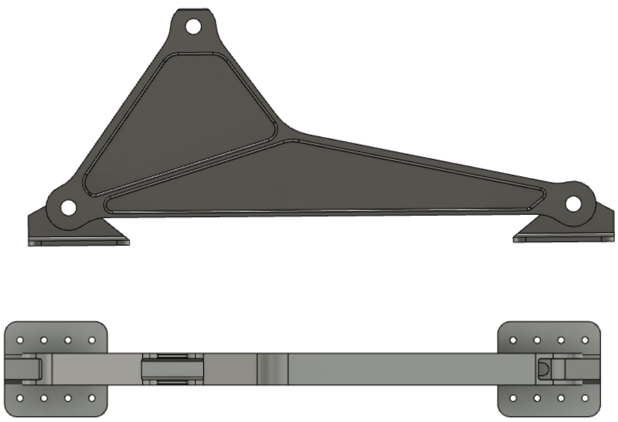

Figure 7. Hinge and brackets front and top view

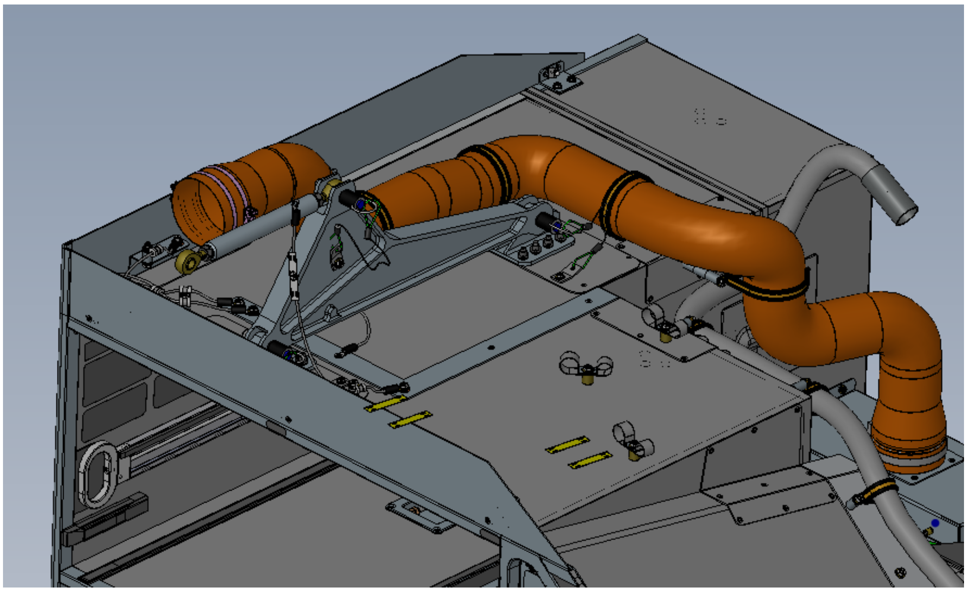

Figure 8. Design Space

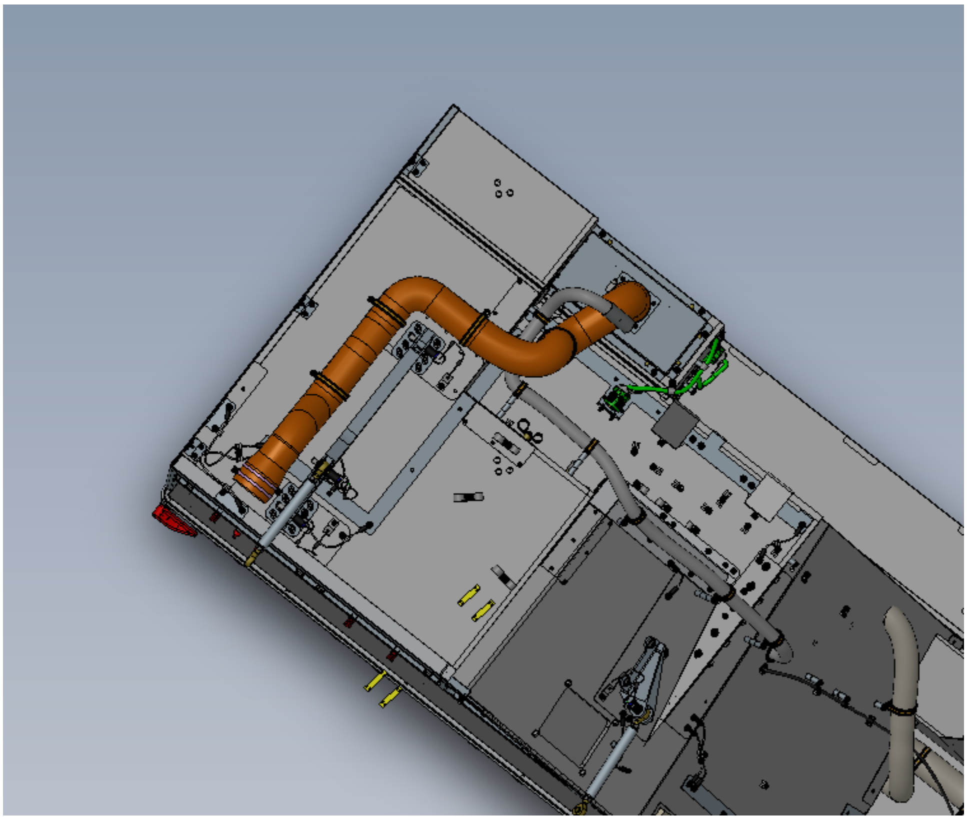

Figure 9. Design space Top

(publication of figures № 8-9 was allowed by Safran Cabin CZ)

2. Stress Analysis.

2.1 Difference between Stress Analysis and Design Optimization.

Although design optimization and analysis can be viewed as corresponding, there are some important. conceptual differences between the two which must be clear in order to make effective use of both approaches.

According to MSC Nastran 2012 Design Sensitivity and Optimization User's Guide Ref. [8]

When we perform an "analysis," we create a mathematical idealization of some physical system in order to obtain estimates of certain response quantities. The class of responses that we are interested in defines the applicable analysis discipline to be used, while the accuracy of these responses is dependent on the quality of the analysis model and our general knowledge of the true system. Our choice of finite element types, representation of boundary conditions, loads, and definition of the finite element mesh all play critical roles in determining how well our model is able to predict the responses of the physical structure. The goal is to obtain an accurate prediction of the responses which can be expected from the real structure. For example, consider the plate subjected to uniform tensile loads in Fig.10The corresponding analysis model in Fig. 10 is a discretized finite element representation of idealized geometry, loads, and boundary conditions.

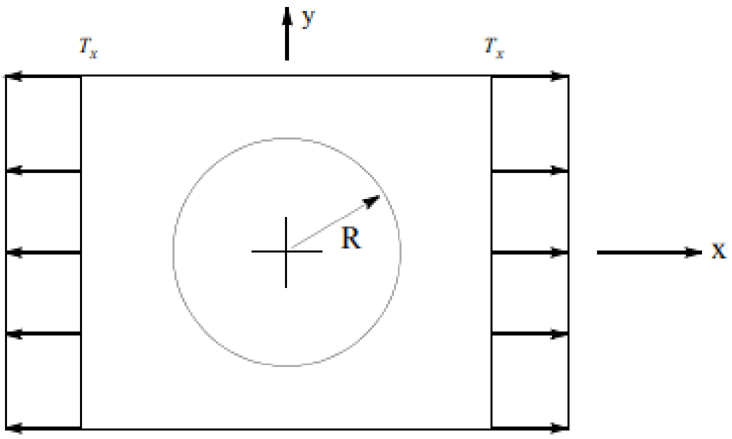

Figure 10. flat plate with hole Ref. [8]

Analysis model

Design model

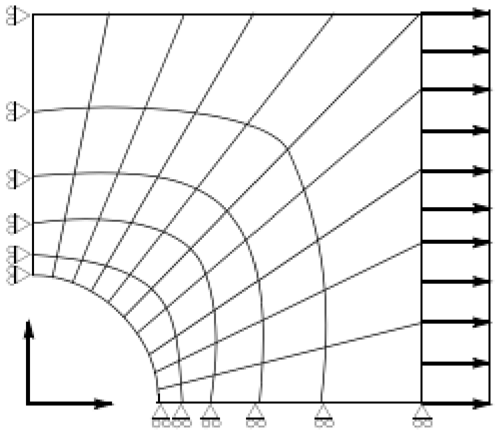

Figure 11. Analysis model Ref. [8]

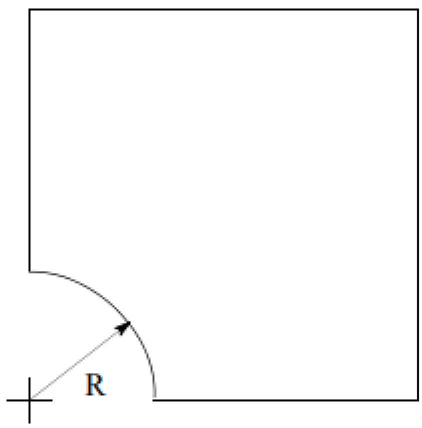

Figure 12. Design model Ref. [8]

Finite element discretization of:

• Structure (Mesh).

• Loads.

• Boundary Conditions

(1/4 Plate Representation).

Find R such that:

• Weight is minimized.

• Stresses do not exceed allowable.

(R is the design variable, weight is the design

objective and stresses are the design constraints.)

In contrast, a design model is an idealized statement of changes which might be made to the structure to improve its performance or response. In order to accomplish this, we need to define what we mean by an

improved design. It may be the minimum weight or maximum stiffness. For design optimization, we will use Topology optimization Ref. [8]

2.2 Hinge and brackets stress analysis

For hinge stress analysis part M.S.C. Patran and M.S.C. Nastran Softwares will be used.

STEP model of hinge and brackets is imported to Patran's work bench.

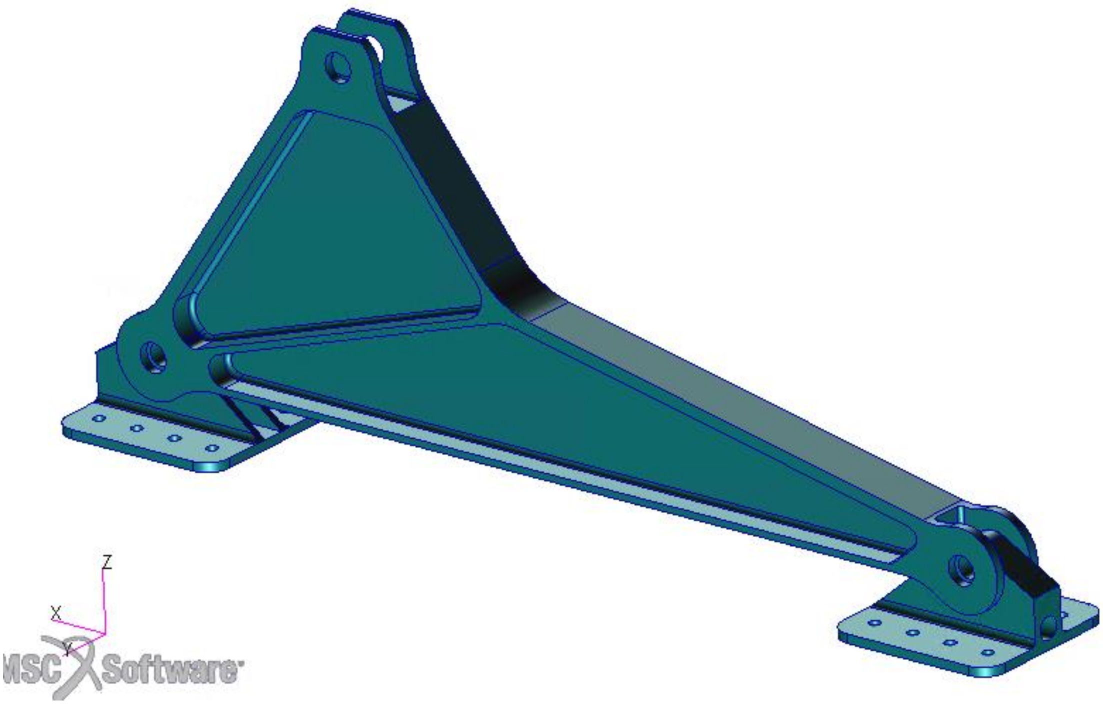

Figure 13.Hinge and brackets 3D model opened in Patran workebnch

After importing 3D model its necessary to determine places where part will constrain and where the load will be applied.

In our case, the external ultimate load $F_x = -15849[N]$ is applied on hinges top and it's constrained by lower pin holes.

For constrains and load, Multi-Point Constraints (MPC) elements RBE3 and RBE2 will be used.

- For distributing external force RBE3
- For constrains RBE2

RBE2 and RBE3 elements are not Exactly "RIGID" elements. They are both variations of an MPC (multi-point constraint), but there are some important differences in terms of the use of RBE2 Vs RBE3 elements.

Comparing RBE2 Vs RBE3 elements, RBE3 is not a "RIGID" element. The motion at the dependent node is a weighted average of the motion at the independent'nodes. This is why it is also known as an interpolation R element. RBE3 elements do not add artificial stiffness to the structure like the RBE2 element does.

In effect, an RBE3 element is nothing but a free body diagram to balance the loads and moments at the 'independent' nodes. Note how the applied moment is simply moved to the CG of the 'independent' nodes.

Because of that effect, the RBE3 element will be used to apply load on the structure and RBE2 element to constrain it.

Material properties are defined in Table

Constitutive model	Linear elastic	unit
Elastic modulus	70 000	[MPa]
Poissons ratio	0.3	[-]

Table 1. Material Properties setup in MSC Patran

These properties are applied to Solid.

After this step location of Nodes for constraining and loads are defined.

