Gümüşhane University Journal of Science

GUFBD / *GUJS* (2024) 14(1): 325-335 doi: 10.17714/gumusfenbil.1329278 Research Article

Topology optimization and numerical verification in an aircraft engine bracket

Uçak motor braketinde topoloji optimizasyonu ve sayısal olarak doğrulaması

İsmail SARAÇ*[1], Alp Timuçin KOÇAK[2]

[1]*Aksaray Üniversitesi, Mühendislik Fakültesi, Makine Mühendisliği Bölümü, 68100, Aksaray*

[2]*Türk Havacılık ve Uzay Sanayii, 06980, Ankara*

• Received: 18.07.2023 • Accepted: 28.12.2023

Abstract

The importance of saving energy and materials by lightening structures is constantly increasing. With its powerful software capabilities, Topology Optimization produces solutions for this exact purpose. In addition, thanks to Topology Optimization, more innovative and competitive structures can be produced. The development of additive manufacturing methods has also increased interest in Topology Optimization. In Topology Optimization, volumetric elements that do not carry any load or carry little load are removed from the structure. Thus, lighter, but sufficiently durable structures can be obtained. In this study, the topology optimization of a bracket used as a fastener in a jet engine was carried out using ABAQUS Finite Element software. Required bracket geometry, load conditions, and material information were obtained from an online design competition announced by General Electric. Ti6Al4V alloy was used as the bracket material. At the beginning of the study, static analysis was performed on the original bracket model to obtain the load paths required for topology optimization. As a result of the static analysis, the load paths within the jet engine bracket were determined and topology optimization was applied to the bracket to minimize the mass without reducing the rigidity. As a result of the analysis studies, it has been proven that nearly 80% material savings can be achieved from the bracket thanks to topology optimization.

Keywords: Bracket, Finite elements analysis, TiAl6V4, Topology optimization

Öz

Yapıları hafifleterek enerji ve malzeme tasarrufu sağlamanın önemi gittikçe artmaktadır. Güçlü yazılım imkanlarıyla Topoloji Optimizasyonu tam bu amaca yönelik olarak çözümler üretmektedir. Bunun yanında Topoloji Optimizasyonu sayesinde daha yenilikçi ve rekabetçi yapılar üretilebilmektedir. Eklemeli imalat yöntemlerinin gelişimi de Topoloji Optimizasyonuna olan ilgiyi arttırmıştır. Topoloji optimizasyonunda, yük taşımayan veya az yük taşıyan hacimsel elemanlar yapıdan çıkarılır. Böylece daha hafif, fakat yeterince dayanıklı yapılar elde edilebilir. Bu çalışmada, bir jet motorunda bağlantı elemanı olarak kullanılan bir braketin topoloji optimizasyonu ABAQUS Sonlu Elemanlar yazılımı kullanılarak gerçekleştirilmiştir. Gerekli braket geometrisi, yük koşulları ve malzeme bilgileri, General Electric tarafından duyurulan bir çevrimiçi tasarım yarışmasından alınmıştır. Braket malzemesi olarak Ti6Al4V Titanyum alaşımı kullanılmıştır. Çalışmanın başlangıcında, topoloji optimizasyonu için gerekli olan yük yollarını elde etmek için orijinal braket modeline statik analiz yapılmıştır. Statik analiz sonucunda, jet motoru braketi içindeki yük yolları belirlenmiş ve brakete, rijitliği düşürmeden kütleyi minimuma indirmek için topoloji optimizasyonu uygulanmıştır. Yapılan analiz çalışmaları sonucunda topoloji optimizasyonu sayesinde braketten %80'e yakın malzeme kazancı elde edilebileceği kanıtlanmıştır.

***Anahtar kelimeler**: Braket, Sonlu elemanlar analizi, TiAl6V4, Topoloji optimizasyonu*

* İsmail SARAÇ; ismailsarac@aksaray.edu.tr

ISSN: 2146-538X e-ISSN: 2146-538X https://dergipark.org.tr/tr/pub/gumusfenbil

1. Introduction

The basic objective of topology optimization is to determine the non-load-bearing or relatively low load-bearing volumetric elements in the structure and to ensure that they are volumetrically removed from the structure. As the non-load-bearing elements will be removed from the structure, the topological density of the structure will decrease, but its strength will not be affected significantly. In market conditions where structures are not only strong but also low cost, aesthetic values have also gained value on the side of the end user. When it is desired to go beyond the classical design and analysis methods, one of the effective methods used is topology optimization.

To perform a topology optimization, the designed CAD geometry is subjected to structural analysis to determine load paths. Afterwards, the geometry obtained by subjecting the structure to topology optimization is corrected and its sharpness is softened. The corrected geometry is subjected to structural analysis for the last time and it is seen whether it successfully meets the applied loads.

