International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 09 | Sep 2021 www.irjet.net p-ISSN: 2395-0072

Topology Optimization of Pivot Bracket With Rigid Bearing

Nitesh Maruti Bade[1], Dr. N. A Rawabawale[2]

[1]PG Student, Department of Mechanical Engineering, MBES College of Engineering, Ambajogai, Maharashtra, India
[2]Head and Associate professor, Department of Mechanical Engineering, MBES College of Engineering, Ambajogai, Maharashtra, India

---***---

Abstract - *This paper deals with the topology optimization of the pivot bracket with rigid bearing used in pneumatic cylinder. The focus of this study is to reduce the weight of the bracket without compromising the structural characteristics. A small reduction in the weight of each bracket will have a great impact on the overall weight of the pneumatic system. Solid thinking Inspire is used for static analysis and topology optimization. The results of the von-mises stress distribution and the displacement of the actual model and the optimized were compared. With the optimized model of pivot bracket, 58.49% weight reduction is achieved to maintain the same structural behavior.*

Key Words: Mounting bracket, Design- optimization, Static analysis, Weight reduction etc.

1. INTRODUCTION

Topology optimization is a mathematical approach for optimizing parts and structures. Introduced in the late 1980s it is today used in the automotive-, aerospace-, marine- and biomedicine industry. It works by optimizing the material distribution in a given design space with a given set of loads, boundary conditions and constraints with the final goal of maximizing the part strength while minimizing weight. (Gould, 2015) It has gained popularity in the past years due to increased availability in Computer Aided Design- software being both easy to use and affordable. The topology optimized designs have efficient material use and can be optimized for manufacturing techniques such as casting and 3D Printing. The resulting designs often have biological-like shapes resembling shapes found in nature such as skeletal, honeycomb and porous shapes containing curvy surfaces with bends, twists and voids. (Gould, 2015) The topology optimization process is linked to and follows the creation of a part which is done using a Computer Aided Design (CAD) -process.

1.1 Objectives

In this paper we are going to optimizing the pivot bracket with rigid bearing of pneumatic system.

Demonstrate the topology optimization- process,

Topology optimize a mounting bracket using solid Thinking Inspire and SolidWorks,

Compare the weight reduction and difference in stress to the original mounting bracket,

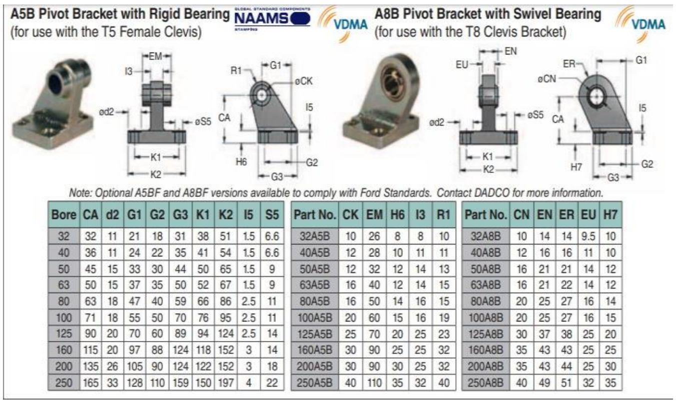
Compare the geometrical differences between each bracket.

2. METHODOLOGY

A standard mounting bracket chosen to demonstrate the topology optimization process. The software used were Solidthinking Inspire 2018.2 and SolidWorks 2019.
Also I considered standard aluminum alloy material, with properties shown as follow:

Material name: aluminum 2024-T3/T6/T8
Young's modulus (E): 75e+03 Mpa
Poisson's ratio (Nu): 0.330
Density: 2.770e-06 kg/mm^3
Yield stress: 275.80e+00 Mpa
Coefficient of thermal expansion: 22.8e-06/ K

In this case I made this model using solidworks 2019 with the reference of standard dimensions of pivot bracket with rigid bearing, bore size 250, part number 250A8B this is ford standard part of DADCO company details as shown in below fig -1.