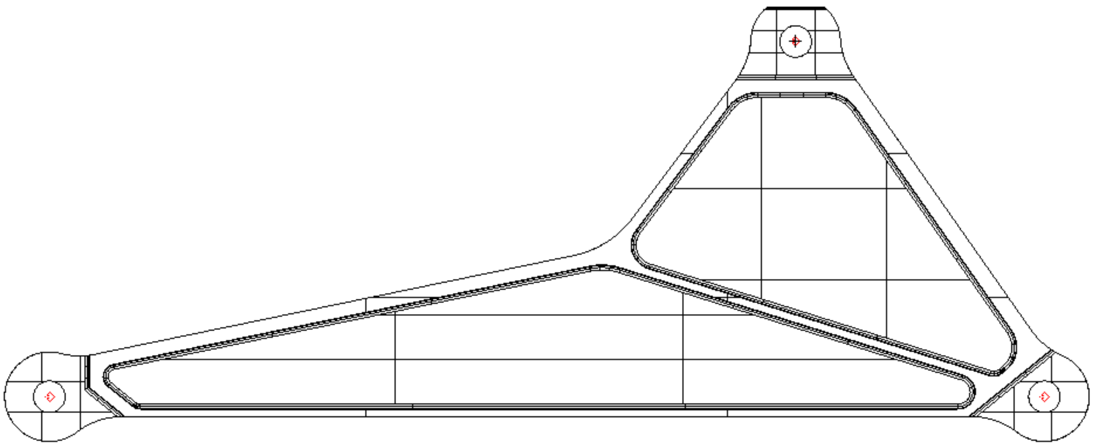

Figure 14. Red squares represent location of the nodes to create MPC elements

According to Multiphysics Cyclopedia Ref. [9] The accuracy that can be obtained from any FEA model is directly related to the finite element mesh that is used. The finite element mesh is used to subdivide the CAD model into smaller domains called Elements which a set of equations are solved. These equations approximately represent the governing equation of interest via a set of polynomial functions defined over each element. As these elements are made smaller and smaller, as the mesh is refined, the computed solution will approach the true solution.

This process of mesh refinement is a key step in validating any finite element model and gaining confidence in the software, the model, and the results

In our case mesh will have the following properties

Element Shape	Tet
Topology	Tet10
Global edge length	8
Elemet	211077

Table 2. Mesh creation inputs

Because the accuracy that can be obtained from any FEA model is directly related to the finite element mesh distribution that is used. Local mesh control on sharp edges and radiuses was done by command mesh seed.

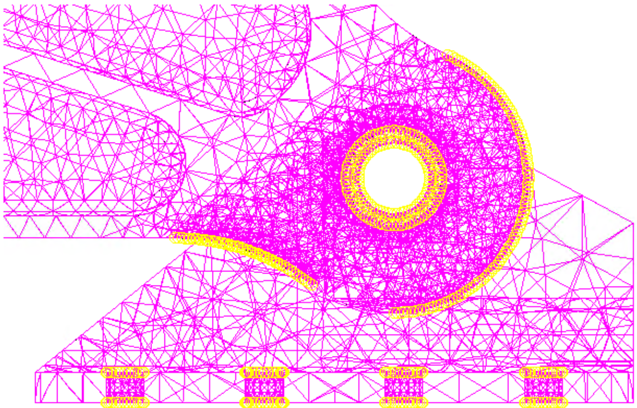

Figure 15. mesh adaptation

When elements and nodes are created, we can create MPC elements for applying load and constraints. For Loading RBE3 will be set up. We have to select one Depended node in our case center node which we created before and Independent nodes that are closest to the hole edge.

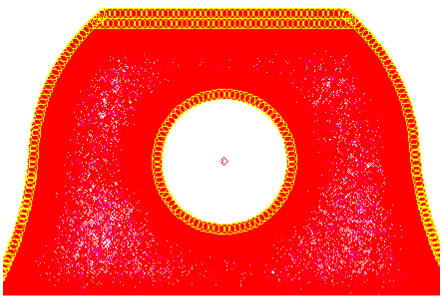

Figure 16. close look to lug where RBE3 element will be created

Dependetn terms		
nodes(1)	DOFs (Max=6)	
3	UX,UY,UZ,RX,RY,RZ	
independent terms		
coeficient	nodes(no max)	DOFs (Max=6)
1.00	64853:64878 65…	UX,UY,UZ

Table 3. Terms for MPC

RBE3 element's dependent node (3) has all 6 digress of freedom (DOF) which will transfer load, to structure's independent nodes have only three.

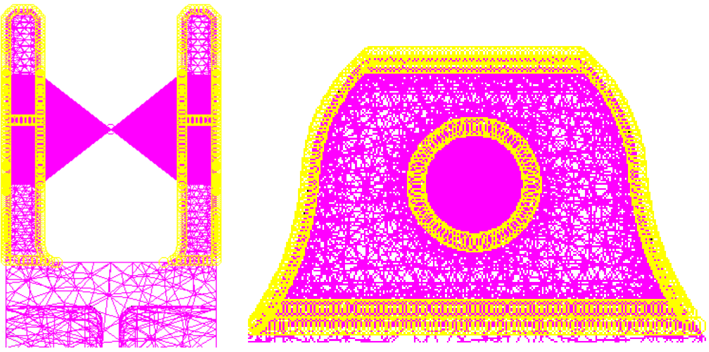

Figure 17. RBE3 element to applying load

Force is applied to MPC Element $F_x = -15849[N]$

MPC elements RBE2 are used to create connection between pin holes and brackets. because originally part is constrained by pin and pin connection is a connection with one degree of freedom. RBE2 element will also have one degree of freedom to Ry direction.

Dependetn terms		
nodes(1)	DOFs (Max=6)	
node 1198:1558 159	UX,UY,UZ,RX,RZ	
independent terms		
	nodes(no max)	DOFs (Max=6)
	center node	

Table 4. Input data for pin connection RBE2.

To constrain the whole assembly of hinge and brackets also RBE2 elements are used

Bolt connection is a connection with zero degrees of freedom.

Dependetn terms		
nodes(1)	DOFs (Max=6)	
node 5869:6985 78(	UX,UY,UZ,RX,RZ	
independent terms		
	nodes(no max)	DOFs (Max=6)
	center node	

Table 5. Input for bolt connection RBE2

These inputs full fill data requirements for M.S.C. NASTRAN solver. Important results are maximum stress and its location on the part

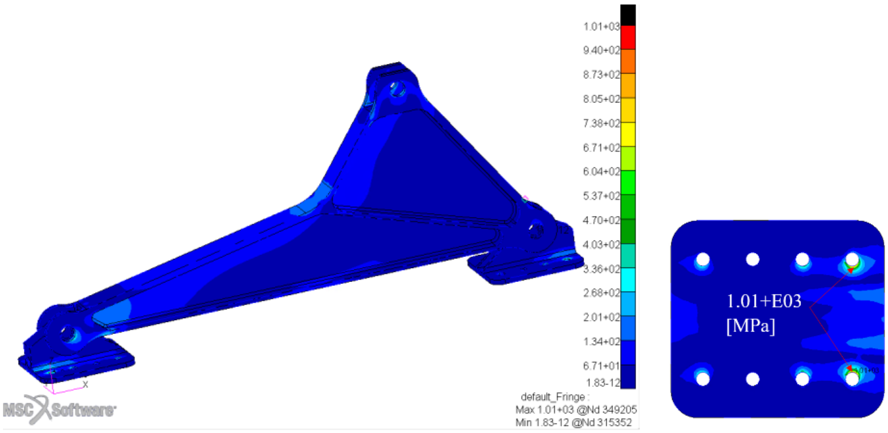

Figure 18. Von Mises stress on the hinge and brackets assembly

Maximum stress occur on rear brackets due to stress concentration on edge of bolt hole. From results we can see that maximum stress on this assembly, under external loading is 1010 [MPa]. which is not realistic, because for bolt connections RBE2 are used. RBE2 elements are rigid elements, their stiffness is unlimited therefore they cause artificial stress peaks. This stress can be neglected. When stress peacks are neglected actual maximum stress on the part is below green color on legend 403 [MPa] maximum stress location, in this case is front lug's connection to part . This part will be modified at the final design.

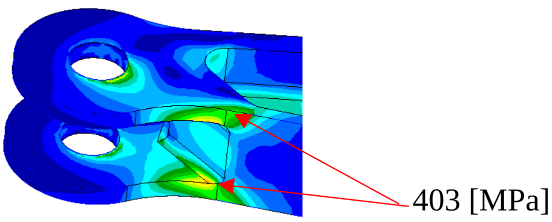

Figure 19. Max stress location on the hinge

Maximum stress locations are important for final design. At redesigned hinge, this place will be reinforced.

3. Elastic Foundation

Product assembly is mounted on Composite sandwich panel with prescribed mechanical properties

Original sandwich panel propertis		
t_f	0.5	[mm]
t_c	24.4	[mm]
E_f	23697	[MPa]
G_{f12}	3410	[MPa]
G_{c13}	40	[MPa]
G_{c23}	20	[MPa]
E_{c3}	140	[MPa]

Table 6. Original sandwich panel and Equivalent homogeneous plate properties

3.1 Hinge and brackets stress analysis on elastic foundation

3.1.2 3D model setup

Process of setting up a 3D model in M.S.C. Patran workbench is similar to only hinge and brackets setup with one difference, this analysis will be done on elastic foundation, sandwich panel, to set up this panel in M.S.C. Patran in CAD software D.S. CATIA on the bottom of brackets surface was created with dimension 650 X 550 [mm]

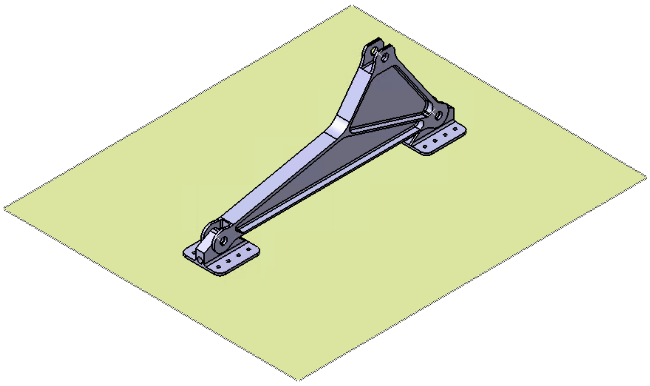

Figure 20. Hinge and brackets on surface D.S. CATIA workbench

3.1.3 material setup

Isotropic material for hinge and brackets is the same

Constitutive model	Linear elastic	unit
Elastic modulus	70 000	[MPa]
Poissons ratio	0.3	[-]

Table 7. Isotropic material properties

For sandwich panel, material setup original sandwich panel material will be used, because M.S.C. PATRAN allow to use composite elements in FEM, and there is no need to calculate equivavelnt sandwich panel.

To set up a sandwich panel in M.S.C. Patran, facsheet and core material parameters have to be defined

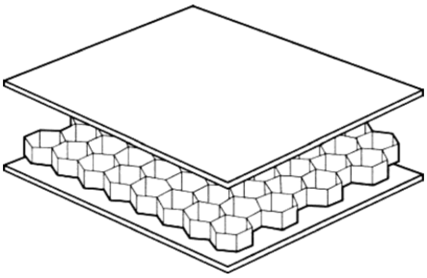

Figure 21. Sandwich Panel construction core between to face sheets

Property name	value	Unit
Elastic Modulus 11	140	[MPa]
Elastic Modulus 22	140	[MPa]
Poissons Ratio 12	0.4	[-]
Shear modulus 12	0.32	[MPa]
Shear modulus 23	20	[MPa]
Shear modulus 13	40	[MPa]

Table 8. 2D Orthotropic Core material properties

Property name	value	Unit
Elastic Modulus 11	23697	[MPa]
Elastic Modulus 22	23697	[MPa]
Poissons Ratio 12	0.15	[-]
Shear modulus 12	3410	[MPa]

Table 9. 2D Orthotropic Face sheet material properties

Upper and lower face sheets are made from the identical material.
When material properties are set up, the next step is to create composite material consisting of core and two face sheets in correct lamination order, thickness and orientation.

Material name	Thicknes	Unit	Orientation
Face sheet	0.5	[mm]	0
Core	24.4	[mm]	0
Face sheet	0.5	[mm]	0

Table 10. Sandwich panel material structure

When sandwich panel material structure is defined, property of surface which was created in D.S. Catia is created

Property name	value	value Type
Material orientation	Sandwich panel	Mat. Prop. Name
Material name	<1 0 0>	vector

Table 11. Surface properties.

3.1.4 discretization of 3D model and applying boundary conditions.

An often-used choice is to discretize the solution domain with tetrahedrons. On the one side a tetrahedron is a relatively simple element, especially regarding meshing aspects, on the other side, it is an efficient element to discretize structures with non-planar surfaces or complex geometries. This element is limited by four triangles and has four vertexes which are in any case grid nodes in the mesh.
For hinge and brackets the same approach will be used as it was done in previous FEA. Foundation Surface will be divided to QUAD elements with QUAD4 topology, with a global edge length of 2mm, such fine mesh was selected to have a node near bracket bolt hole centers for connection.

- Connection of hinge to the bracket is done same, like in previous approach by RBE2 elements with one degree of freedom Ry.

- For fixing bracket on foundation RBE2 elements with zero DOFs are used. Dependent nodes are nodes around the bolt hole surface and nodes of the sandwich panel under the bolt hole. Independent node is node in bolt surface center

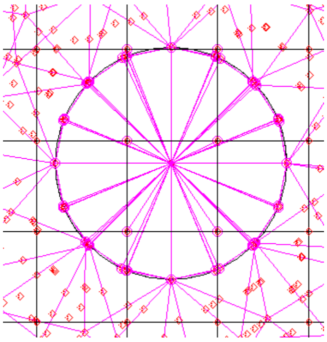

Figure 22. Example of RBE2 element for connection bracket and sandwich panel surface.