It is aimed to optimize the vehicle parts designed with structural optimization in terms of size, shape and topology. One of the most effective solutions to reduce the fuel consumption of vehicles and therefore the emission rates is to reduce vehicle weight. Nowadays, weight reduction studies with topology optimization method are widely carried out in the automotive industry. (Kahraman & Küçük, 2020). Topaç et al. (2017) carried out the structural design of the lower part to be used in the front suspension of a military vehicle with the help of topology optimization. In the final design, a 19.25% mass reduction was achieved compared to the preliminary design. Koçar (2018), in his study on dry cargo trailers, stated that the material reduction process performed to lighten the trailer chassis negatively affects the fatigue life because it causes stress concentrations. The numerical model was verified by measuring stress values at critical points on the chassis. As a result of the study, it was determined that weight reduction through thickness optimization was more appropriate. Top et al. (2019), applied topology optimization to the handbrake fastener, which will be produced using the selective laser sintering (SLS) method, one of the additive manufacturing technologies, and lightened the structure.

In aircraft, if the weight that the aircraft will have to carry is reduced, the flight will be more efficient and fuel savings will be achieved. The structure obtained as a result of topology optimization is suitable for production with additive manufacturing. By the end of optimization, complex geometries that cannot be produced with conventional production methods are generally obtained. Due to the nature of additive manufacturing, it is very suitable for the production of topologically optimized parts in terms of its flexible production method, since it provides layer-by-layer progress by adding parts rather than removing them (Attaran, 2017; Brighenti et al., 2021; Saleh Alghaamdi et al., 2021).

The first numerical procedure method for topology optimization with the finite element method was detailed by Rossow and Taylor (1973). In the late 1980s, Bendsøe and Kikuchi (1988), dealt with this issue in much more detail.

Parts that are being used to fix the load carrying parts at the land, sea and air vehicles are called as brackets or fittings. Aircraft brackets are different types of structural components which stands for carrying the manuever loads of the aircraft when flight control surfaces (aileron, flap, elevator etc.) axises are attached and engine structure is mounted.

Aircraft brackets are subjected to compressive, tensile, shear and combined loads. These brackets are usually manufactured from 7000 series aluminum alloys like 7050-T7451 or 7010-T7651, by forging or NC machining methods. Those aluminum alloys are able to carry heavy loads with their excellent strength values being much lighter than steel. Titanium alloys are extremely expensive, but they are the most effective solution when very high strength and lightness are desired. If the loads on the structure are very high and have a compressive character, Titanium alloys are preferred.

A bracket, manufactured from Titanium alloy may be heavy if it’s not optimized. Plastics, composites and ceramic materials cannot always be used in those kinds of structures because of their properties (being soft, brittle, having low strength etc.). It’s important in aviation to design light structures without ignoring the safety. With topology optimization, it’s possible to get lighter structures without reducing the strength. Topologically optimized parts cannot usually be produced with conventional manufacturing techniques because of their complex shapes. But with recently used 3D printing methods like SLM (selective laser melting i.e.) help us to manufacture these kind of metal parts. In the optimization, the loads applied to the aircraft are gradually transferred to the brackets, thus obtaining the loading conditions of the bracket. These loads are simulated separately for all load cases. Initial loads are obtained from a method called G-FEM (global finite element method) and detailed partial loads are obtained from D-FEM (detail FEM).

In this study, it is aimed to lighten an aircraft engine mounting bracket by using the Topology Optimization technique without reducing the strength value. Accordingly, firstly, the load paths were determined by performing linear static analysis on the bracket whose loading map is known. Then, Topology Optimization was applied to the bracket according to the condition of reducing its volume by 80%. The reason for choosing a volume reduction ratio as high as 80% is to show the effectiveness of Topology optimization. Under four different loads, the lightened bracket obtained as a result of this optimization, which was made by taking into account the result of the combined cluster of the loads, was subjected to the four load effects separately and the topology optimization was verified.

2. Material and method

Required bracket geometry (Figure 1), the loads and material specifications were taken from an online design challenge performed by General Electric (Grabcad, 2022). 4 load cases were directly determined by GE according to the real flight conditions. As stated, Ti6Al4V alloy was defined for the bracket material and material properties were employed (ASM, 2023). Ti6Al4V alloy properties were given in the Table 1. Ti6Al4V is a material suitable for Selective Laser Melting Additive Manufacturing, one of the Powder Bed Melting methods, it is also the most extensively used Titanium alloy and has a large number of applications in the aerospace and automotive industries.

