IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

TOPOLOGY OPTIMIZATION OF CONNECTING ROD USING ANSYS WORKBENCH 18.1

[1]K. Archana, [2]Y. Anantha Reddy, [3]P. Naveena, [4]K. Sai Anusha

[1]Asst professor, [2]Asst professor, [3]Asst professor, [4]Asst professor

Mechanical Engineering

St Martins Engineering college, Dhulapally, India

Abstract: Topology Optimization (TO) advantages are comprehend to a more extent cause to manufacturing freedom that Additive Manufacturing (AM) offer, analyzed to other conventional manufacturing methods. AM has the advantage of designing shallow and critical patterns formerly not feasible, and consequently opens up a whole new design spectrum. This paper study the capabilities of using topology optimization as a tool to find lighter and stronger patterns for the connecting rod in the si engine. By giving it more mass the connecting rod is optimized, subjecting it to load cases and pushing the topology optimization to generate the connecting rod meet the weight need without exceeding the yield strength. Result validation showing that the force/pressure loadings are the most prominent. The present topology optimization tools in ANSYS don't support optimization because features of thermal and thus optimization in the currented work has only been able to consider static structural loads. Nonetheless, this is possible to optimize the connecting rod because static structural loads and achieve a connecting rod which satisfies the weight need. However, optimization tools because thermal loading would be a desirable feature in the future.

I.INTRODUCTION

The Crank train is the heart of the reciprocating piston engine, and the purpose is to translate the linear motion of the pistons into rotary motion for the purpose of extracting useful work. This is typically composed of a connecting rod, crankshaft, and flywheel or power takeoff device. The crankshaft is usually composed of one or multiple throws, to which the connecting rod is attached with either fluid film journal bearings or rolling element bearings. In addition to its translating linear main function to rotary motion, typically the crankshaft drives many of the accessories of engine such as: valve train, system of lube, charging system, or cooling system. Crankshaft rotation direction can be either clockwise or counterclockwise, depending on driveline packaging and requirements. If the valve train mechanism is designed appropriately, in some applications it can rotate in both directions. The connecting rod in its simplest form is a beam with two pin joints at either end. As it is connected to the piston One side sees relatively low rotation speeds , as it is attached to the crankshaft while the other sees high rotation speeds. The connecting rod will commonly support the bushings or bearings for two ends of its uses. This is one of the most stressed parts in the engine and subjected to compressive, bending stress, and high tensile. It can be forged, cast. crankshaft operating temperatures and connecting rod are commonly oil operating temperature of among 80-110ºC, but peak oil temperatures can rise to 150ºC. Common ambient temperatures are from -30ºC to 50ºC. Crankpin bore Connecting rod temps are similar, but pin end of piston temperatures are larger as they are exposed to the piston the bottom side. The piston pin end ,connecting rod commonly sees 150-180ºC. The connecting and crankshaft rods are commonly in the cylinder block immersed in an oil bath. As a combustion product, sulfuric acid and water are present and the piston rings into the oil bath often work past.

II.LITERATURE REVIEW

Vikas Singh[1], Sumit Kr. Verma[2]: "Design and Analysis of Connecting Rod for Different Material Using Ansys Workbench 16.2", this study aims to carry out for the load strain, stress, total deformation and analysis of factor of safety of pin end of the connecting rod of different materials. In this work existing connecting rod material are replaced by beryllium alloy and magnesium alloy. Fea analysis was carried out by considering five material aluminium 360, forged steel, titanium alloy, ti-13v-11cr-3al, magnesium alloy, beryllium(alloy 25). In this study a solid 3d model of connecting rod was developed using solid works-2016 software and analysis was carried out by ansys 16.2 software and useful factors like stress, strain etc.

[1]**Parekh Rajesh,** [2]**Parmar Prutviraj: "**Solution of Manufacturing Defects in Connecting Rod". Therefore we have analyzed all manual machining process to get solution of these two problems. In cap tight problem the cap is very tight with rod due to this cap can't separate from connecting rod with normal force. There for induced in assembling of engine. The crank is fitted in to the big end diameter of connecting rod. They have found two problems in manufacturing of connecting rod, cap tight and oversize of big end diameter during inspection of connecting rod.

Leela Krishna Vegi[1]**, Venu Gopal Vegi**[2]**:** " Design And Analysis of Connecting Rod Using Forged steel". A parametric model of Connecting rod is modeled using CATIA V5 R19 software and to that model, analysis is carried out by using ANSYS 13.0 Software. The best combination of parameters like Von misses Stress and strain, Deformation, Factor of safety and weight reduction for two wheeler piston were done in ANSYS software. Forged steel has more factor of safety, reduce the weight, increase the stiffness and reduce the stress and stiffer than other material like carbon steel.