- To distribute external force $F_x = -15849[N]$, the RBE3 element will be used.
- For displacement constraints, sandwich panel surface edges are used.

Translations <T1 T2 T3>
<0., 0., 0.,>
Rotation <R1 R2 R3>
<0., 0., 0.,>

Table 12. Displacement properties for surface edge nodes

These inputs full fill data requirements for M.S.C. NASTRAN solver

3.1.5 Stress analysis post-processing

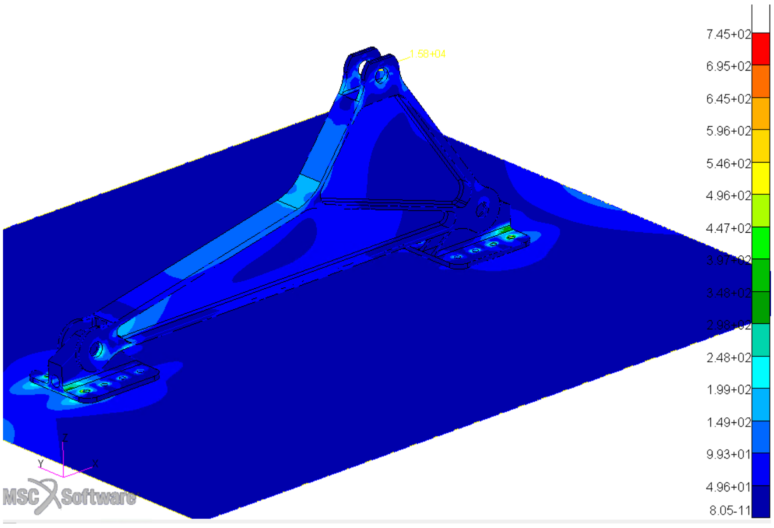

Figure 23. Von Mises stress on structure

With the fact that artificial stress peaks are caused by RBE2 elements. Maximum stress are lower than without elastic foundation with the same material and mesh setup for solid isotropic bodies (hinge and brackets). Maximum stress occur on rear bracket last two bolts hole edges but they are 27% lower than in analysis where structure isn't moundted on elastic foundation.

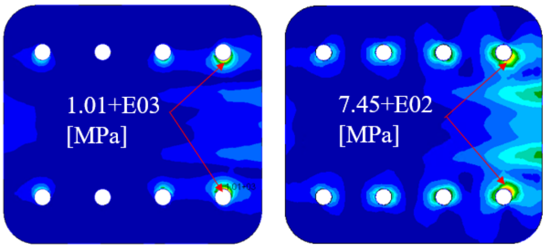

Figure 24. Bracket stress comparison without and with elastic foundation

When artificial stress peaks will be neglected stress on whole structure doesn't exceed the value of 336 [MPa] critical part is still same. Front lug's lower part.

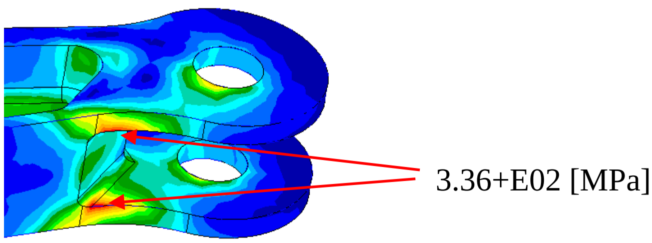

Figure 25. Max stress region on hinge

In topology optimization design variable stress constrains can be used. At this FEA stress is lower to compared without elastic foundation , in topology optimization, it will be the different final result.

4. Topology optimization on Elastic foundation

Topology optimization is one of the structural optimization techniques that optimizes the distribution of material within a specified design space for a given loading and boundary conditions while fulfilling the performance requirements of the product. Most of the topology optimization techniques are carried out by collective use of Computer Aided Design concept, Finite Element Analysis concept and different optimization algorithms in consideration of different manufacturing techniques

The task of this master thesis is to run the optimization process of structure which is mounted on the elastic foundation

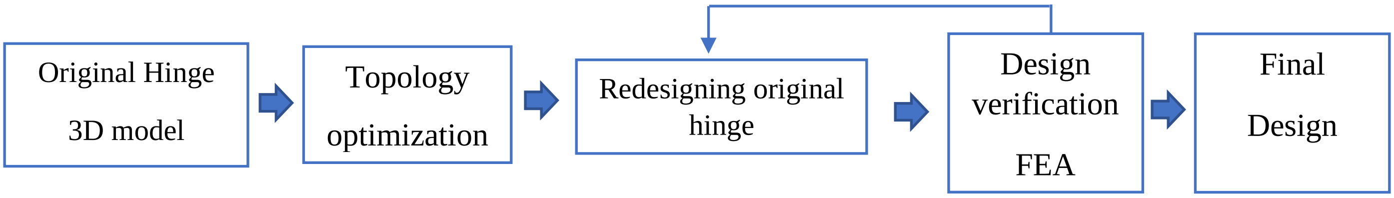

Figure 26. Topology optimized design process

4.1 Topology optimization with mass constraints

The topology optimization approach is considered among the most interesting fields of structural optimization. It is considered as an important field and a promising area that meets a great interest from mechanical designers and manufacturers. It is a relatively new but rapidly expanding research field. It also has important practical applications in the automotive and aerospace industries.

Topology optimization strives to achieve the optimal distribution of material within the finite volume design domain; which maximizes a certain mechanical performance under specified constraints. Its algorithms selectively remove and relocate the elements to achieve optimum performance. It can provide a good configuration concept for the structure as minimum compliance or maximum stiffness design. Ref. [10]

For this optimization M.S.C PATRAN and M.S.C NASTRAN will be used. Setup of the model for topology optimization is resembling M.S.C PATRAN FE stress analysis with exceptions of objectives and constraints. Design domain is still the same hinge and two brackets are mounted on sandwich panel

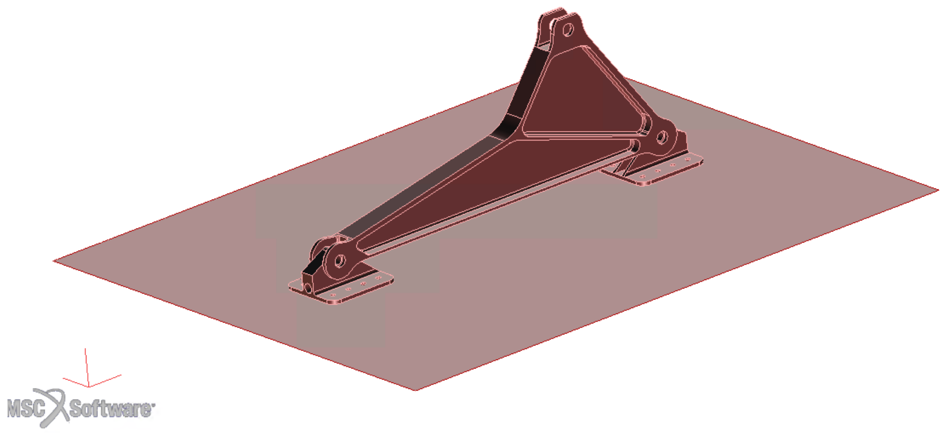

Figure 27. structure in M.S.C. PATRAN workbench

Setup of

- Materials
- Properties
- Loads
- Displacements
- Mesh
- Connections of structure members

for topology optimization at M.S.C. PATRAN is equal to FE stress analysis.

Difference is in objectives and constrains.

- Design objective for this optimisation will be Minimize complianse.
- Design constraint mass fraction 0.6 and symmetry to ZX plain

Mass fraction 0.6 represents Mass Fraction of topology designed elements (FRMASS).

For design domain hinge and brackets are selected.

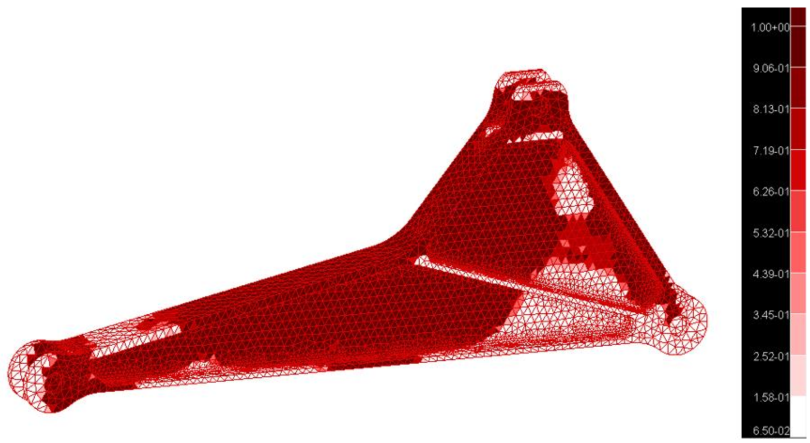

Figure 28. Design cycle 10

10 design cycles were calculated.

Legend bar in Fig. 28 represents elements density distribution where 1 equals original material density and 0 to void material which can be removed.

For post_processing of the results, Paraview software will be used. For generating specific extension file for ParaView software. custom python script, provided by supervisor was used.

On threshold 0.5 are shown elements in the range of 50-100% density of original material

Figure 29. Design model opened in ParaView software

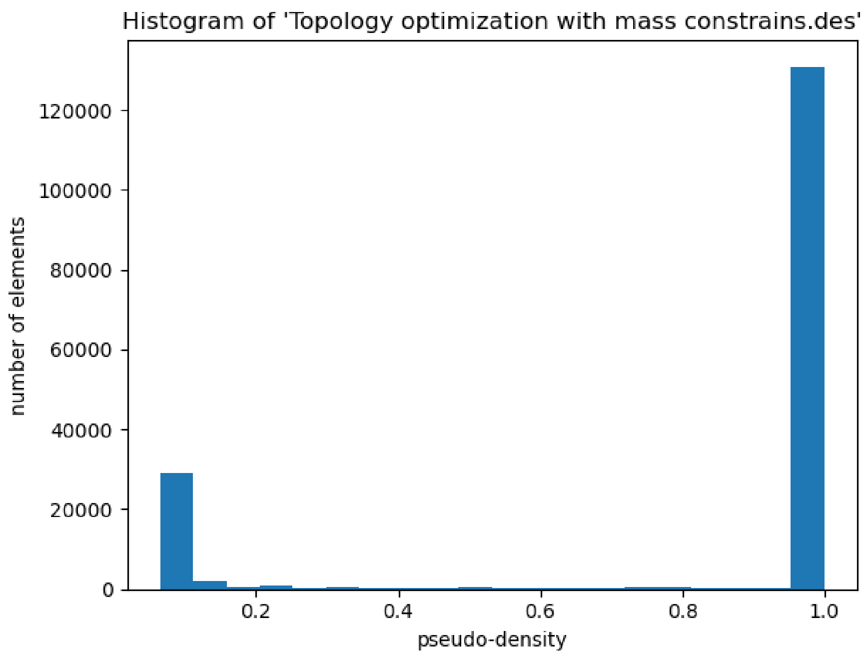

Figure 30. Histogram of optimization with mass constraints

This histogram represents data of elements with intermadiate densities. For finial design it elements with pseudo density more than 0.5 will be used.

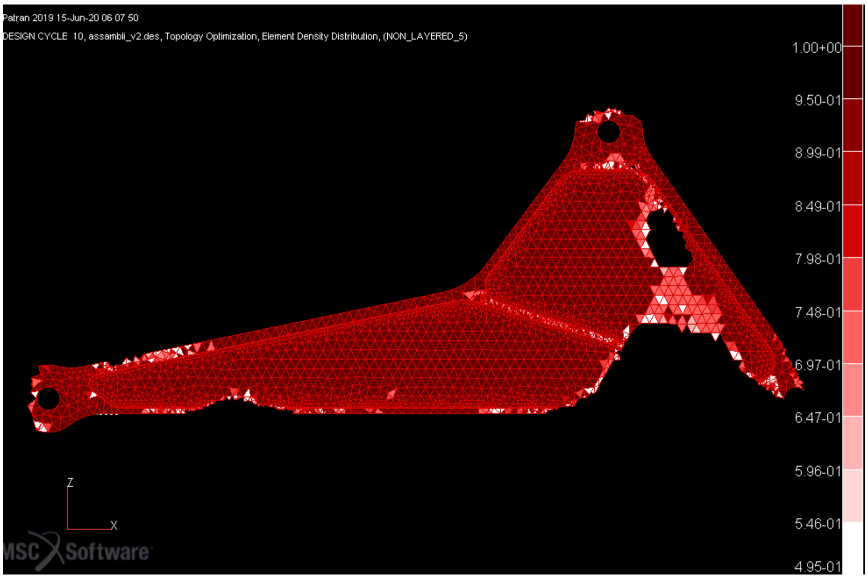

Figure 31. threshold 0.5

On threshold 0.5 are shown elements in the range of 50-100% density of original material.

4.2 Topology optimization with buckling constraints

Following topology optimization will be performed with buckling constraints.

Stability and buckling have attracted considerable attention since early times of structural optimization, due to their importance in the design of structural elements. Moreover, the optimal design according to weight or compliance minimization may naturally lead to structural configurations showing poor stability Ref. [11]

For this optimization M.S.C PATRAN and M.S.C NASTRAN will be used. Setup is alike setup for topology optimization with miminmization compline with mass constraint.