Bore	CA	d2	G1	G2	G3	K1	K2	I5	S5	Part No.	CK	EM	H6	I3	R1	Part No.	CN	EN	ER	EU	H7
32	32	11	21	18	31	38	51	1.5	6.6	32A5B	10	26	8	8	10	32A8B	10	14	14	9.5	10
40	36	11	24	22	35	41	54	1.5	6.6	40A5B	12	28	10	11	11	40A8B	12	16	16	11	10
50	45	15	33	30	44	50	65	1.5	9	50A5B	12	32	12	14	13	50A8B	16	21	21	14	12
63	50	15	37	35	50	52	67	1.5	9	63A5B	16	40	12	14	15	63A8B	16	21	22	14	12
80	63	18	47	40	59	66	86	2.5	11	80A5B	16	50	14	16	15	80A8B	20	25	27	16	14
100	71	18	55	50	70	76	95	2.5	11	100A5B	20	60	15	16	19	100A8B	20	25	27	16	15
125	90	20	70	60	89	94	124	2.5	14	125A5B	25	70	20	25	23	125A8B	30	37	38	25	20
160	115	20	97	88	124	118	152	3	14	160A5B	30	90	25	25	32	160A8B	35	43	43	25	25
200	135	26	105	90	124	122	152	3	18	200A5B	30	90	30	25	32	200A8B	35	43	44	25	30
250	165	33	128	110	159	150	197	4	22	250A5B	40	110	35	32	40	250A8B	40	49	51	32	35

Fig -1: Standard DADCO product specs

In solidworks software, by using commands like sketch and extrude, extrude cut, plane etc. I made the 3d model of A5B pivot bracket with rigid bearing as shown in below fig -2.

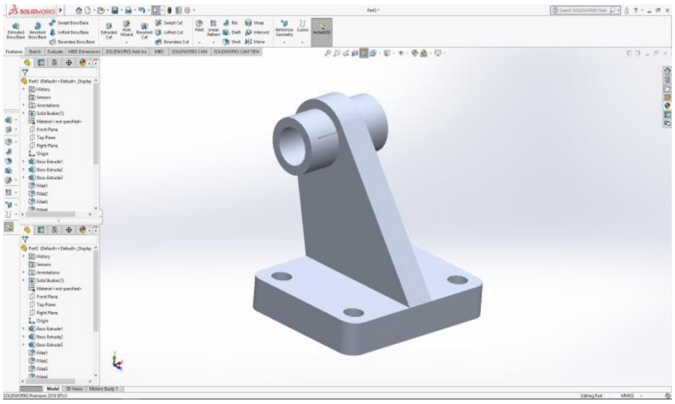


Fig -2: 3D model of pivot bracket with rigid bearing

2.1 Topology optimizing Mounting bracket using Inspire 2018

After modeling in solidwork, I saved it in iges. Format for easily importing it into solidthinking inspire 2018

And then process started for optimizing it with keeping maximum stiffness ration and simultaneously reducing its weight.

2.2 making partition to differentiate design space

Fig -3: partition tool used

After importing model in solidthinking we must have make it ready for optimization, by making partitions for all holes where fasteners are going to attach.

2.3 Creating Design- and Non-design spaces

Solid Thinking Inspire would not recognize the four mounting holes as well as the one large hole for the pivot pin without defining the design space.

As shown in below Fig -4 red selected area is design space and grey color is parted space.