[1]**Ganta.Krishnarjuna Reddy** [2]**badde Naik Vadithe: "**DESIGN AND ANALYSIS OF CONNECTING ROD" As a connecting rod is rigid, it may transmit either a push or a pull and so the rod may rotate the crank through both halves of a revolution, i.e. Piston pushing and piston pulling. Earlier mechanisms, such as chains, could only pull. In a few two-stroke engines, the connecting rod is only required to push. In which it undergoes structural deformations. Thus in this project we are modeling a connecting rod in solid works 2016 design software and doing static structural analysis in ansys work bench 14.5 software.

III.DESIGN OF CONNECTING ROD

The connecting rod must be arranged for high cycle fatigue to withstand the large number of engine cycles and also for stiffness in supporting fluid film bearings. At TDC-Exhaust as illustrated on the connecting rod occur the maximum tensile forces. Since they are near zero during valve overlap gas pressure forces can be neglected.

However, a longer connecting rod has more weight and increases the engine deck height. These conflicting requirements must be balanced. Typical conrod ratios are:

$$\lambda = \frac{r}{L} \qquad \lambda \approx 0.20 - 0.35$$

Where:

r = Crank radius, or stroke/2

L = Connecting rod length, pin to pin

3.1 Crankpin of Connecting Rod Bore Cylindricity

As we progress down the connecting rod length, various pattern criteria are needed. The connecting rod crankpin end is mainly arranged for stiffness and not strength, the crankpin journal bearing and maintain roundness as the connecting rod needs to support. As the engine developments through its cycle, to stretch the rod tensile inertia forces attempt and deform the connecting rod crankpin end, as shown in **Figure 1** Due to the Poisson effect the connecting rod Stretching onward the cylinder bore causes the connecting rod to pinch in along the cap split, which may lead to the bearing contacting the crankshaft and the oil film breaking down. If rolling element bearings are used, the rolling elements pinching may occur leading to spalling or skidding.

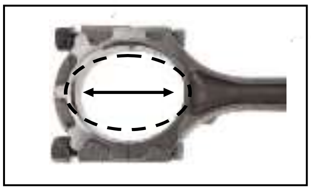


Fig 1: Crankpin bearing bore Deformed shape

3.2. Connecting Rod-to-Cap Alignment

Connecting rod-to-cap arrangement is critical to a successful rod pattern, and ensuring crankpin journal bearing life. Angular split lines present high of a challenge in relation to the split line by reason the direction of the applied forces, but are sometimes needed to aid in service the engine in the vehicle.

Some alignment methods are:

- Increased Shank Fastener.
- Ring Dowels.
- Separate Dowel Pins.
- Serration.
- Stepped surface.
- Cracked

IV.ANALYSIS OF CONNECTING ROD

4.1 Geometry

The geometry has been designed in Catia V5 and imported into ansys 18.1 in format of .igs file.

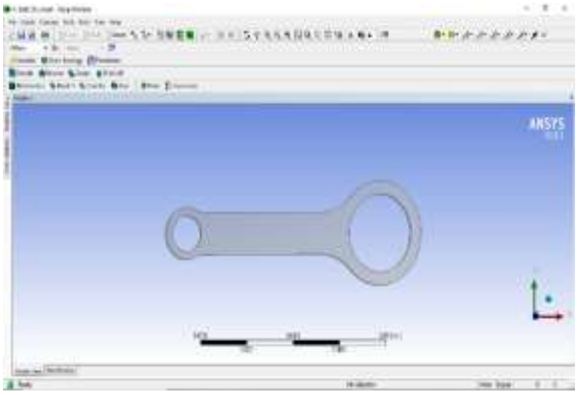


Fig. 2: connecting rod

4.2 Analysis and TopologyOptimization

In Mechanical, meshing and controlling possible contacts are important before run- ning any analysis or optimizations. The geometry is in this thesis only optimized with respect to static structural loads such as casing force and pressure due to limi- tations with the TO solver. The thermal loading is only used for design validation.Penalty factor was set to 3 in this thesis, for all the simulations and a convergence accuracy of 0,1 % for 2D model, 0,5% (as suggested accuracy) for the 3D model.