With the addition to design objective

- Design objective, compliance minimization
- Design constraint mass fraction 0.6, buckling factor set to 1 and symetry to ZX plain

For displaying results ParaVeiw software will be used

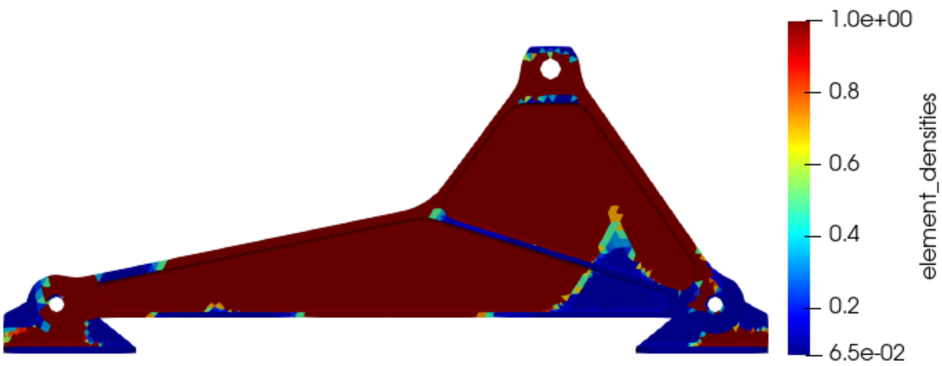

Figure 32. Design cycle 10

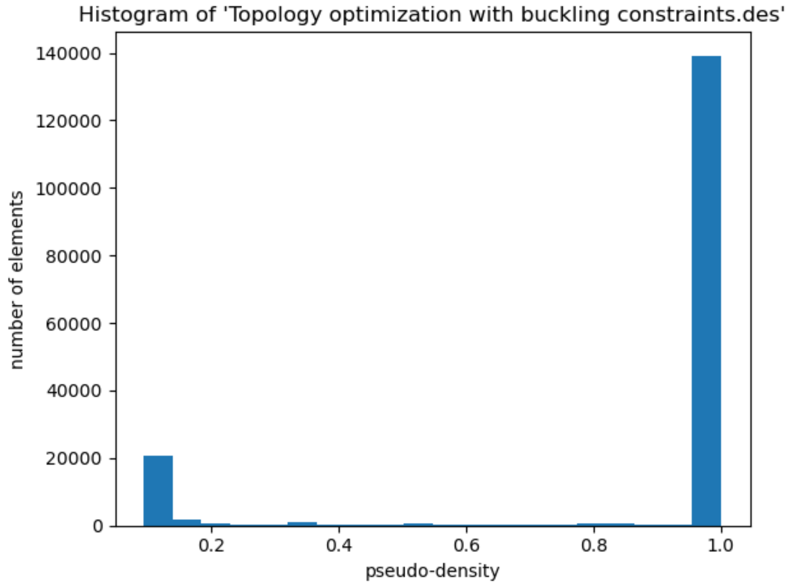

Figure 33. Histogram of optimization with buckling constraints

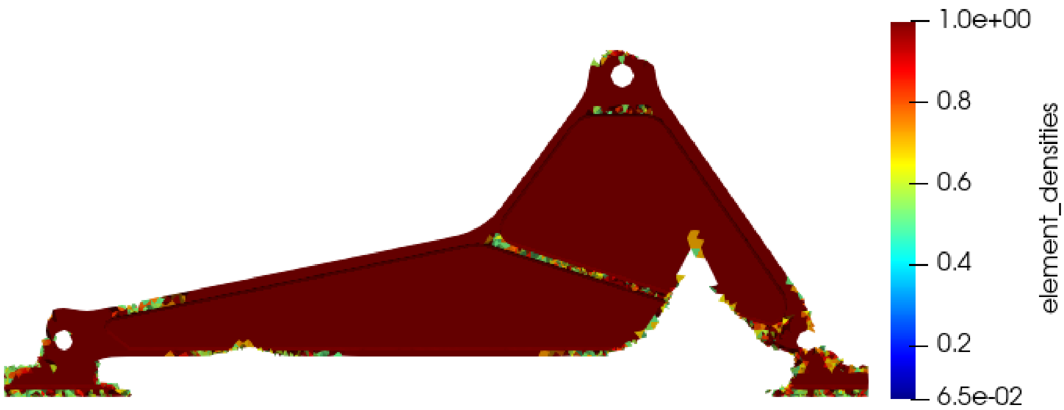

Figure 34. threshold 0.5

For modification of the original part shape of the threshold 0.5 will be used.

In comparison to pervious optimization (Topology optimization with mass constraints) after optimization volume of the lower stiffener is bigger, and density of elements near the hinge where external load is applied is maximum.

For the next optimization, it is necessary to modify design domain (hinge)

4.3 Topology optimization of the modified hinge on elastic foundation with mass and stress constrains

Pervious optimization results show the optimization of an original hinge, in this optimization Domain, will be bigger, element size smaller and design objective and constraints with addition.

- Design objective, Minimize Compliance
- Design constraint mass fraction 0.25
- Design constraint Von Mises Stress 300 [MPa]
- Casting constraint to Y axis with two Dies (equvalent of two dies forging)

Casting costraints inshure design of hollow spaces in the structure after optimization, it helps to genereate shape that can be manufacutred by multi axis milling.

Stress constraints in topology optimization, therefore allow for a greater weight saving and simplify the subsequent design work.Aim in using stress constraints in topology optimization is not to perfectly control the stress level but to avoid high stress concentrations, and thus generate a design that does not have to undergo severe modifications in order to be developed into a final design that fulfills the stress requirements. Ref. [12] Pockets on the original hinge were filled by Extrude command in Fusion 360

And this modifed hinge was inserted to D.S. CATIA software assambly where it was mounted to hinges and sandwich panel assembly.

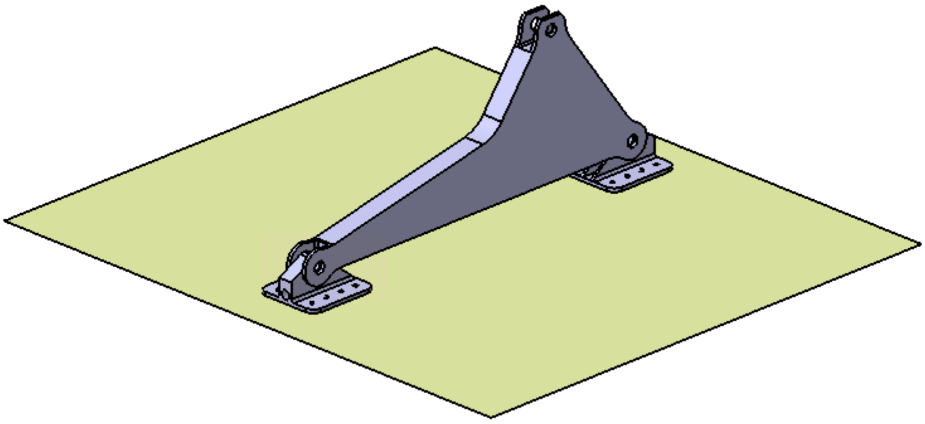

Figure 35. Modified hinge in D.S. CATIA work bench.

The finer mesh was used to achieve precize results.

Element Shape	Tet
Topology	Tet10
Global edge length	2
Node	120907
Elemet	695844

Table 13. mesh properties

Like in pervious optimizations RBE3 element is used to apply external load, RBE2 elements are used to constrain parts to each other and to the plate. Plate is fixed to all 6 DOFs.

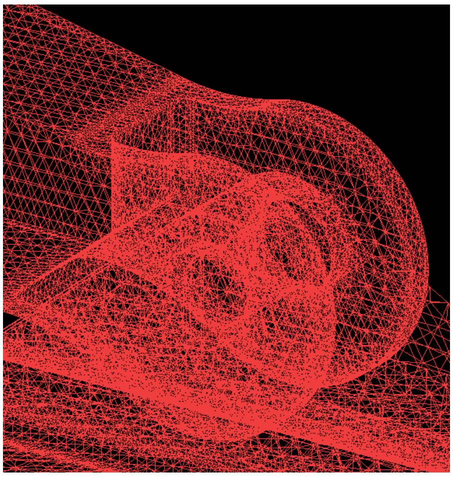

Figure 36. Mesh example on the front lug .

31 design sycles were needed to achieve optimization result

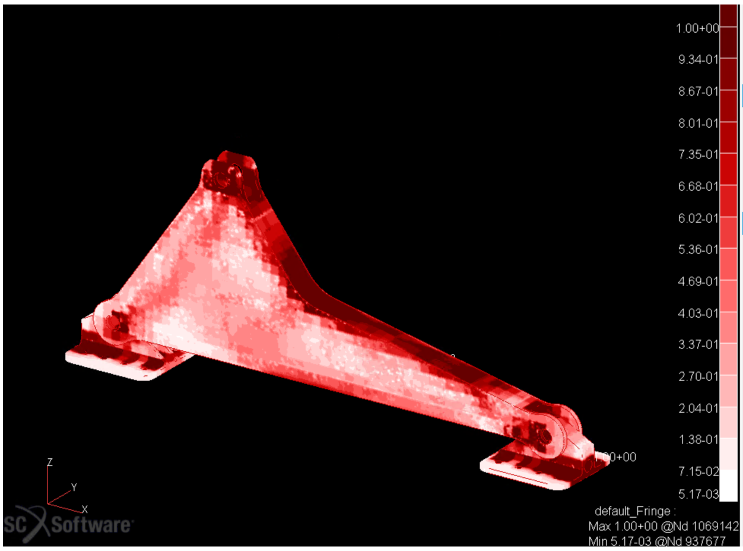

Figure 37. Optimization result.

M.S.C. PATRAN post-processing option doesn’t clearly show achived result, for Post-processing ParaView software will be used.

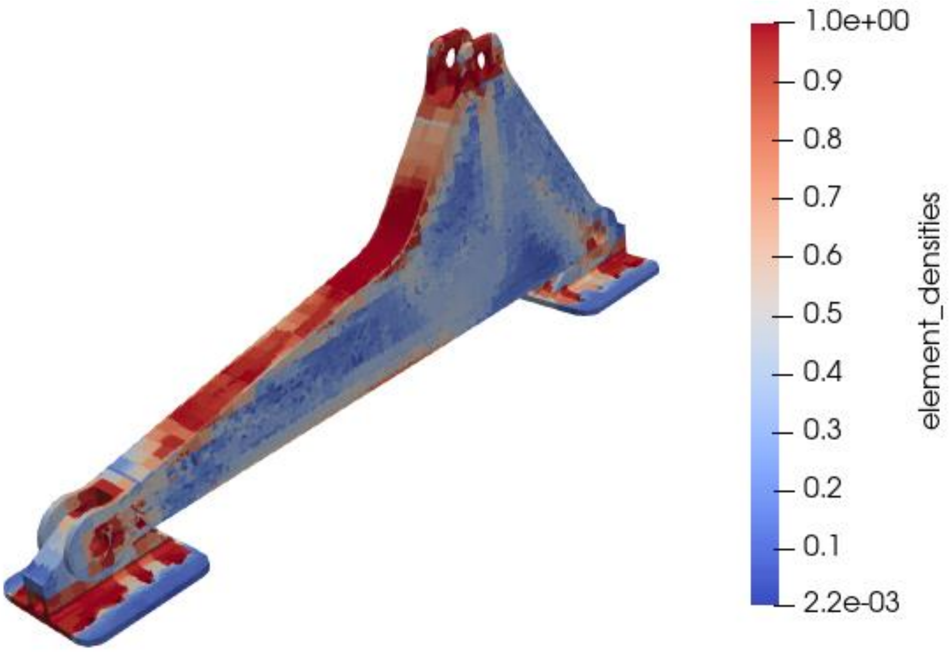

Figure 38. Optimization result in ParaView work bench.

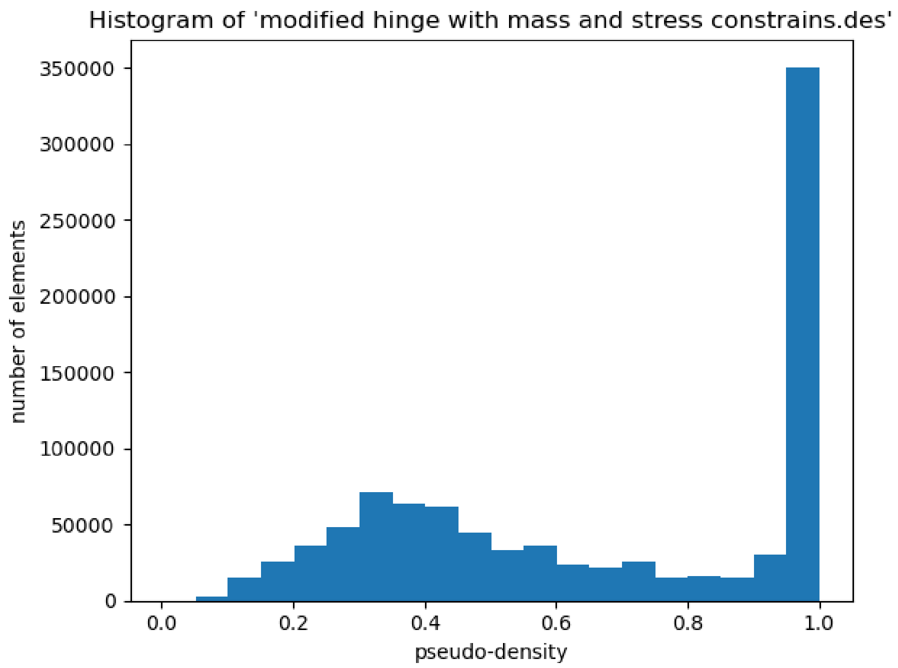

Figure 39. Histogram of modified hinge optimization

There are many elements with intermediate densities and it would be hard to decide which threshold shape will be used in the new design, compared to previous results where histogram showed rather 0 or 1 densities.

One of the reason is that the optimization formulation is hard to optimize. E.g. achieving low stress needs to add material, but the material is constrained on 0.25, so that optimizer have to make a trade-off and does not have "space" to converge to 0 or 1 design.