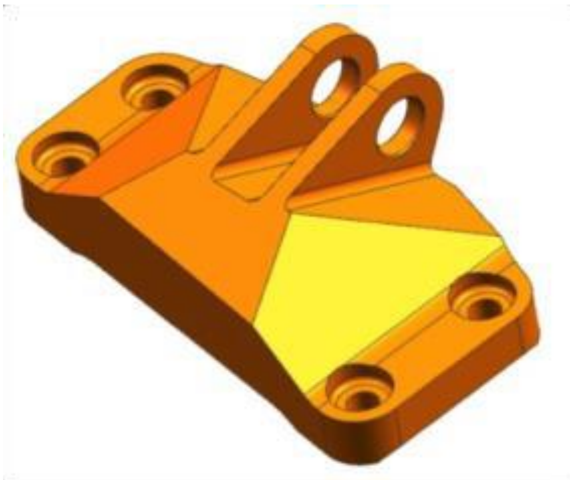

Figure 1. Bracket geometry (Grabcad, 2022).

Table 1. Material specifications of Ti6Al4V (ASM, 2023).

Parameter	Magnitude	Unit
Elasticity modulus	113800	MPa
Poisson's ratio	0.33	
Yield point	903	MPa
Density	4.5	g/cm^3

For the static and topology analysis, the ABAQUS finite element software was used. Bracket material properties need to be introduced to the program. At this point, it is necessary and sufficient for the analysis to be made to introduce the elasticity module, yield strength and Poisson ratio of the material to the system. As the elastic property of the material, the modulus of elasticity (E) is 113800 MPa and the Poisson ratio (ʋ) is 0.33. When the load conditions are applied to the part, the reaction of the part against all these loads must remain in the linear elastic region, that is, in the Hooke's law region. The yield strength, which characterizes the plastic property of the material, is also introduced to the system as 903 MPa. In addition to these three properties, entering the density value of the material is sufficient for linear static analysis. Because the material is metal powder, its mechanical behavior is isotropic and this property was introduced into the program.

Besides the material and geometry, four discrete load conditions that the bracket is encountered were stated. Also, it is declared that the bracket is attached with 4 rigid fasteners to a structure and loads are applied to the rigid pin, which is in contact with the clevis arms of the bracket. In Figure 2, four discrete load conditions are depicted and rigid pin is shown in purple.

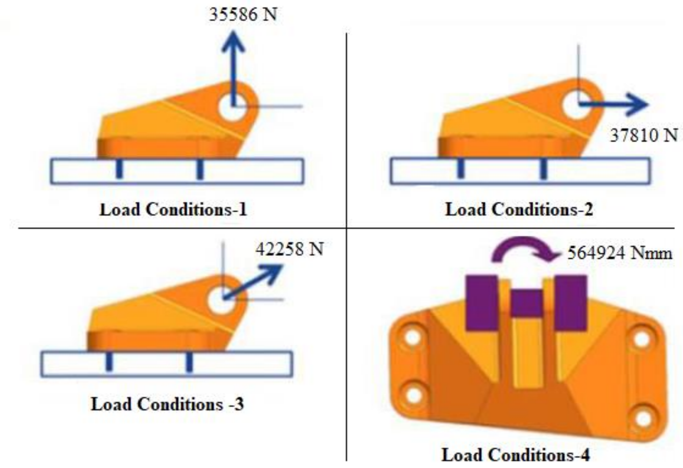

Figure 2. Load conditions

2.1. Static analysis

Before performing a Topology Optimization, static analysis should be done to define the load paths. To simulate the rigid pin, an analytical or discrete rigid cylindrical surface, which has the same diameter with clevis arms holes, could be created. However, in that case, a contact definition would be required between the bracket clevis arms and the pin. Therefore, this would make the analysis non-linear, increase the computational time and decrease the solution accuracy. In this study, to simulate the rigid pin, a reference node (i.e., constraint control point) was created right in the middle of the clevis arms of the bracket. Then, depending on the load conditions, a kinematic and continuum distributing coupling constraint were defined between the reference node and the clevis arm hole surfaces. In this method, applied forces or moments on the reference node are distributed onto specified group of node or surfaces as can be seen in Figure 3. Since a typical application of coupling constraints stated as defining a rigid body motion of a group of nodes with a single reference node, aforementioned method were employed in this study (Saleh Alghamdi et al., 2021). In Figure 4, a typical application of coupling constraints for the twisting motion can be seen.

By employing coupling constraint, since the analysis remained linear, not only the solution accuracy increased, but also the computational time decreased significantly.