4.3 Analysis setup

Fig.3 shows the typical setup in Workbench to carry out Pre- & Post-processing, Topology Optimization and design validation. Default CAD software in ANSYS 18.1 and forward is Space Claim, a so called direct modeler. When the geometry has been created or imported, the next step is to enter ANSYS Mechanical, solid mechanics software made by ANSYS. When Mechanical is opened, it automatically reads the up-stream data being the geometry and engineering data which contains the material properties.

Fig.3. ANSYS Workbench typical setup

In Mechanical the meshing is done as well as the various analyses depending on what analysis systems are linked together in workbench. In Fig.3 three analysis systems are linked together, the first three are Steady-sate static structural and topology optimization. The latter two are coupled only to read the results from the topology optimization in order to perform a design validation. When meshing is done in Mechanical and the load cases are applied and evaluated, the topology optimization can be initiated. In the TO analysis system the objective, response constraints and manufacturing constraints are defined as well as what solver to be used, what region to be opti- mized and what region to be excluded. For example, aerodynamic surfaces are not an option to optimize. So they are kept untouched by choosing them as exclusion region. The file format of a finished TO-geometry for post-processing is STL (Stereolithog- raphy) and contains faceted data. The faceted geometry has to be post-processed in order to smooth out sharp edges and possibly correct bad geometry before carrying out the design validation.

4.4 Preparing the Topology Optimization Analysis

When preparing geometry for Topology Optimization it's usually necessary to add some wrapping (or packing) geometry depending on what areas one wants to opti- mize. In Fig.4.the geometry is given a packing volume in order to find an alter- native solution to the connecting rod. The geometry is thus given a large wrapping volume so that new load paths can be created when subjected to a load case. Preprocessing can be done in Catia software. In this thesis ANSYS SpaceClaim and Catia has been used.

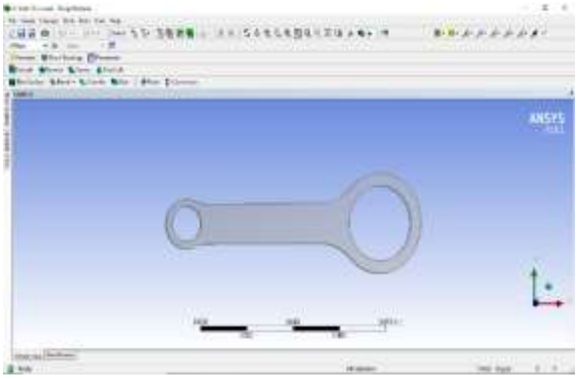


Fig.4. Preprocessing example in Space Claim of geometry for a certain 3D case

4.5 Material

The material data used is an approximation of the non-linear material data for aluminum alloy used in ANSYS Workbench AM library, see Tab.4.1. It is an approximation due to topology optimization only working for linear material properties in ANSYS 18.1.

Table 1 Material data Aluminum alloy

Property	**Value**	**Unit**
Density		kgm^{-3}
Coefficient of Thermal Expansion		K^{-1}
Young's Modulus	70e+9	Pa
Poisson's Ratio	0.3	
Bulk Modulus		Pa
Shear Modulus		Pa
Isotropic Thermal Conductivity	0.57	$W m^{-1} K^{-1}$

One important aspect to consider is that material data for any material to be manu- factured by AM has an uncertainty in material properties, expected being somewhere casting. Thus, test coupons should be printed for every manufactured part, in different directions so that material properties can be achieved and controlled for each batch.

4.6 Boundary Conditions and Loads

There exists in total one loads and one fixed support boundary conditions, relevant for the connecting rod optimization, which are listed in Table 2.

Table 2: Loads and boundary conditions for the manifold

Load type	Unit
Inlet Force	[N]
Exhaust Force	[N]
Inlet Moment	[Nm]
Exhaust Moment	[Nm]
Pressure	[Pa]
Thermal boundary low (T_{low})	[K]
Thermal boundary high (T_{high})	[K]

4.7 Topology Optimization parameters and constraints

In this section all parts of the topology optimization system are explained. Analysis settings, design region, objective, response and manufacturing constraints to be able to understand the results.

4.8 Analysis settings

In the analysis settings of the topology optimization system it is possible to define some input settings for the solver. Maximum number of iterations is an option that is set to 500 by default but can be changed to another desired number. The solver will iterate until it converges or until it has been iterating the maximum amount of times and is interrupted. For numerical reasons, the density of an element cannot be exactly zero. The min- imum normalized density has to be specified and it can be of any value between 0 and 1.By default it is set to 0.001 and is kept like that throughout this thesis. The convergence accuracy has to be specified to be equal to 2% or less, default is se to 0.1.