Figure 40. threshold 0.5

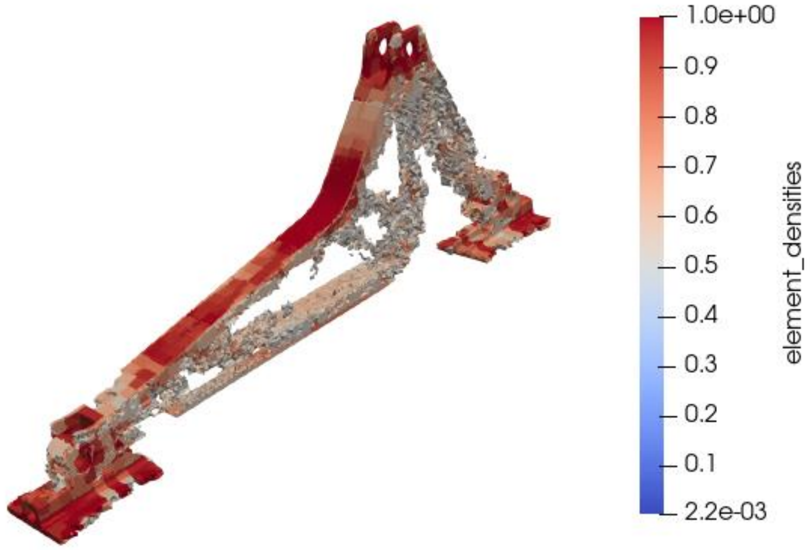

Figure 41. threshold 0.5

threshold 0.5 gives the outline for original part modification. Form this optimization importance of lower stiffnere is clear. Areas between upper and lower stiffneres can be removed on the original part.

4.4 Topology optimization of the modified hinge with mass and stress constraints

For the understanding the effect of elastic foundation on topology optimization result it is important to optimize part with same:

- Shape
- Material
- loading
- constraints
- mesh

but without constraining it to the elastic foundation.

Topology optimization will be done by using M.S.C PATRAN/Nastran software, for displaying results ParaView software will be used.

The hinge will be constrained to the brackets by two RBE2 elements at the lower lugs properties for pin RBE2 elements will be show at the table 14, properties of the bracket bolt RBE2 elements will be shown at table 15

Dependetn terms		
nodes(1)	DOFs (Max=6)	
node 7869:8985 986	UX,UY,UZ,RX,RZ	
independent terms		
	nodes(no max)	DOFs (Max=6)
	center node	

Table 14. Example pin constraint RBE2 elements properties

Dependetn terms		
nodes(1)	DOFs (Max=6)	
node 9569:10985 11	UX,UY,UZ,RX,RY,RZ	
independent terms		
	nodes(no max)	DOFs (Max=6)
	center node	

Table 15. Example bolt RBE2 element properties

Assembly is constrained by bracket bolt RBE2 elements to all 6 DOFs

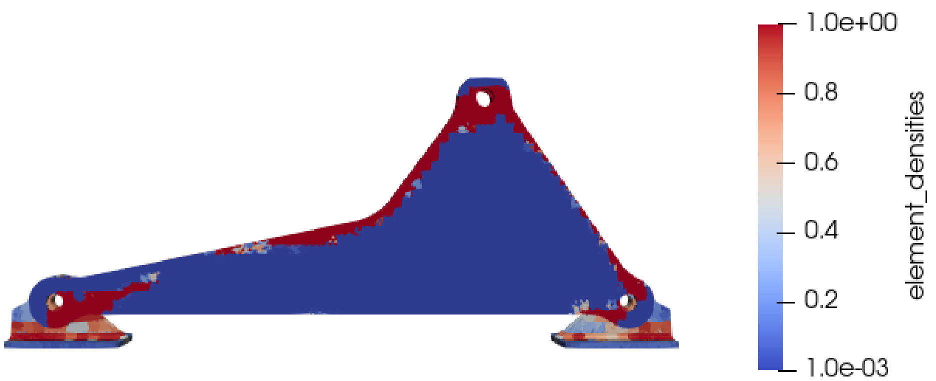

Figure 42. Result of the optimization without elastic foundation

From fig.42 we can see that most elements on the hinge surface have element densities close to zero. The histogram on fig.43 provides data about the quantity of elements with various densities. We can see that for the given design objective and constraints part have more elements than is required to withstand the applied load.

.

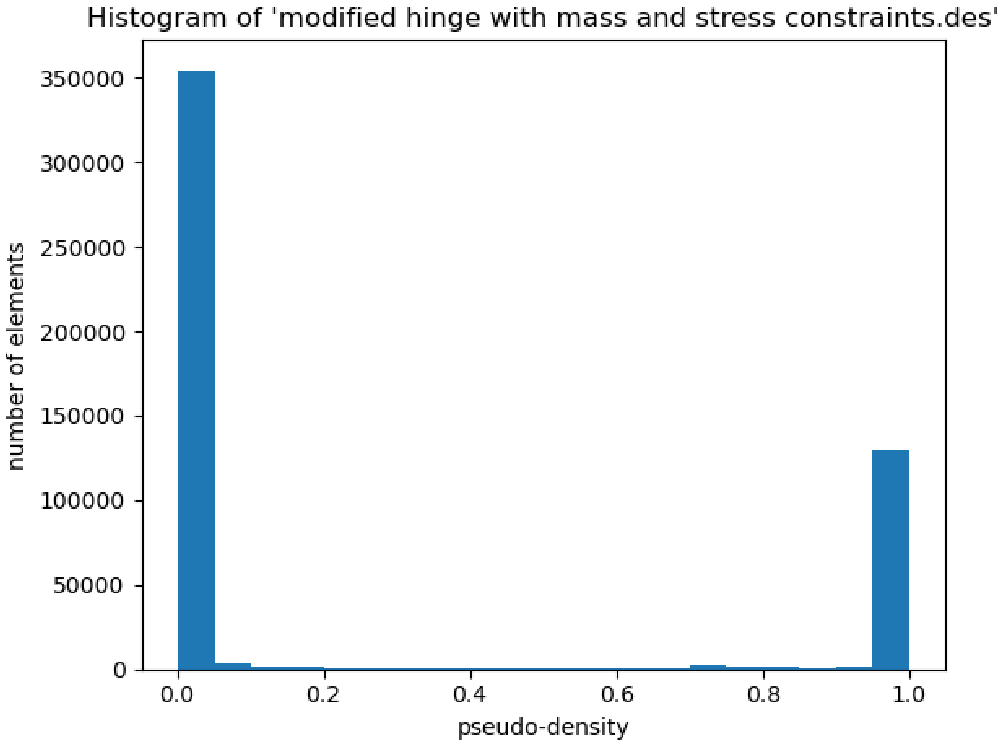

Figure 43. Histogram of topology optimization of the modified hinge without elastic foundation

From histogram we can see that most element densities are divided to 0 and 1 values for final design optimization we will use the shape of threshold 0.5 (fig.44)

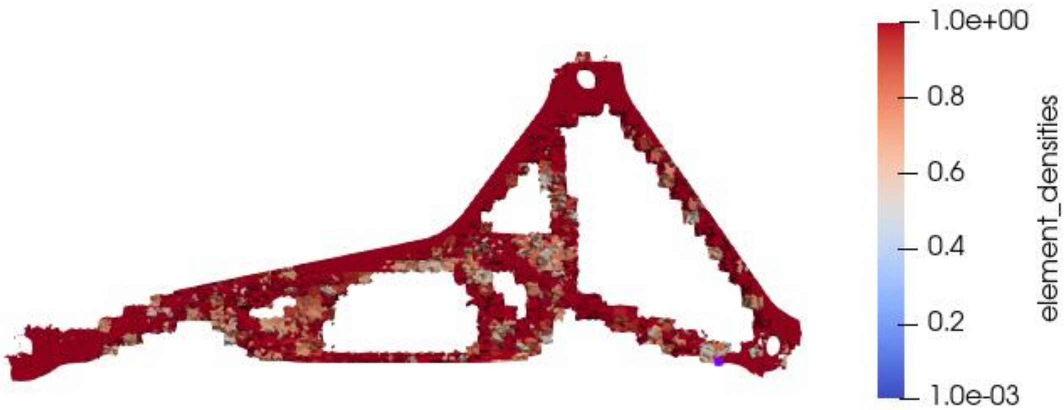

Figure 44.Threshold 0.5 of the modified hinge without elastic foundation

Comparison of Fig.44 and Fig.45 (threshold 0.5 of the topology optimization of the modified hinge on elastic foundation) provides us data that element number after optimization with elastic foundation is higher than without elastic foundation, comparison of this two results also show that element densities with a value close to 1 are more spread on optimization result without elastic foundation, so elements are much more loaded in that case.

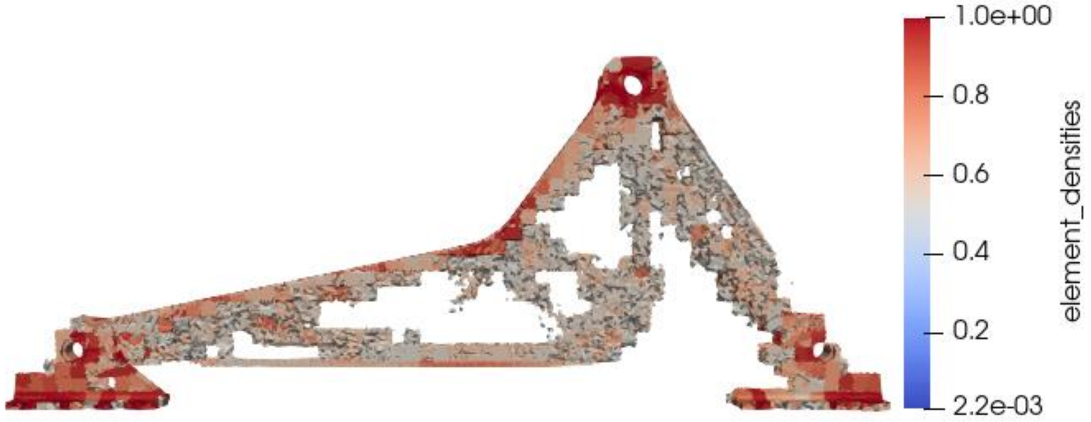

Figure 45. Threshold 0.5 of topology optimization of the modified hinge on the elastic foundation

4.5 Shape optimization of the hinge in FUSION 360 Software

FUSION 360 product of AUTODESK, Inc. software will be used for the shape optimization process, both original hinge and modified hinge with the larger domain will be optimized. The hinge will be fixed by pin and load will be applied on the upper lug pin surface.

To perform shape optimization part, have to be divided to elements, at the end of each element is a node, these elements and nodes make up a mesh.

Currently, all solid elements in Fusion 360 are tetrahedral (consisting of four triangular faces and six edges each). Linear tetrahedral elements have four nodes. Parabolic tetrahedral element adds a mid-side node along each edge, resulting in a total of ten nodes per element. There are two variations of Parabolic tetrahedral elements with and without curved edges. Which are automatically selected during meshing process.

Material properties are same to all optimization process.

For optimization of the original part, hinge is meshed with 1mm edge length elements.

some places on hinge will not be include to the optimization process these areas are Preserve regions. They are set up on lugs because brackets of the hinge are standardized part of production and any changes of the hinge lugs inside dimensions will require change of bracket shape. Preserved regions are marked by green color on the following figure.

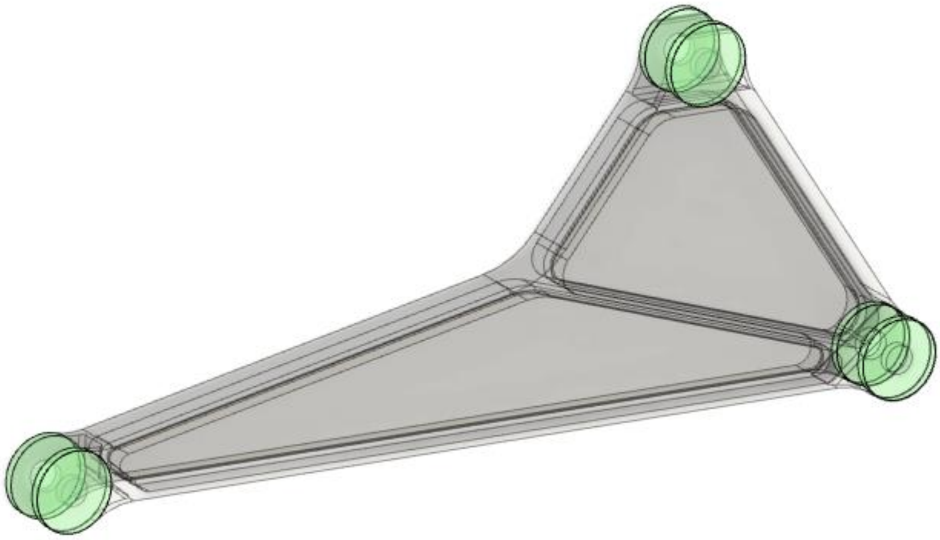

Figure 46. preserved regions (green)

Shape optimization criteria are

- Target mass ≤ 30%
- Stiffness Maximize

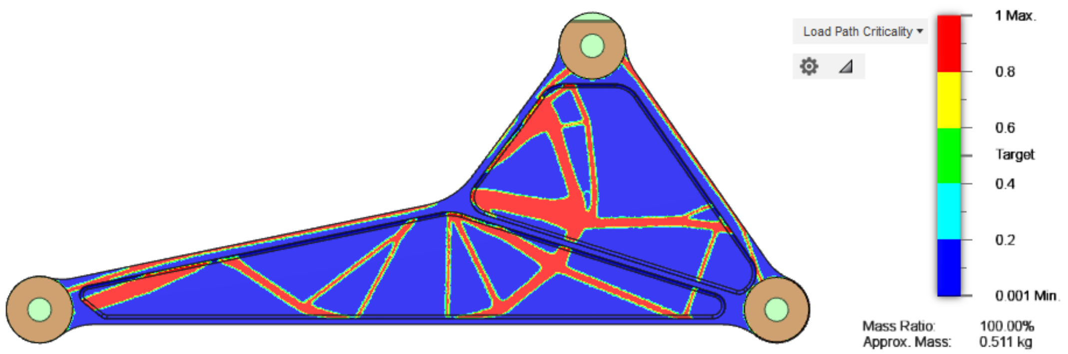

Figure 47. Original hinge Shape optimization Result

For residing original we are interested on shape of threshold 0.5 which is marked on the legend bar - Target