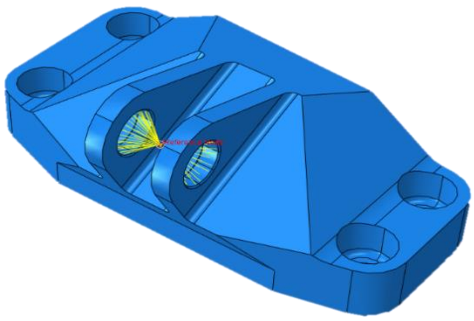

Figure 3. Reference node and coupling constraints

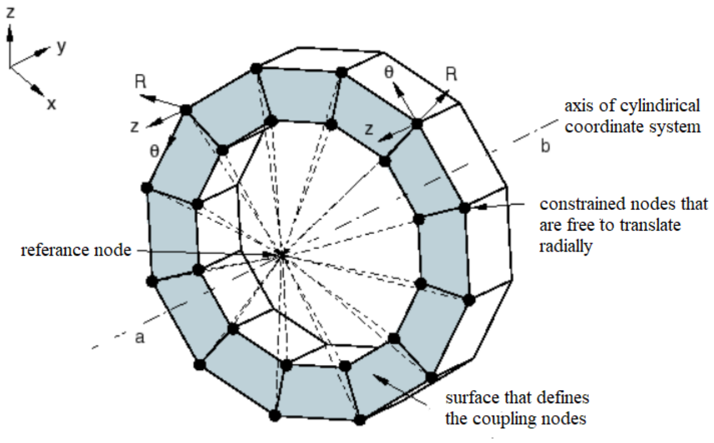

Figure 4. An example of coupling constraint (Abaques documentation, 2023)

The most important difference of this study from similar studies is that no pin-bolt model is used. Accordingly, instead of using bolts and pins, it was decided to take their effects into consideration. Thus, instead of nonlinear contact modeling, a linear analysis technique was chosen, avoiding any contact definition between geometries.

After defining a reference node and coupling constraints to simulate rigid pin, given load conditions in Figure 2 were applied to the reference node individually. To simulate given load conditions, four separate static analysis were performed.

It was stated that the bracket was mounted to the structure with 4 rigid fasteners. Therefore, for all load conditions, the all six Degrees Of Freedoms (DOF) of fastener surfaces of the bracket were fixed by encastred boundary conditions as illustrated in Figure 5.

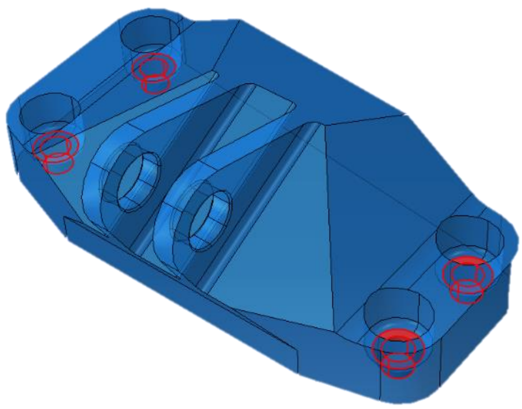

Figure 5. Encastred constraint surfaces

Then, given loads and moment in Figure 2 were applied to the reference node as following. While applying loads and the moment, depending on the load condition, all DOF of the reference node were constrained except for the load or moment direction to make sure that loads or the moment applied through only the given direction. To simulate given vertical static load, 35586 N concentrated force was applied in the positive z-direction for the 1st Load Condition. All DOF of the reference node were constrained except for z-direction. To simulate given horizontal static load, 37810 N concentrated force was applied in the negative y-direction for the 2nd Load Condition. All DOF of the reference node were constrained except for y-direction. To simulate given angled static load, 31404 N concentrated force was implemented in the negative y-direction and 28276 N concentrated force was entered in the positive z-direction for the 3rd Load Condition. All DOF of the reference node were constrained except for y-direction and z-direction. To simulate given static moment, 564924 Nmm concentrated moment was applied in the negative z-direction for the 4th Load Condition. All DOF of the reference node were constrained except for translational and rotational z-direction.

Since the bracket geometry is complex enough, the tetrahedral elements were used instead of the hexahedral elements in this analysis. Although the hexahedral element demands less computational time than the tetrahedral element, it can be used for more simple geometries. Also, although the quadratic geometric order lasts longer, rather than the linear geometric order, quadratic geometric order was preferred, since it is stated in the literature that quadratic geometric order is more convenient than linear geometric order for the topology optimization (Brighenti et al., 2021). Therefore, a C3D10: A 10-node quadratic tetrahedron element type in ABAQUS was employed for the whole analysis cases. The defined finite element mesh is illustrated in Figure 6. C3D10 mesh type is used for general purposes and is very suitable for use in complex geometries. The mesh type consists of 4 integration points. This mesh type is typically preferred for linear problem solving that do not contain any contact scenarios. The results can rapidly take. The recommended mesh size (4 mm) was determined by the help of the auto-size function of Abaqus. The curvature regions were even more concentrated meshed. Minimum 2 elements were used through thickness, to control the structure precisely. Maximum curvature deviation was 0.03. So, 26 elements were assigned to the perimeters of each circle around fasteners. Finally, 156696 nodes and 100405 elements were created. At the end of the study, mesh size was reduced and element number was increased, there was no change at the stress result values bigger than 10%, so the mesh criteria was verified.