4.9 Optimization Region

The region to be optimized is controlled by defining the design and exclusion re- gions. The design region is allowed to be optimized whilst the exclusion region is a fixed geometry and cannot be optimized by the solver. In this thesis, the inside of the torus is always kept as an exclusion region as it is important to not change any aerodynamic surfaces for the performance of the turbine. All boundary conditions automatically become exclusion regions, but there might exist additional geometry that cannot be optimized.

4.1. Objective

The purpose of the topology optimization is its objective. The objective is often a minimization problem where in ANSYS there exists three different objectives for static structural linear optimization. The most common is to minimize compliance, which is a synonym for maximizing the stiffness. If the geometry is optimized with a linked modal system, then the objective is by default to maximize the frequency or minimize for mass or volume. It is possible to do a topology optimization with several objectives, if a modal system and a static structural system are linked together, then the topology optimization system reads and iterate over both systems. As there are two linked systems, two objectives can be defined, for example maximizing frequency and minimizing com- pliance. It is possible to link more than two systems together.

4.11 Response Constraints

The desired response during an optimization can be controlled by its response con- straints. For mass and volume there exist an option to specify the amount, in percent, to retain of the design region. In the stress constraints the possibility to define the maximum allowed stress either locally or globally is set. The mass or volume to retain has been altered in this thesis between 50 and 5% and the Global von-Mises stress constraint to less than the yield point with some margin.

4.12 Manufacturing Constraints

If a component is not to be manufactured through additive manufacturing then there might be constraints on what geometry actually is possible to manufacture. ANSYS has a set of manufacturing constraints which, if defined, the solver has to take into account when optimizing the geometry.

Table 3: Manufacturing constraints supported ansys to solvers.

Constraint type	Description
Member Size	Especially useful for AM
Pull Out Direction	Allows for mold-based manufacturing processes
Extrusion	Constant cross section along the selected plane
Cyclic	Controlling how sectors are repeated along specified axis
Symmetry	Enforce the design to be symmetric

Member size constraint specifies the minimum thickness of the supporting struc- tures and maximum thickness of connected parts. Minimum and maximum member size are a common manufacturing constraints in topology optimization, a minimum member size of 2.5 x the mesh size is set as default by the solver if nothing else is specified.Pull-out direction is constraints that can be applied and specified in either X,Y or Z direction. This constraints is to allow for manufacturing by molding, the constraint ensures that no concave shape is created inside of the part so that it gets trapped. Using the Extrusion constraint, it is possible to ensure that the resulting cross section of the final design is kept constant along the selected plane. Cyclic controls how sectors are repeated along the axis specified and yields a design which is symmetric with respect to the axis of rotation. The symmetry constraint is more generic than the cyclic constraint in the sense that symmetry is kept with respect to the user defined plane, but not controlling whether there should be any repetitive sectors or not.

4.13 Limitations and Difficulties

Doing topology optimization in 2D and working with an axisymmetric model has certain limitations in ANSYS 18.1. From*"Topology Optimization does not support: Axisymmetric model when you wish to define a Global von-Mises Stress Constraint and Local von-Mises Stress Constraint"*. This put restrictions on what is possible to achieve in these 2D cases, as more responsibility falls into the engineers hands, requiring a feeling or intuition for which output is good enough. Another aspect when choosing a geometry is to think of phenomenons such as over- hang, see section2.2, trapped powder and how to get it out. Furthermore, 2D geometry with axisymmetric loads and optimization output is just an approximation, or suggestion, for what should be further investigated in 3D.

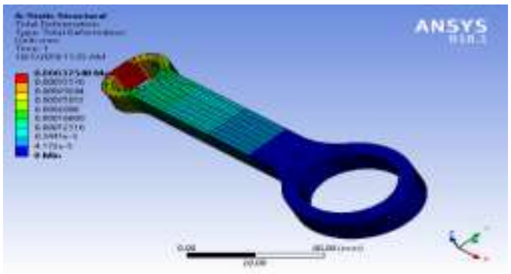
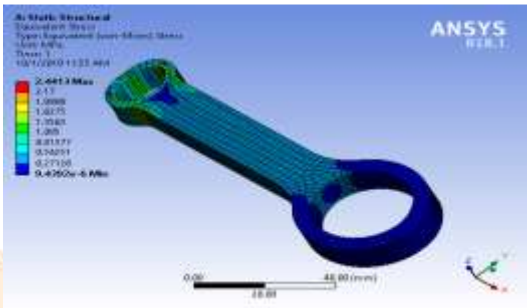