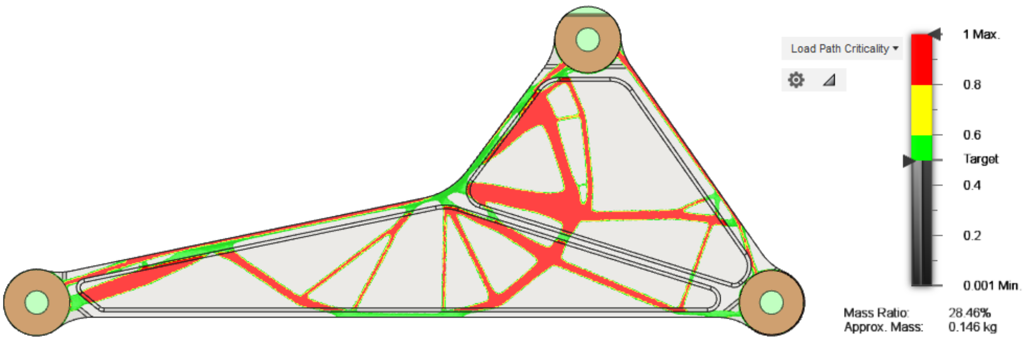

Figure 48. Threshold 0.5

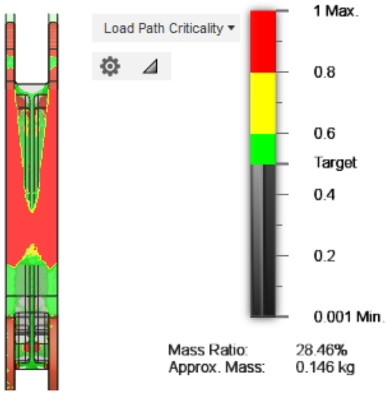

Figure 49. Rearview of the hinge, threshold 0.5

Load path criticality on legend bar represents a discrete variable that ranges from 0 to 1. A value of 1 represents region in the model that is critical to resist the applied load. A value of 0 represents a region in the model that is not critical to resist the applied load. When attempting to achieve the target mass, Shape Optimization removes the elements with the lowest Load Path Criticality values.

From this result, we can see the most critical areas of the hinge which resists to the applied load. Optimization for modified hinge is set up in same way with only difference in

Shape optimization criteria

- Target mass $\leq$ 25%

Because of the larger volume of the modified hinge.

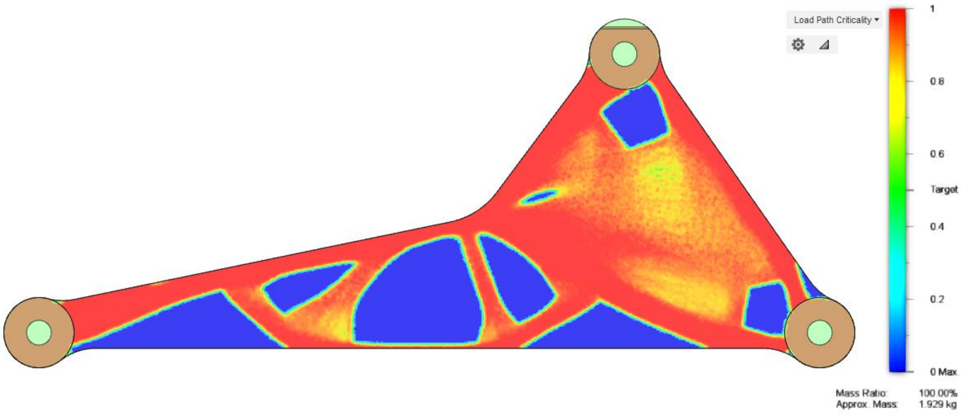

Figure 50. Modified hinge optimization result

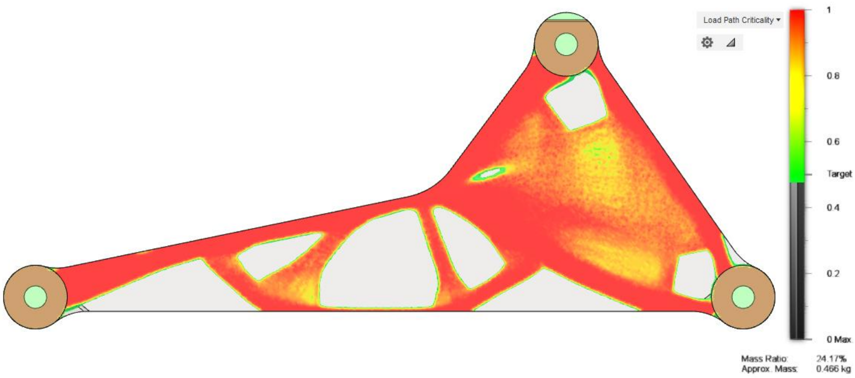

Figure 51. threshold 0.5

At threshold 0.5 weigt of the optimized hinge is 0.466kg which is 0.049kg lower than original hige. This diesign is hollow inside

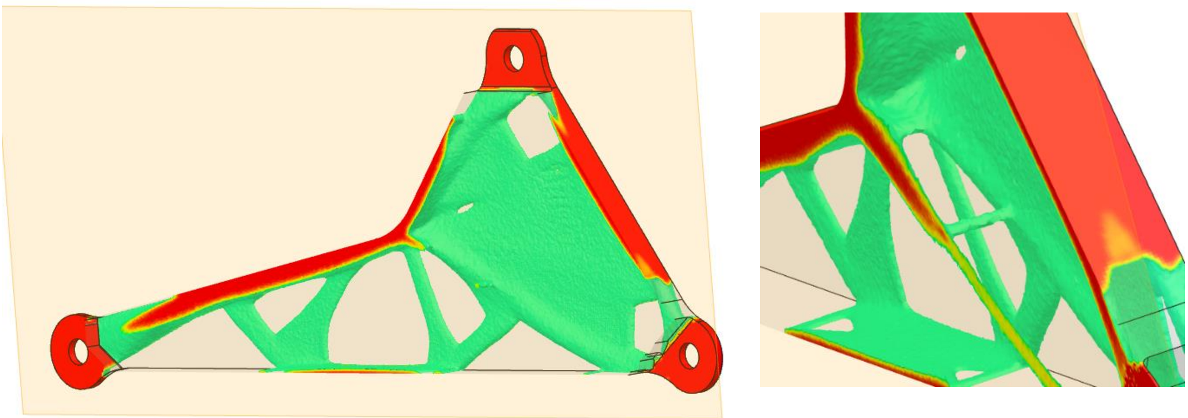

Figure 52. slice plane of optimized shape

This design shape cannot be the manufactured by classical manufacturing process such as milling, the additive manufacturing process is required.

5. Redesigning original hinge based on optimization results

Multiple optimizations were done to full fill the required information for modification of the original hinge. M.S.C. PATRAN and M.S.C. NASTRAN results were based on element densities, Fusion 360 results are based on load path crticality. Combination of these two approaches will give effective inputs for original hinge modifications.

5.1 Combining results

From stress analysis of the hinge, critical places of the structure become visible (front lug)

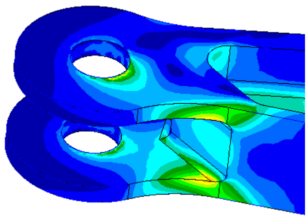

Figure 53. Maximum stress location on the hinge

From optimization with mass constraints it become clear that is possible to remove material on rear stiffener.

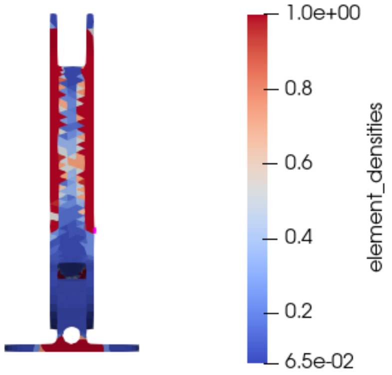

Figure 54. Rearview of hinge (result of optimization with mass constrains)

From optimization with buckling constraints binding of lower stiffener is clear. Removing this part of structure will have big impact on part stability under external load.

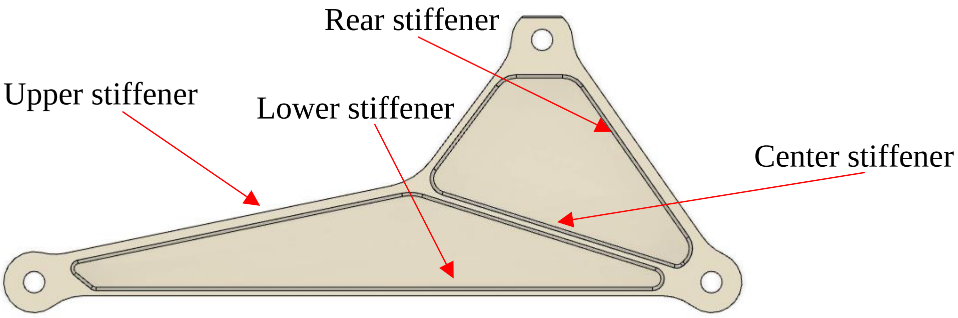

Figure 55. Stiffener locations

Results from the optimization of the modified hinge with mass and stress constrains show an alternative way of how area between upper, lower and rear stiffeners can be filled with material.

Results from FUSION 360 are providing information about load path crticality in the structure of hinge. Modified hinge optimization result is 0.05kg lighter than the original part, but due to complex shape it would be inefficient price for manufacturing on comparison to the classical manufacturing process.

For cost and manufacturing process efficiency modified shape have to be manufactured by classical manufacturing approaches like multi axis milling, casting, or forging.

5.2 Redesigning of the original hinge

Inputs for redesigning of the original part are clear. The final shape will contain corrections form result which was achieved from static stress analysis and optimizations.

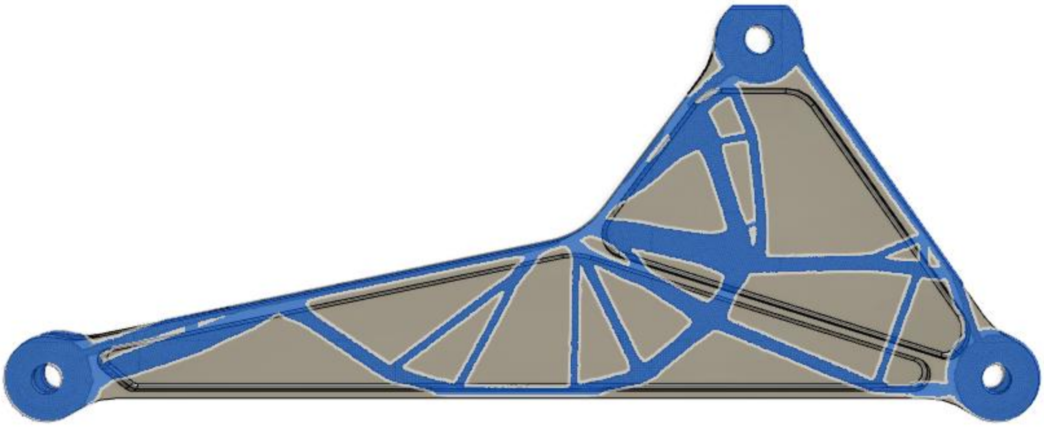

Figure 56. Layering of mesh from Fuson360 on original hinge

From static stress analysis result maximum stress on hinge occurs on the connection between front lug and lower stiffener that place will be redesigned. Hinge redesigning constraints are that it must change original hinge in galley structure without modification of brackets and their locations. This means: The distance between lugs will be same.

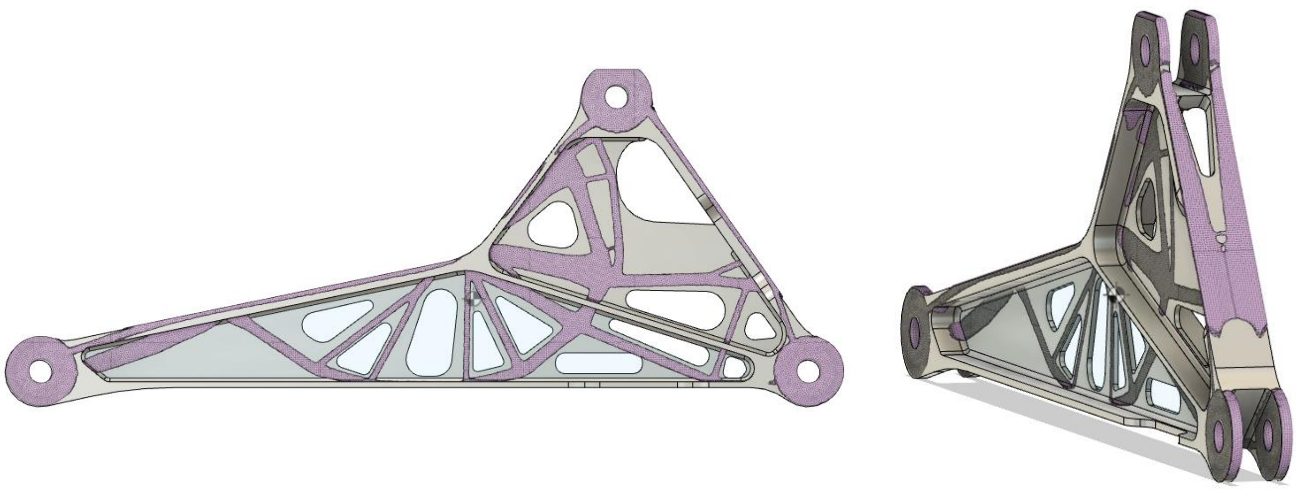

Figure 57. Layering of FUSION360 optimization mesh on redesigned hinge

Cutouts on original part are based on results from the optimization process, critical place on the front lug is reinforced.