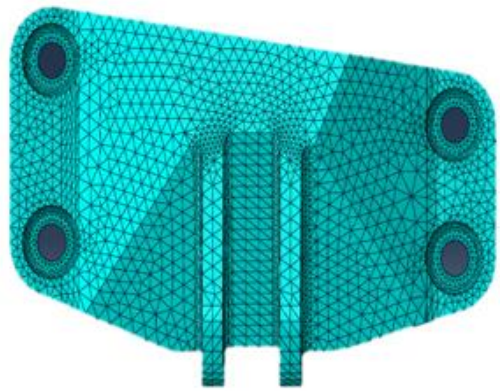

Figure 6. Finite element mesh structure

2.2. Topology optimization

After performing an elastic statical analysis, topology optimization has been done to the bracket geometry. All load cases and boundary conditions were considered as a whole scenario. Load cases were thought to be applied to the bracket individually. The bracket had to withstand all 4 loading scenarios separately. Abaqus benefits from TOSCA module, while performing topology optimization. TOSCA is an integrated topology optimization solution of Abaqus, Dassault systems. Both statical analysis and topology optimization were so could be performed in the same CAE. (Computer Aided Engineering) environment. 6 categories of items should be considered while performing a topology optimization.

The 1st one is defining an optimization task. The load appliance zones and the boundary conditions should have been frozen as design, so they shouldn't have been effected from topological volumetric changes. Frozen areas could be seen as red marked in Figure 7.

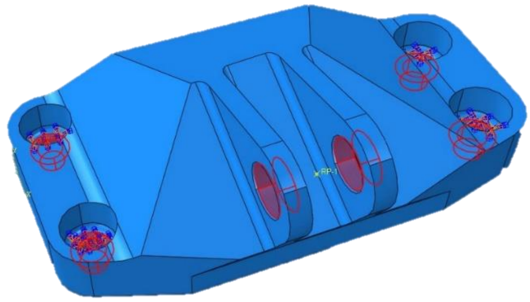

Figure 7. Boundary conditions and loading areas (as red marked)

The material update technic was determined to be as normal instead of conservative or aggressive types. TOSCA provided to choose either the general algorithm or condition-based algorithm for the solution. The general algorithm (sensivity-based) method was chosen, because it was more robust.

The 2nd item to be considered is the definition of the design responses. Strain energy and volume were selected as the case's design responses. The cumulative strain energy was taken into account because of the presence of 4 load cases.

The objective function is the 3rd item to be considered. The objective function is to minimize the strain energy. So, to get the minimum strain energy on the structure, the stiffness will be reevaluated on the bracket. The 4th item is to define the constraints. It was aimed to reduce the bracket volume by 80%.

The 5th item is to define geometrical restriction. The minimum feature size of the whole bracket (as wall thickness) was defined as 1.27 mm. So unsensible geometric results (very thin walls, unproductable features etc.) were avoided. Local stop conditions are just valid for shape optimization issues, so no stop condition was applied. The topology optimization parameters are summarized in Table 2 below.

Table 2. The topology optimization parameters for the study

Area of definition	Parameters
Optimization algorithm	General, sensitivity-based
Target volume reduction	20% of the original volume
Frozen regions	Clevis and bolt interface regions with pin and bolts
Stopping criterion	None limited to 25 iterations
Min. Wall thickness	1.27 mm

3. Results and discussion

All four static analysis results are shown in Figure 8 as Von Mises stress distribution. It can be understood from the results that the majority of the bracket structure remained undeformed for all load conditions and the maximum Von Mises stresses were calculated in the adjacent of either the rigid pin contact surfaces or the four fastener contact surfaces. Therefore, it can be concluded that the case study with the load conditions is very much convenient for topology optimization since the bracket has unnecessary and unused volume and mass for the load conditions. The maximum Von Mises stresses are calculated as 982.1 MPa, 585.4 MPa, 842.8 MPa, 314.2 MPa for the load condition 1, 2, 3, and 4, individually.