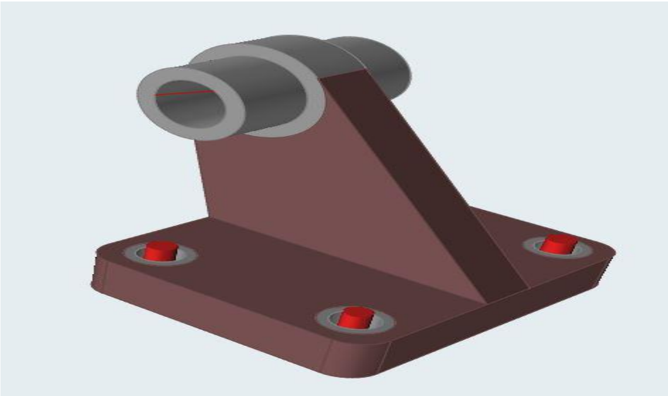


Fig -4: design space selection

2.4 Applying forces and supports

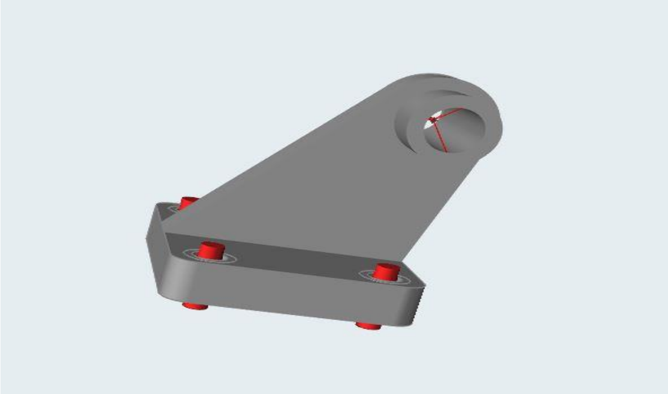


Fig -5: support and forces applied

After designing the space defined we have to apply supports to the four holes where it will get fixed

And also we have to apply load to the pin hole of 500 KN for getting maximum factor of safety and maximum stiffness ratio I assumed this much of load.

After FEA analysis done we have to go for topology study.

2.4 Topology optimization:

Now the model is ready for topology optimization process the bracket was now ready to be optimized. The settings for optimization were found under the Optimize- tool. In there the type of optimization, percentage of mass to be reduced while retaining part stiffness, minimum and maximum element sizes was assigned etc.

The settings used for optimization are as shown in below fig -6.

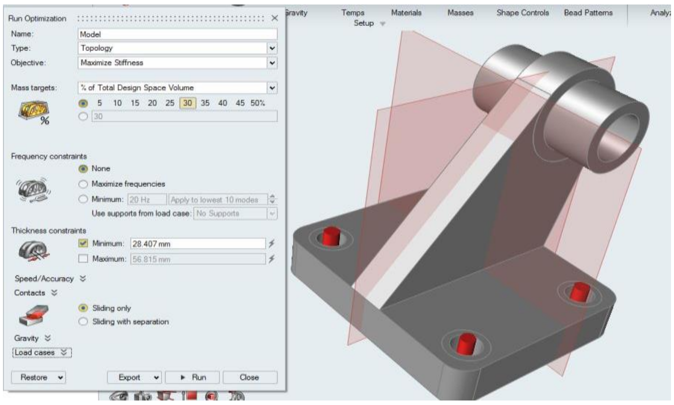


Fig -6: topology optimization setting

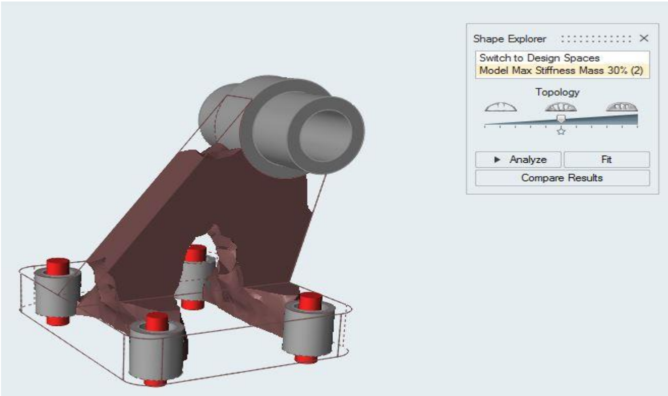


Fig -7: topology optimized model

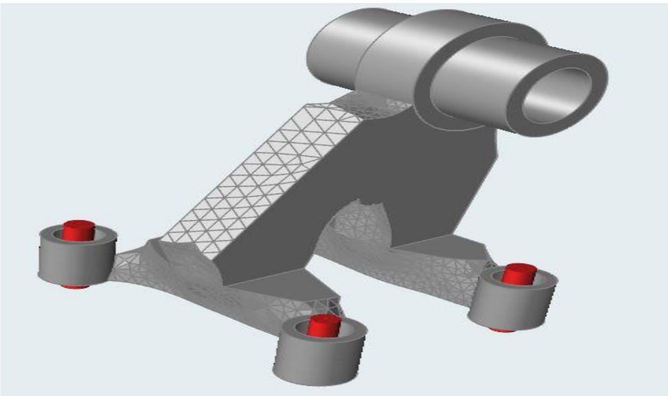


Fig -8: smooth fit model

As shown in fig -7 we can see the optimized model with some rough surface and after commanding it smooth fit we can see the smooth fit model in fig -8

But this model needs to be remodel, for that, we have a great tool called polynurb tool in solidthinking inspire.