- Weight of original hinge 515 grams
- Weight of redesigned hinge 451 grams

Weight reduction after optimization 13% (64 grams)

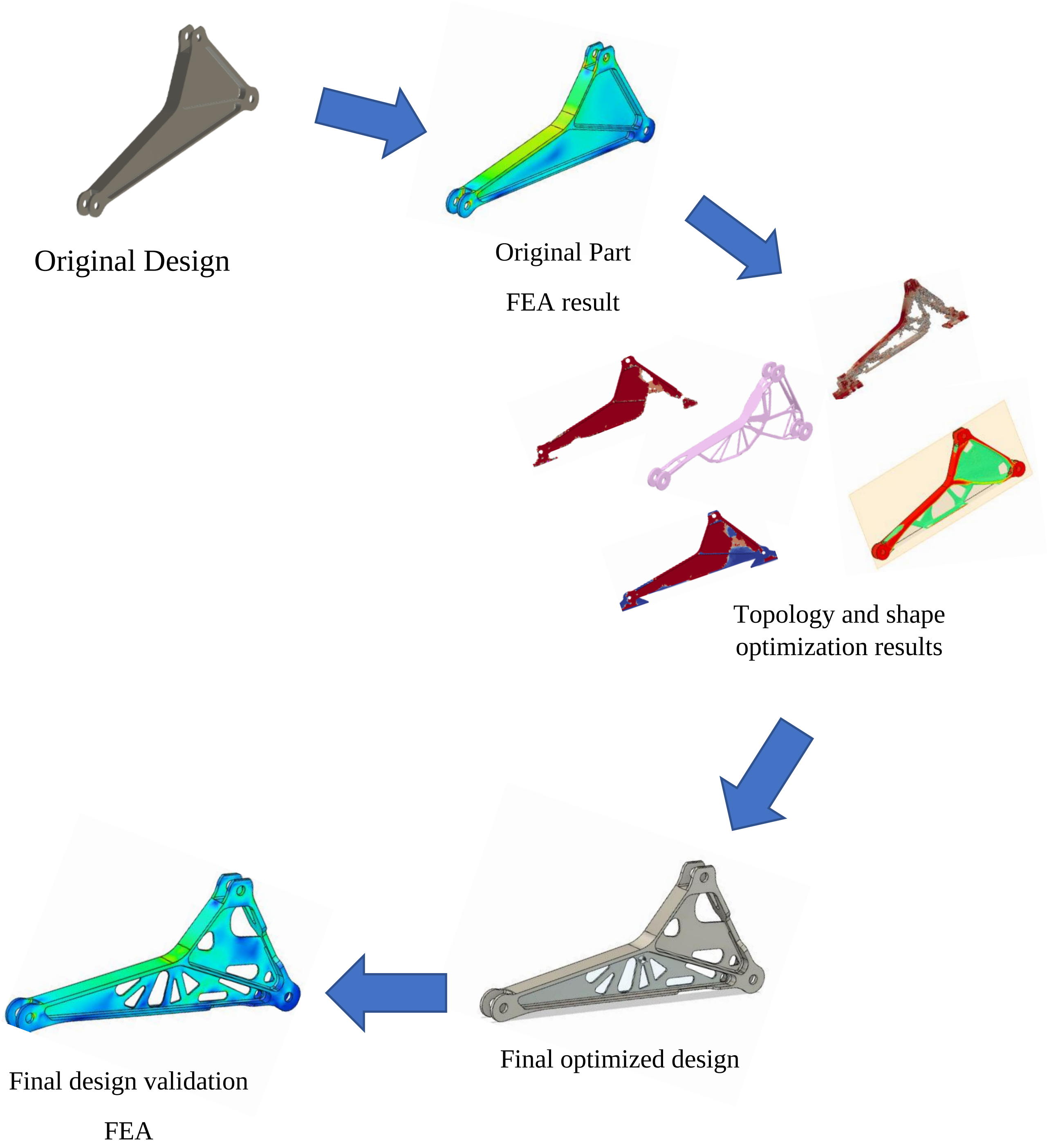

Figure 58. Optimization process

5.3 New design hinge validation based on stress and buckling analysis

Validation of the redesigned hinge is required to ensure that it will withstand applied external load without higher stress compared to original shape hinge.

For Finite Element Analysis M.S.C. PATRAN/NASTRAN software will be used.

STEP file is imported to M.S.C. PATRAN workbench and mesh seed are created on places where local mesh control is needed (edges, small radiuses, lug holes), the same external load is applied to the upper lug -15849[N]. Material properties are same.

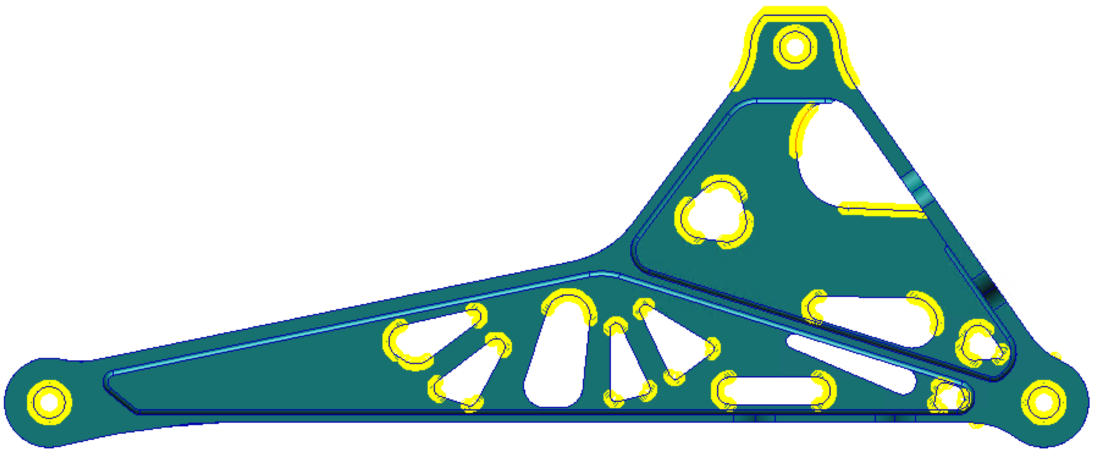

Figure 59. modified hinge at work bench with mesh seed

Element Shape	Tet
Topology	Tet10
Global edge length	8

Table 16. Mesh properties for redesigned hinge

Hinge is constrained by RBE2 with one degree of freedom (RY). Properties of RBE2 elements are provided in Table 17

Dependetn terms			
nodes(1)		DOFs (Max=6)	
node 1569:1785 118		UX,UY,UZ,RX,RZ	
independent terms			
	nodes(no max)		DOFs (Max=6)
	center node		

Table 17. Example constraint RBE2 elements properties

RBE2 elements constraint the part also. Constraint properties are in the table below.

Translations <T1 T2 T3>
<0., 0., 0.,>
Rotation <R1 R2 R3>
<0., 0., 0.,>

Table 18. Constraint properties

External loading -15849[N] is applied at top lug RBE3 element properties of that element are provided at the following Table 19.

Dependetn terms			
nodes(1)		DOFs (Max=6)	
3		UX,UY,UZ,RX,RY,RZ	
independent terms			
coeficient	nodes(no max)		DOFs (Max=6)
1.00	46853:46878 65…		UX,UY,UZ

Table 19. RBE3 element properties

Constitutive model	Linear elastic	unit
Elastic modulus	70 000	[MPa]
Poissons ratio	0.3	[-]

Table 20. Material properties

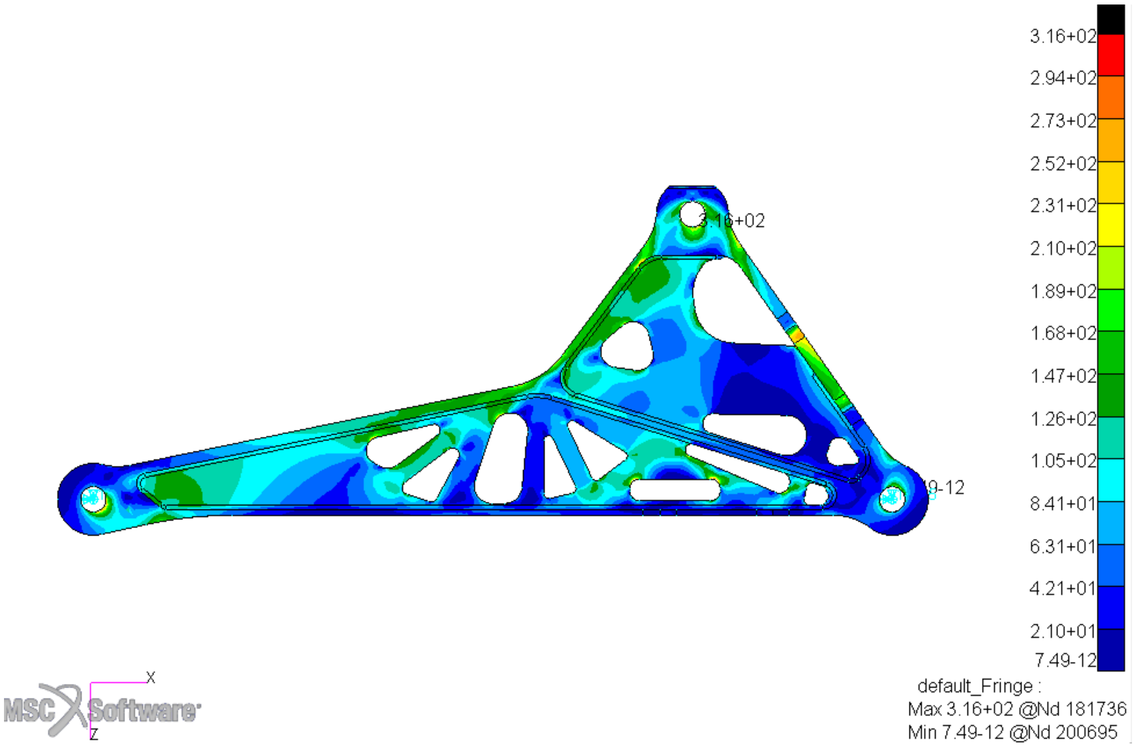

Figure 60.Result of stress analysis, Maximum Von Mises stress 316 [MPa]

Figure 60 represents stress distribution at the redesigned hinge. compared to results of FEA with and without elastic foundation 336[MPa] (Fig.25) and 403 [MPa] (Fig.19) maximum stress on redesigned hinge is lower.

Maximum stress at the redesigned hinge occurs at the upper lug hole inside the surface. This part of the hinge wasn't modified.

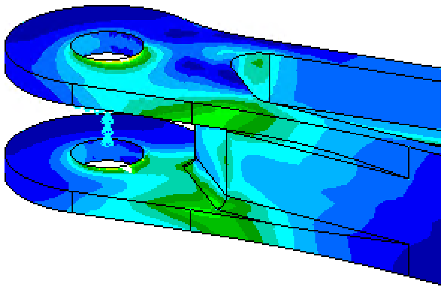

Figure 61.Front lug lower surface after redesigning and reinforcement

Maximum stress on the lug lower surface is 189 [MPa] significantly lower (44%) than the original hinge FEA on elastic foundation.

Even if static stress determines thar part will not lose stability at applied load, buckling failure can occur. Because buckling necessarily isn't function of stress, it is one of geometric instabilities, meaning that it can occur at lower stress values.

For buckling analysis FUSION 360 with NASTRAN solver will be used.

Setup is similar to FEA, material properties and external loading values are same.

The part is constrained specialized pin constrain at FUSION 360 software.

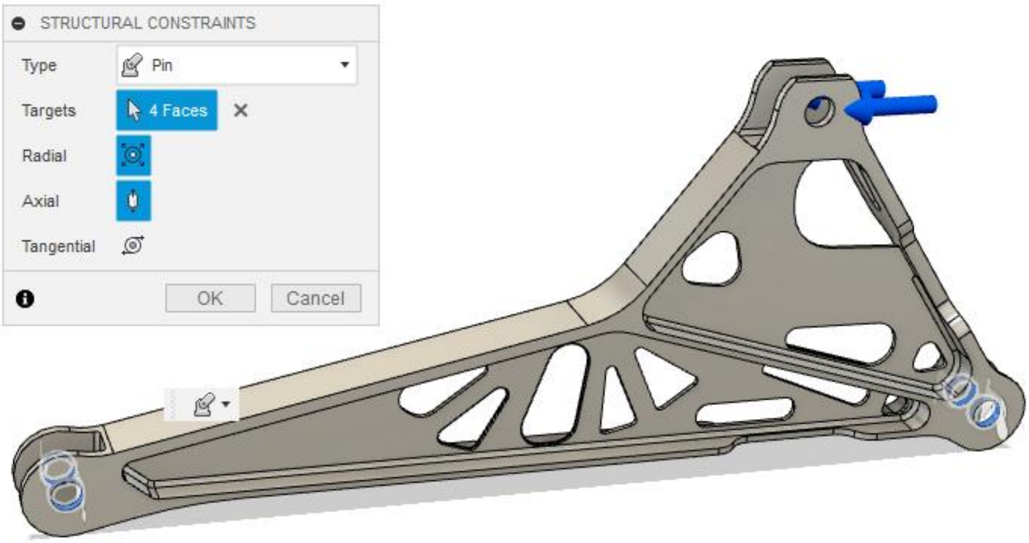

Figure 62.Redesigned hinge structural constraints for buckling analysis

Element Shape	Tet
Global edge length	8

Table 21. Mesh properties

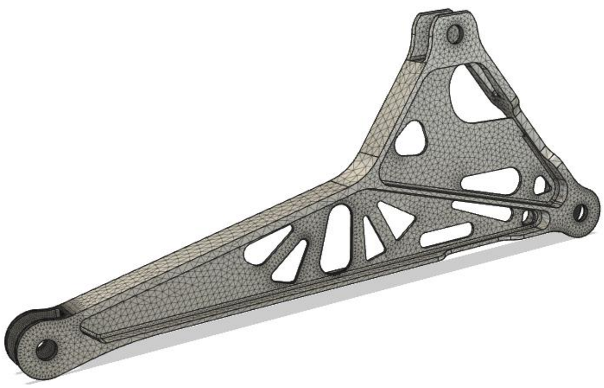

Figure 63.Hinge discretization with local mesh control

Constitutive model	Linear elastic	unit
Elastic modulus	69000	[MPa]
Poissons ratio	0.33	[-]
yeld strength	275	[MPa]
ultimate tensile stregnt	310	[MPa]

Table 22.Material setup for buckling analysis

(6061 aluminum from FUSION 360 material library)

15 buckling modes were analyzed but we are interested at lowest mode with positive and negative load buckling factor.