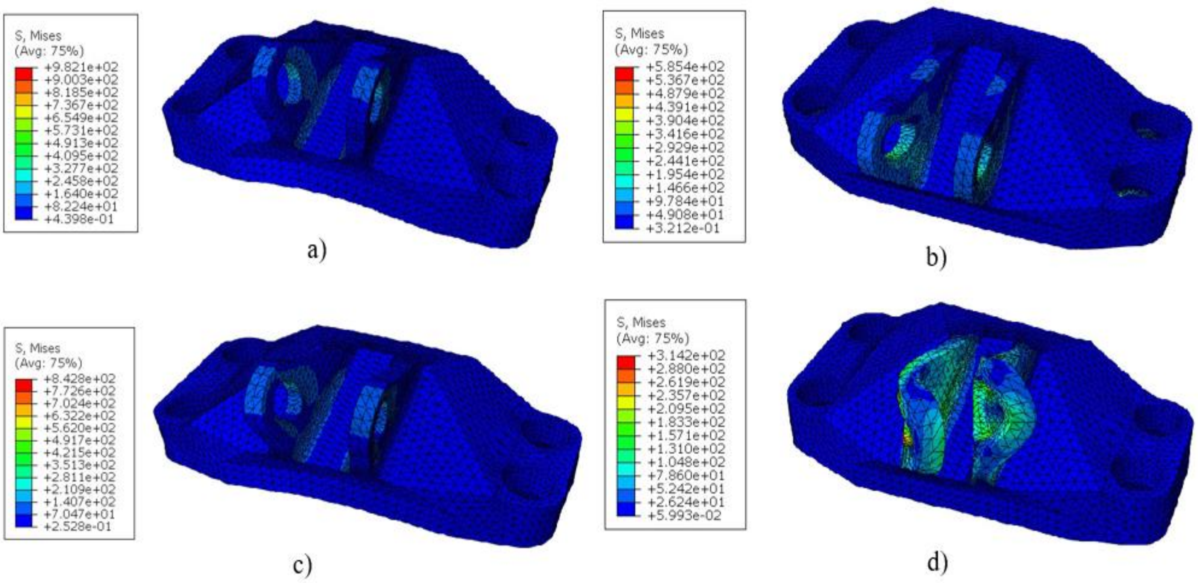

Figure 8. Von Mises stresses of Load Condition-1(a), Load Condition -2 (b), Load Condition-3 (c), Load Condition-4 (d) in the bracket

Totally 25 cycles of topological attempts were applied to the geometry to make a stable logical topological optimization. The number was determined empiric. In the Figure 9, the unsmoothed mesh result geometries can be examined.

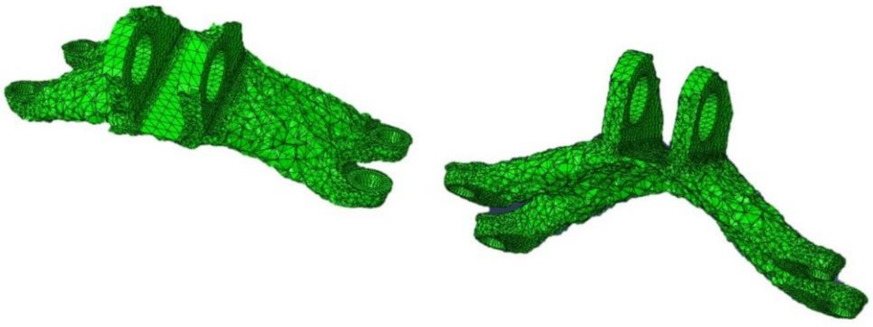

Figure 9. Topologically optimized shape, after 25 design cycles

The optimized structure should withstand 4 loading conditions within the linear elastic limits. These statical calculations are performed while having run cycles for the optimization. So, for the 4 loading cases, the final structure's stress distributions are shown below (Figure 10-13).

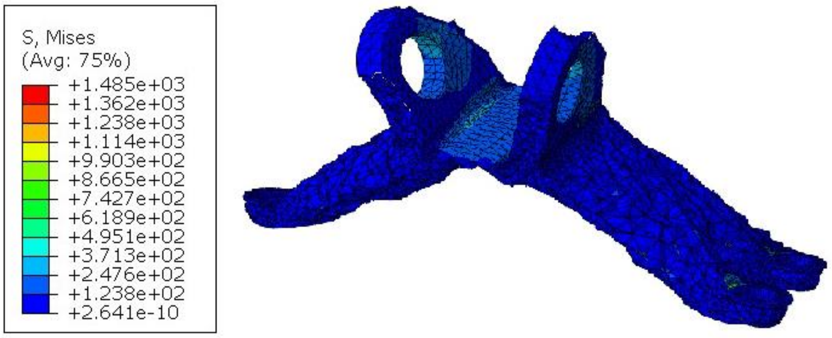

Figure 10. Optimized structure under Load Condition-1

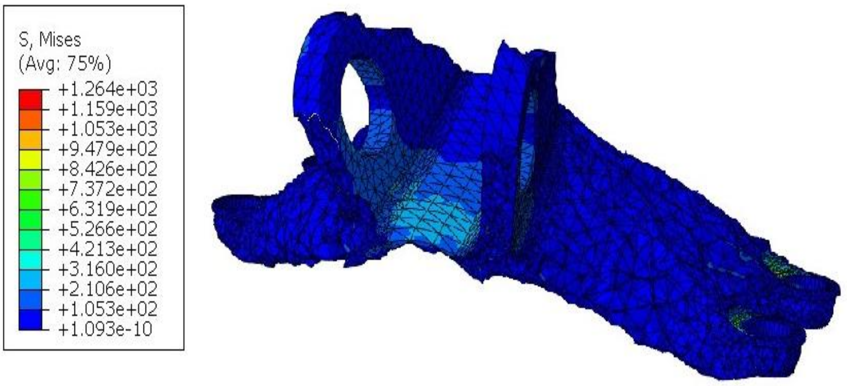

Figure 11. Optimized structure under Load Condition-2

As can be seen from the results, optimized structure can withstand all 4 loading conditions. The red stress concentrations are peak stresses. They do not exceed to the 1st neighbour cells of their regions, so they aren't taken into account. The other colour values in the legend are under the yield strength of the material. A smoothing operation is then required for stable manufacturing without any sharp edges. Programs like Ansys Space Claim, Abaqus, Blender can be used for stl (stereo lithography) data's smoothing operation. Abaqus extract option was chosen for the initial smoothing operation (Figure 14). More refinements can be applied.