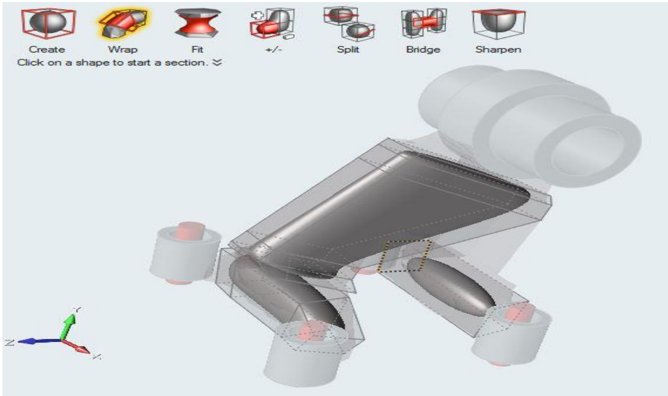


Fig -9: polynurb tool modeling

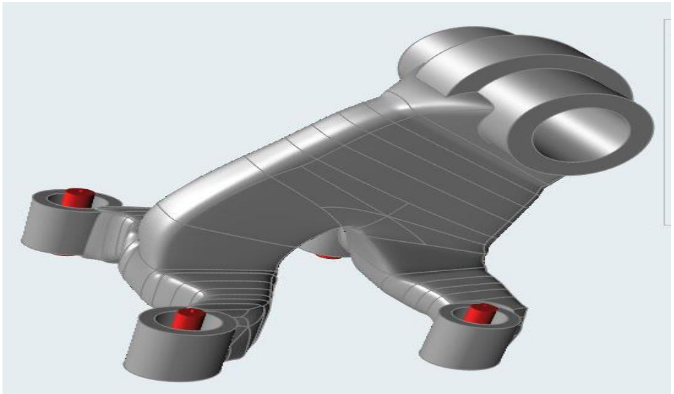


Fig -10: topology optimized model

The above figure 10 show that the model is now ready for analysis and comparison with existing model to conclude the objective of the paper as disscused below.

3. FEA ANALYSIS AND THEIR RESULTS

After modeling we analyzed the existing model with same load of 500 kn and same material as of aluminum 2024 T3 with bracket weight of 4.77 kg shown by software.

for better comparison and also better understanding topology optimization process and its generated design study.

As below we got some FEA results of existing model of pivot bracket in solidthinking inspire 2018:

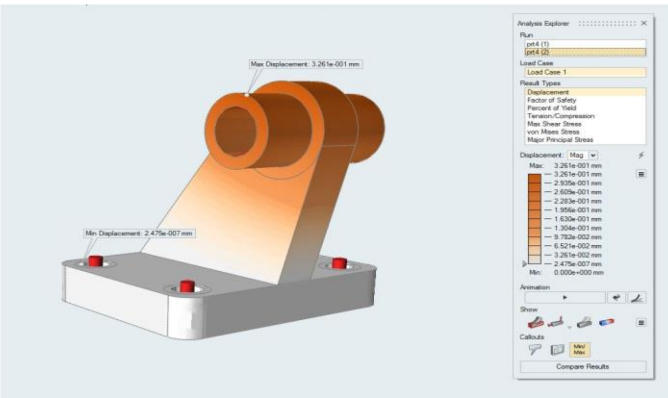


Fig -11: displacement result of existing model

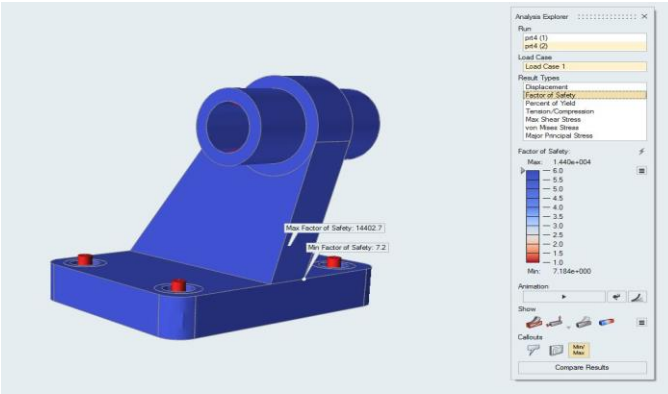


Fig -12: factor of safety of existing model

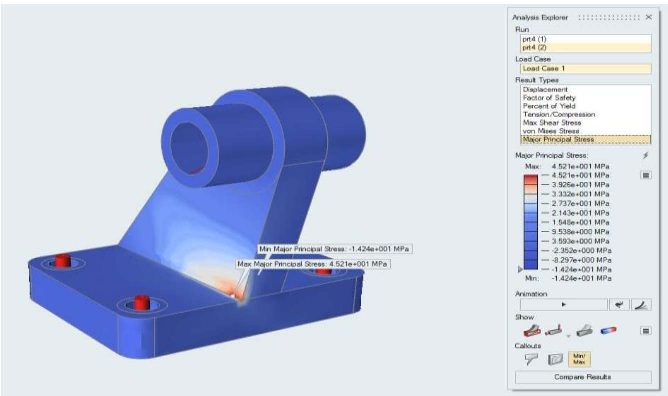


Fig -13: major principal stresses of existing model

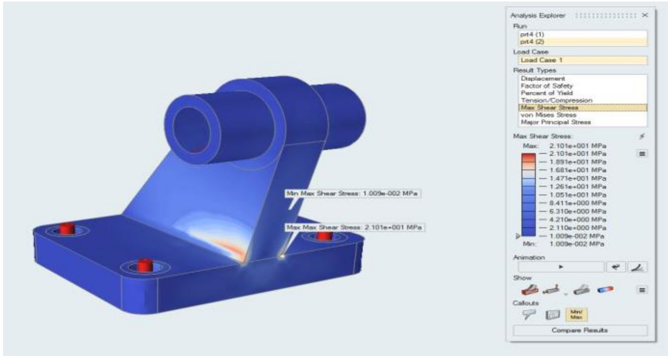


Fig -14: max. Shear stress of existing model

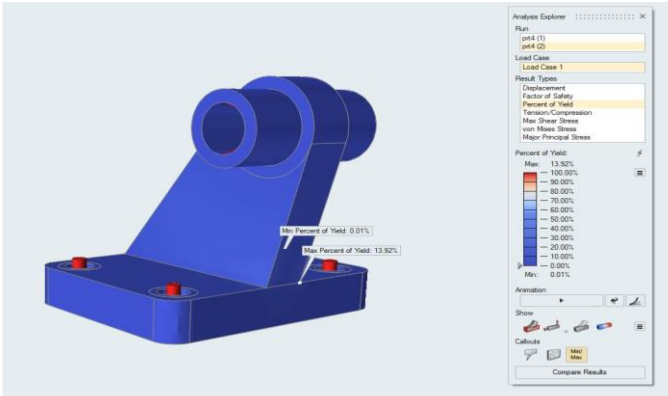


Fig -15: yield percentage of existing model

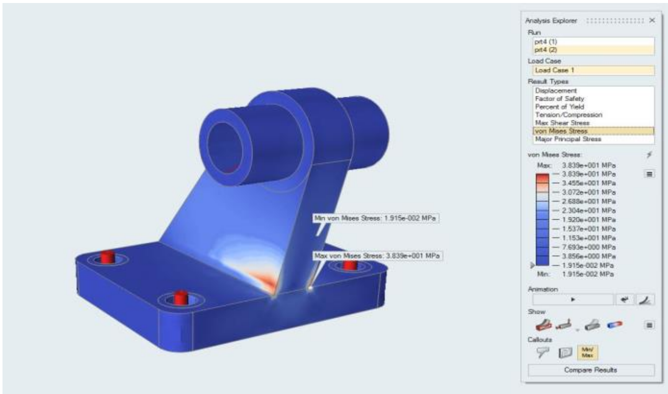


Fig -16: von-mises stresses of existing model

The above figure shows the different stresses and displacement result of FEA analysis of existing model of pivot bracket with rigid bearing.