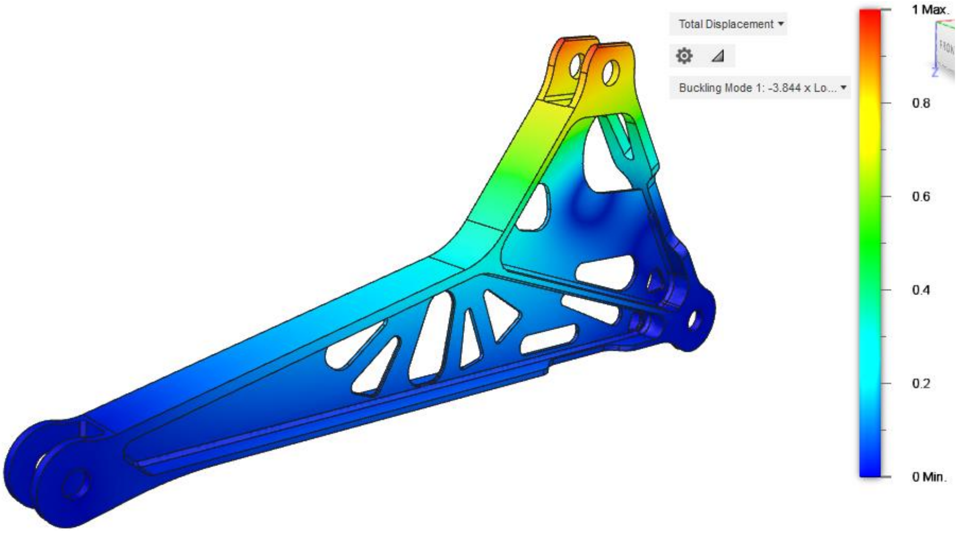

Figure 64.Buckling mode 1, Buckling load factor -3.844

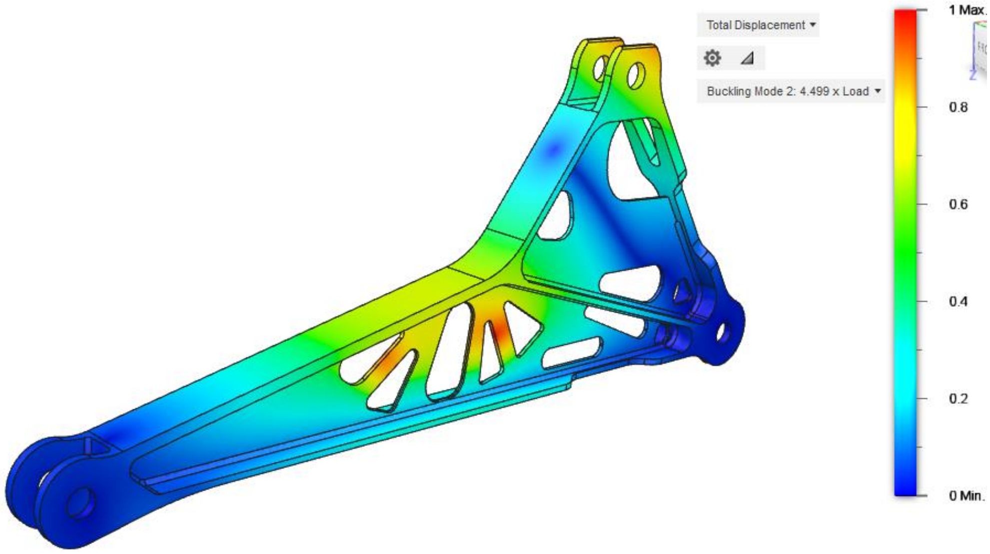

Figure 65.Buckling mode 2, buckling load factor 4.499

The Buckling Load factor can be interpreted in the following Table 23

Buckling Load Factor	**Buckling Prediction**	**Conclusion**
BLF>1	Not expected	Applied loads are below critical loads.
BLF=1	Expected	Applied load is the critical load and buckling will occur.
BLF<1	Expected	Applied load is above the critical load and buckling will occur.
-1<BLF<0	Expected in reversed load scenario	Applied load is above the critical load magnitude but is in the opposite direction. This type of result could cause buckling in another mode. Investigation to understand if the load is correct and would cause tension should be done.
BLF=-1	Expected in reversed load scenario	The applied load is the critical load magnitude but is in the opposite direction. Other modes should be checked to validate buckling will not occur.
BLF<-1	Not expected	The applied load is below the critical load and in the opposite direction. Buckling will not occur even in a different Mode.

Table 23. The Buckling Load factor interpretation

Results from buckling analysis fulfill requirements to proof that redesigned hinge can withstand applied load without losing stability. Ideal material for it manufacturing of redesigned hinge will be 7075 aluminum, because stresses of static analysis are above yield strength of 6061 aluminum 241 [Mpa].

Comparison of displacements of original and modified hinge are done in FUSION360 software. Material is the same, applied ultimate load is the same.

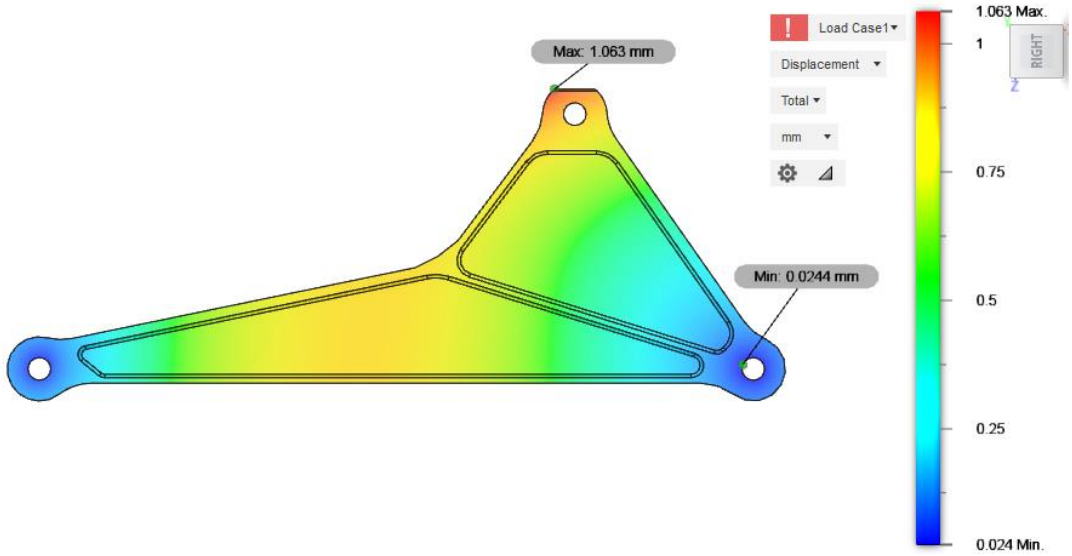

Figure 66. displacement of original hinge (Max. 1.063 [mm])

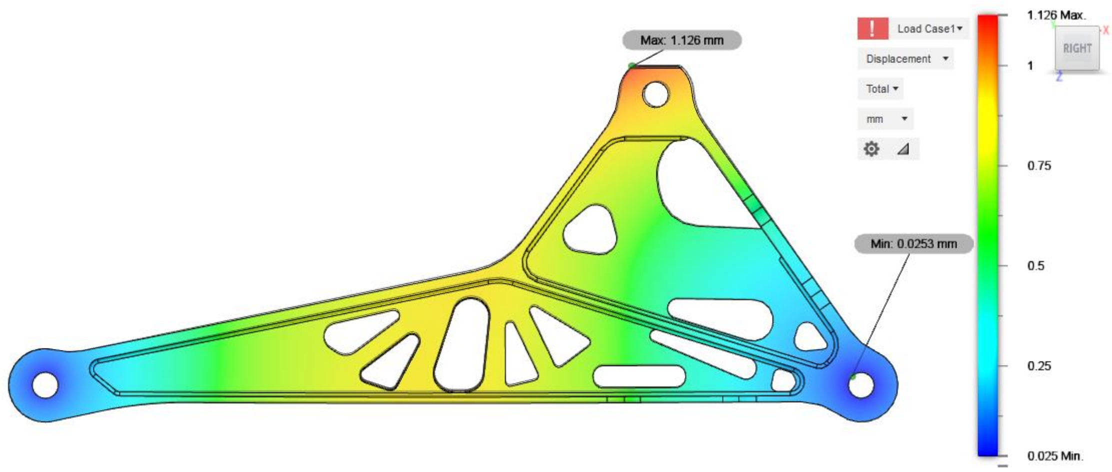

Figure 67 displacement of original hinge (Max. 1.126 [mm])

Results from displacement analysis show minor 1% increasing of maximum displacement 0.937 [mm] .

6. Conclusion

From the case study, it can be concluded that topology optimization with different design objectives and constraints is a powerful design tool to reduce the weight of structural products. The effect of elastic foundation on Finite element analysis was clearly shown at Fig. 24 bracket stress comparison without and with elastic foundation. Decreasing of local stresses affects especially on stress constrained topology optimization which is shown at fig.44.

the reduction of weight saves a huge amount of material and efficiency at aircraft structure. This master thesis shows that resultant shape after topology optimization on the elastic foundation can be produced not only by additive manufacturing but also manufactured by classical manufacturing processes like multi axis milling, two die casting or forging techniques.

For final shape design domain original part was used. This strategy directly connects to manufacturing, the same approach for manufacturing process can be applied. Connection to the brackets will be same without any modifications and rod for application of external load will not also require modifications. This strategy significantly decreases cost of production

From the case study result, which is 13% weight reduction, it can also be concluded that topology optimized design not only for additive manufacturing can reduce a huge portion of the mass at the aerospace industry.

References

[1] KOHN, R.V. and STRANG, G., Optimal Design and Relaxation of Variational Problems, Communications on Pure and Applied Mathematics, 39, 1986, pp. 1-25 (Part I) 139-182 (Part II) 353-377 (Part III)

[2] AMBROSIO, L. and BUTTAZO, G., An Optimal Design Problem with Perimeter Penalization, Calculus of variations and Partial Differential Equations 1, 1993, pp. 55-69.

[3] SIMP Method for Topology Optimization, available from: http://help.solidworks.com/2019/english/SolidWorks/cworks/c_simp_method_topology.htm?format=P&value= Last accessed 10.05.2020

[4] Review of Wire Arc Additive Manufacturing for 3D Metal Printing - Scientific Figure on ResearchGate. Available from: https://www.researchgate.net/figure/Classification-of-metal-additive-manufacturing-technologies_fig1_332886203 Last accessed 15.05.2020

[5] A Metal Powder Bed Fusion Process in Industry: Qualification Considerations - Scientific Figure on ResearchGate. Available from: https://www.researchgate.net/figure/Main-steps-of-the-additive-manufacturing-AM-process_fig1_337263199 Last accessed 15.05.2020

[6] About additive manufacturing, Powder Bed Fusion, Additive Manufacturing Research Group, Loughborough University. https://www.lboro.ac.uk/research/amrg/about/the7categoriesofadditivemanufacturing/powderbedfusion/ Last accessed 15.05.2020

[7] GIBSON, IAN, ROSEN W. David, and STUCKER Brent. “Additive Manufacturing Technologies: Rapid Prototyping to Direct Digital Manufacturing.” Springer, 2010

[8] MSC Nastran 2012 Design Sensitivity and Optimization User’s Guide PP. 27-28

[9] Multiphysics Cyclopedia Finite Element Mesh Refinement, https://www.comsol.com/multiphysics/mesh-refinement Last accessed 21.05.2020

[10] TOVAR, Andres & Patel, Neal & Niebur, Glen & Sen, Mihir & Renaud, John. (2006). Topology Optimization Using a Hybrid Cellular Automaton Method With Local Control Rules. Journal of Mechanical Design. 128. 1205-1216. 10.1115/1.2336251.

[11] FERRARI F., SIGMUND O. Revisiting topology optimization with buckling constraints. Struct Multidisc Optim 59, 1401–1415 (2019). Available at https://doi.org/10.1007/s00158-019-02253-3 Last accessed 21.06.2020

[12] HOLMBREG Erik, TORSTENFELT Bo KLARBRING Anders, Stress constrained topology optimization. Springer-Verlag Berlin Heidelberg 2013 PP. 33-35

List of Figures

List of Tables