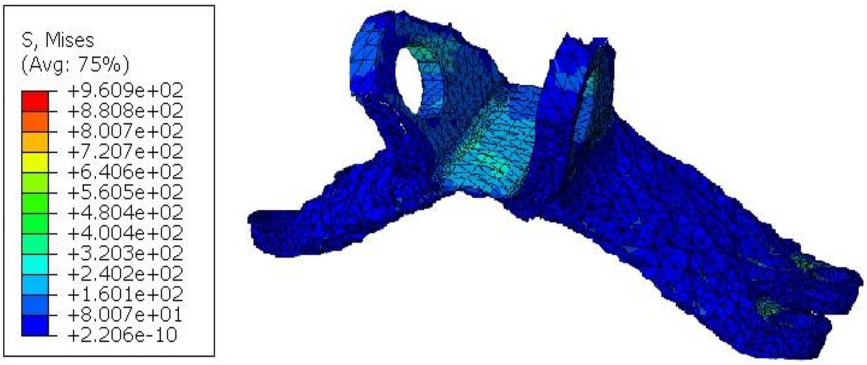

Figure 12. Optimized structure under Load Condition-3

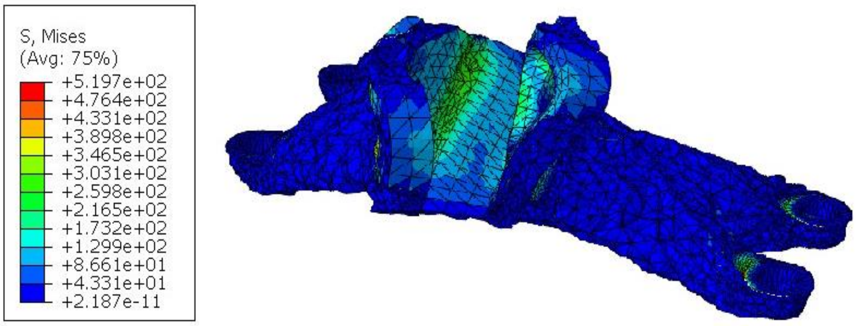

Figure 13. Optimized structure under Load Condition-4

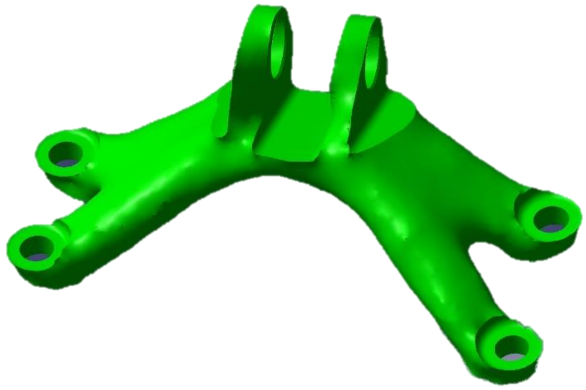

Figure 14. Smoothed bracket structure

The resulting geometry weights 0.41 kg while the initial geometry was 2.06 kg. Final result geometry can then be manufactured with Selective Laser Melting (SLM) additive manufacturing method. SLM technology arose in 1995 at the Fraunhofer Institute ILT in Germany. The ASTM International F42 standards committee has investigated this method in selective laser sintering category. But the method, actually melts the spherical metal powder and doesn't sinter it. SLM uses laser light source to melt the metal powder to manufacture parts in the concept of additive. An insert gas is used to stabilize the environment. With the help of optics, the computer can easily program the laser head to create the part. This method allows to create very complex shapes. It is suitable for both end-use parts and prototype parts. The result shows durable and dense structures that are robust in usage.

By using an.stl extension file, the geometry is sliced into sections. Parameters and process values, support geometries are then prepared with the help of a software programme. Different types of software programmes are available like Eiger from Markforged (2023). The laser beam is directed in the X and Y directions with two high frequency scanning mirrors and remains in focus along the layer utilising an F-Theta lens arrangement. Layers are usually 30-60 µm thick. Topologically optimized shapes generally cannot be produced by conventional manufacturing techniques like NC machining or casting. Additive manufacturing is the appropriate method for optimized parts with its flexible nature. Additively manufactured parts have different strength values than the casted or machined parts because of the grain properties. If an electron beam was used instead of laser, it would then be EBM, electron beam melting (Meinery et al., 1998; Nematollahi et al., 2019; Hopkins, 2021; DMLS vs SLM, 2023).