Table -1: existing model results

name of result	Value		Unit
	Max	Min	
Displacement	3.261e-001	2.475e-007	mm
Factor of safety	1.440e+004	7.2	
Percent of yield	13.92%	0.01%	Percent-age
Tension/Compression	3.676e+001	-3.839e+001	Mpa
Max shear stress	2.101e+001	1.009e-002	Mpa
Von-mises stress	3.839e+001	1.915e-002	Mpa
Major principle stress	4.521e+001	-1.424e+001	Mpa

The above table shows that different results of FEA analysis like its displacement and also the factor of safety as 7.1 with max. Displacement of 0.3261 mm.

Like this results below are some of topological optimized result of redesigned model with same material as aluminum 2024-T3 and default mesh element size like above result with weight 2.79 kg which is 58% reduced from existing model:

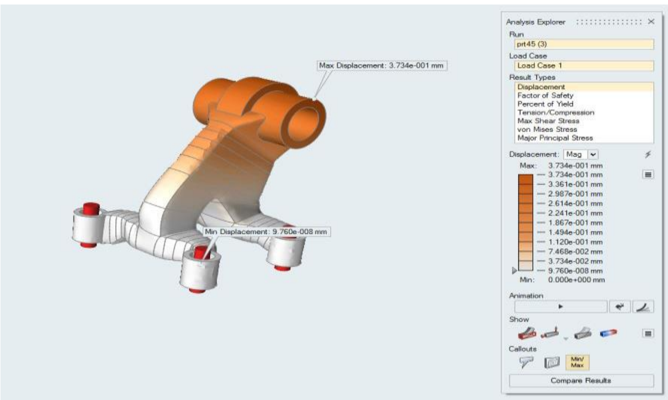


Fig -17: displacement of optimized model

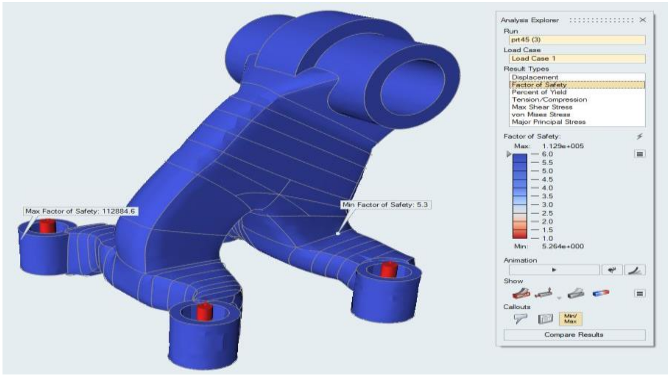


Fig –18: factor of safety of optimized model

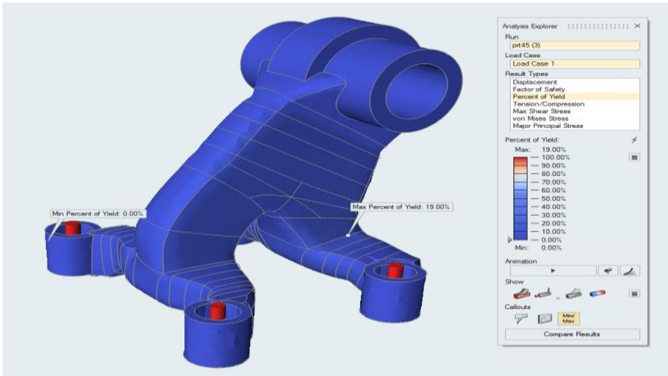


Fig -19: yield percentage of optimized model

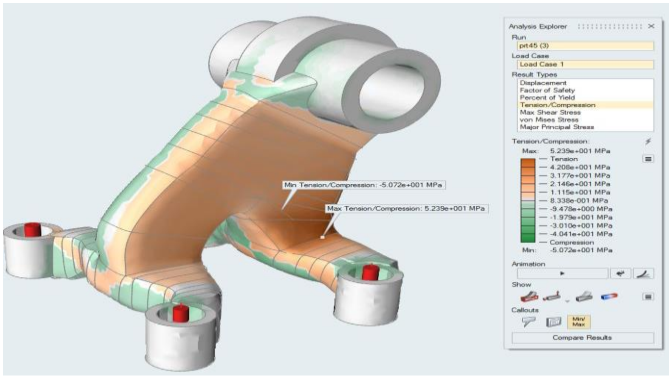


Fig -20: tension/compression of optimized model

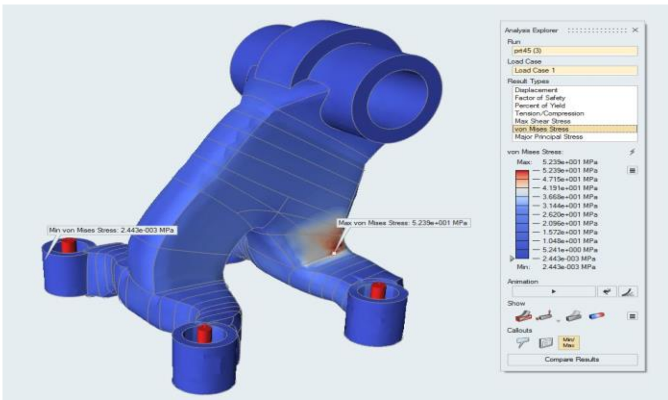


Fig -21: von-mises stress of optimized model

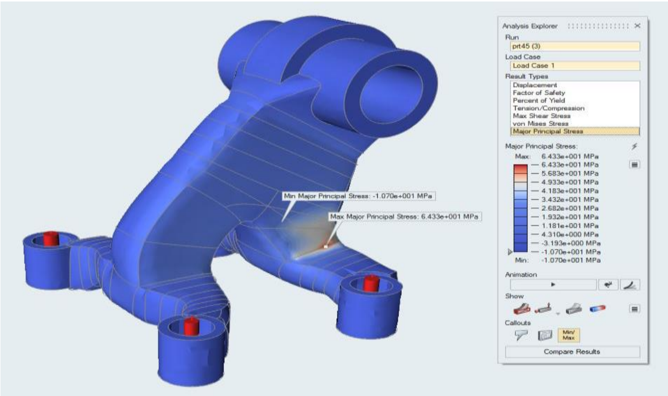


Fig -22: major principle stresses of optimized model

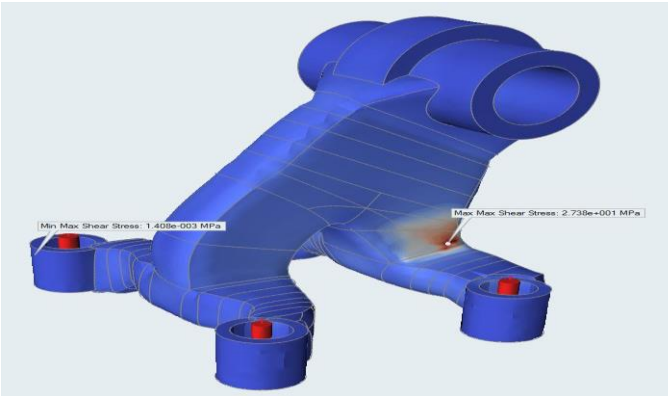


Fig -23: max shear stress of optimized model

Table -2: FEA results of topology optimized model

Name Of Result	Value		Unit
	Max	Min	
Displacement	3.734e-001	9.760e-008	mm
Factor of safety	1.129e+005	5.3	
Percent of yield	19%	0%	Percent -age
Tension/Comp ression	5.239e+001	-5.072e+001	Mpa
Max shear stress	2.738e+001	1.408e-003	Mpa
Von-mises stress	5.239e+001	2.443e-003	Mpa
Major principle stress	6.433e+001	1.070e+001	Mpa

Above figures and tables show the topology optimized model results with weight of 2.79 kg with displacement of 0.3734 mm and factor of safety of 5.3.

4. COMPARSION

Above result shows FEA results and change in geometries and their analysis values. Now we can more understand this results by comparing existing model with topological optimized model.

The full comparison as explained below in the tables:

Table -3: comparison of weight

Existing model weight and material		Optimized model weight and material	Material saved (%)
Material	Aluminum	Aluminum	58.49%
Weight	4.77 Kg	2.79 kg	

The above table shows weight reduction after optimization and with same material.

By this topology optimization method we have saved 58.49% weight from existing model without compromising the performance and safety of the bracket.

The below Table -4 shows Existing models displacement is 0.3261 mm, and optimized models displacement is 0.3734 mm which will not affect on the performance of the bracket.