4. Conclusion

In this work, topology analysis of a bracket used as a fixing part in aircarft engine was carried out using ABAQUS software. The analysis has been implemented on a bracket with pre-defined load cases and bracket geometry.

By the end of the study, it was shown that 80% lighter weight can be achieved in the bracket structure compared to the original design. According to the weight calculations, the weight of the original bracket design was 2.06 kg, but as a result of optimization, it became 0.41 kg. The bracket has therefore been lightened considerably.

It is often not possible to manufacture the structures obtained as a result of the topology analysis with classical manufacturing methods. Additive manufacturing methods are the most suitable methods at this point. Selective Laser Melting (SLM) method can be used in the production of the geometry obtained in this study. In the future, more comprehensive studies can be conducted in which the structural parts for which topology analysis has been performed are subjected to mechanical tests after their production with the SLM method.

Author contribution

The authors equally contributed to the article.

Declaration of ethical code

The authors of this article declare that the materials and methods used in this study do not require ethics committee approval and/or legal-special permission.

Conflicts of interest

There is no conflict of interest between the authors

References

Abaques documentation. (2023, February 1). https://classes.engineering.wustl.edu/2009/spring/mase5513/abaqus/docs/v6.6/books/usb/default.htm?startat=pt08ch28s03aus108.html

Attaran, M. (2017). The rise of 3-D printing: The advantages of additive manufacturing over traditional manufacturing. *Business Horizons*, *60*, 677-688. https://doi.org/10.1016/j.bushor.2017.05.011

ASM Aero space specification metals inc. Titanium Ti-6Al-4V Grade 5 (2023, April 8) https://asm.matweb.com/search/SpecificMaterial.asp?bassnum=mtp641.

Bendsoe, M. P., & Kikuchi, N. (1988). General optimal topologies in structural design using a homogenization method. *Computer methods in applied mechanics and engineering*, *93*, 291-318.

Brighenti, R., Cosma, M. P., Marsavina, L., Spagnoli, A., & Terzano, M. (2021). Laser-based additively manufactured polymers: a review on processes and mechanical models. *Journal of Materials Science*, *56*, 961-998. https://doi.org/10.1007/s10853-020-05254-6

DMLS vs SLM 3D Printing for Metal Manufacturing Selective Lazer Melting (2023, February 11) https://www.element.com/nucleus/2016/dmls-vs-slm-3d-printing-for-metal-manufacturing.

Grabcad community GE jet engine bracket challenge (2022, December 8). https://grabcad.com/challenges/ge-jet-engine-bracket challenge.

Hopkins, D. (2021). The critical of ultrasound by advancing additive for industrial. *Quality*, *60*(10), 38-38.

Kahraman, F., & Küçük, M. (2020). Otomotiv endüstrisinde topoloji optimizasyonu ile ağırlık azaltma uygulaması üzerine bir araştırma. *Avrupa Bilim ve Teknoloji Dergisi*, (20), 623-631. https://doi.org/10.31590/ejosat.789424

Koçar, O. (2018). Dorse tasarımında stres dağılım analizi ve topoloji optimizasyonu. *Karaelmas Fen ve Mühendislik Dergisi*, *8*(1), 309-316. https://doi.org/10.7212%2Fzkufbd.v8i1.1073

Markforged 3D Printers Metals/Carbon Fiber/Plastics (2023, April 12) https://markforged.com/software.

Meiners, W., Wissenbach, K., & Gasser, A. (1998). Shaped body especially prototype or replacement part production. DE Patent, 19.

Nematollahi, M., Jahadakbar, A., Mahtabi, M. J., & Elahinia, M. (2019). Additive manufacturing (AM). In Metals for biomedical devices (pp. 331-353). Woodhead Publishing.

Rossow, M. P., & Taylor, J. E. (1973). A finite element method for the optimal design of variable thickness sheets. *Aiaa Journal*, *11*(11), 1566-1569.

Saleh Alghamdi, S., John, S., Roy Choudhury, N., & Dutta, N. K. (2021). Additive Manufacturing of Polymer Materials: Progress, Promise and Challenges. *Polymers, 13*(5), 753. https://doi.org/10.3390/polym13050753

Top, N., Gökçe, H., & Şahin, İ. (2019). Eklemeli Imalat Için Topoloji Optimizasyonu: El Freni Mekanizmasi Uygulamasi. *Selçuk-Teknik Dergisi*, *18*(1), 1-13.

Topaç, M. M., Bahar, E., Kaplan, A., & Sarıkaya, E. Z. (2017). Topoloji optimizasyonu yardımıyla, askeri taşıt bağımsız ön süspansiyonu için alt salıncak tasarımı. IDEFIS, 2017, 2nd.