Table -4: Displacement Result

Name Of Result	Value		Unit
	Max	Min	
Displacement of Existing Model	0.3261	0.00000 02475	Mm
Displacement of optimized model	0.3734	0.00000 009760	Mm

The below Table -4 shows FOS of optimized model is 5.3 and existing one is 7.2

Table -4: Factor of safety

Name Of Result	Value	
	Max	Min
Factor of safety of existing Model	1.440e+004	7.2
Factor of safety of optimized model	1.129e+005	5.3

Table -5: Yield percentage

Name Of Result	Value		Unit
	Max	Min	
Yield percentage of existing model	13.92%	0.01%	Perce-ntage
Yield Percentage of optimized model	19%	0%	Perce-ntage

Table -6: Tension compression comparison

Name Of Result	Value		UNIT
	Max	Min	
Tension/compression of existing model	36.76	-38.39	Mpa
Tension/compression of optimized model	52.39	-50.72	Mpa

Table -7: Max shear stress comparison

Name Of Result	Value		UNIT
	Max	Min	
Max shear stress of existing model	21.01	0.01009	Mpa
Max shear stress of optimized model	27.38	0.001408	Mpa

4.7 Von mises stress distribution

Table -8: comparison of von-mises stress distribution

Name Of Result	Value		Unit
	Max	Min	
Von-mises stress of existing model	38.39	0.01915	Mpa
Von-mises stress of optimized model	52.39	0.002443	Mpa

Table -9: comparison of Major principle stress

Name Of Result	Value		Unit
	Max	Min	
Major principle stress of existing model	45.21	14.24	Mpa
Major principle stress of optimized model	64.33	10.70	Mpa

The above table shows the comparison between existing model and topologically optimized model.

As we can see there is no extra change or reduction in performance of the pivot bracket.

With less weight its cost and performance is got improved.

5. CONCLUSIONS

Topology optimization is a powerful tool in reducing part weight while retaining stiffness.

It enables a seamless and efficient optimization for design engineers to improve an existing design. When used in great numbers, the combined weight reduction leads to cost

Savings in operation such as in the airline industry. Not only does the topology optimization reduce weight but also parts no longer requiring welding seams further reduces the Weight.

Topology optimized versions of a standard mounting bracket were produced using solidthinking Inspire. The successful steps included importing a CAD model, performing a FEA-analysis, defining optimization method and part design- and Non-design space, running topology optimization and remodeling of bracket by referring the generated design by topology.

The specified optimization type used in optimization case was to reduce weight, while maximizing part stiffness.

After all this we have successfully reduced weight by 1.98 kg from 4.77 kg of aluminum bracket that's the 58.49% of weight reduction achieved.

6. FUTURESCOPE

Again we can do study and optimize it by using lattice structure analysis using same software as of used by this project.

And also can have some improvement in design and modelling with other loading conditions and also by lowering FOS.

REFERENCES

[1] Koushik. S. (2013), Static and Vibration Analysis of Engine Mounting Bracket of TMX 20-2 using OptiStruct, Altair Technology Conference 2013, India

[2] P. D. Jadhav, Ramakrishna(2014), Finite Element Analysis of Engine Mount Bracket, International journal of advancement in engineering technology, 1, 2349-3224 V. Doundkar, D. Ghatage, M. Madkaikar,

[3] A. Kulkarni, (2011), Topology Optimization for Engine Mounting Arm with Fatigue, HTC 2011

[4] Y.S. Kong, S. Abdullah, M.Z. Omar, S.M. Haris, "Topological and Topographical Optimization of Automotive Spring Lower Seat", Latin American Journal of Structures, 2016.

[5] https://www.dadco.net/catalog-en/?page=product&cid=pivot_bracket_with_rigid_bearing&id=250A5B

[6] Yugi Ogata, Satoru Suzuki, Masami Hiraoka, "Development of Topology Optimization Method for reduction of Transmission Housing Weight", SAE Technical Paper, 2005-01-1699, 2005

[7] Pan, X. Y., Zonni, D., Chai, G. Z., Zhao, Y. Q., & Jiang, C. C. (2007). Structural optimization for engine mount bracket (No. 2007-01-2419). SAE Technical Paper.

[8] https://www.dadco.net/wp-content/uploads/2013/03/hp_catalog.pdf