Fig 4: Total Deformation



V. RESULTS AND DISCUSSIONS

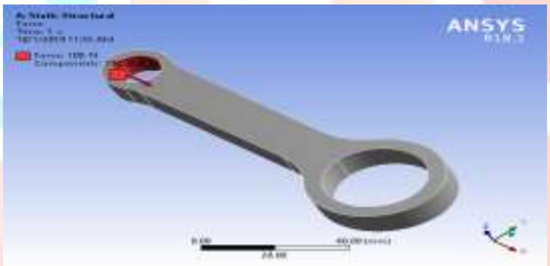


Fig 5: force applied to one end of connecting rod

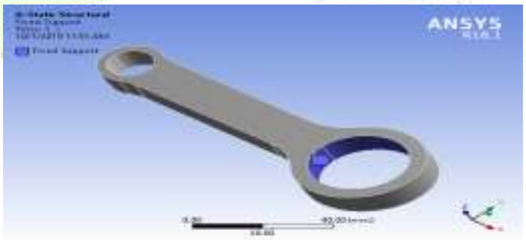


Fig 6 fix support at another end

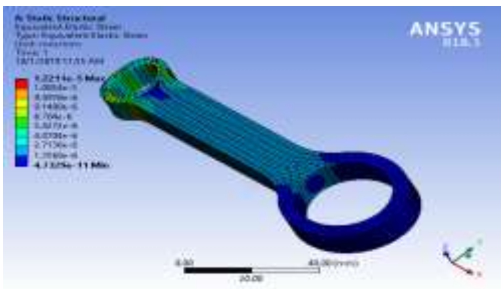


Fig 7 Equivalent strain

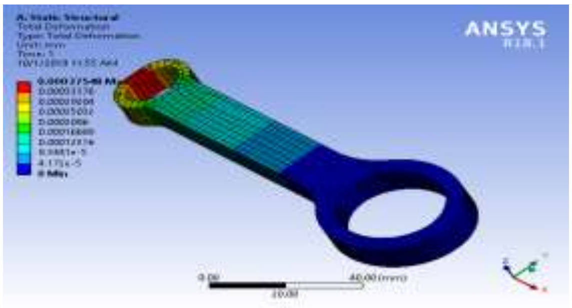


Fig 8 Total Deformation

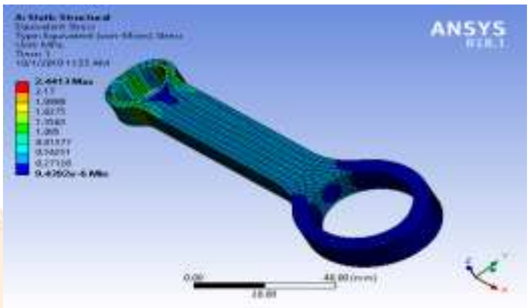


Fig 9 equivalent stress

After these results we have to assign these results to topology optimization for optimize the weight. Here showing the procedure how to assign results to topology optimization.

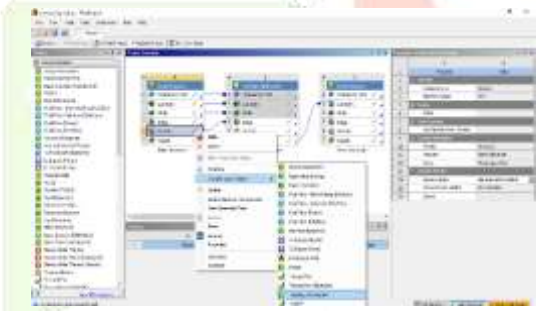


Fig 10 static structural solution transfer to topology optimization

After transfer data to topology optimization, we give the some constraints like responsive, objective constraints.

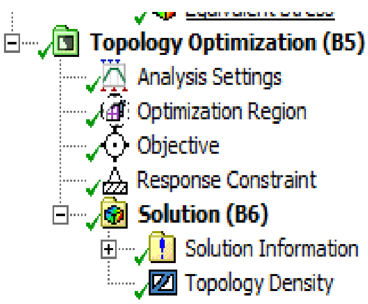


Fig 11 topology optimization

After solving the results like shows topology density

Fig 12 topology density shows optimized design

This final results in topology optimization should be validating is must, so we have to transfer the results in to transfer to validate system design.In validating system design we have convert uneven curves in to proper way. The design should be redraw as per topology density, remove extra weight by redrawing. This design import to static structural analysis.

Fig 13re draw of connecting rod

Optimized connecting rod static structural analysis

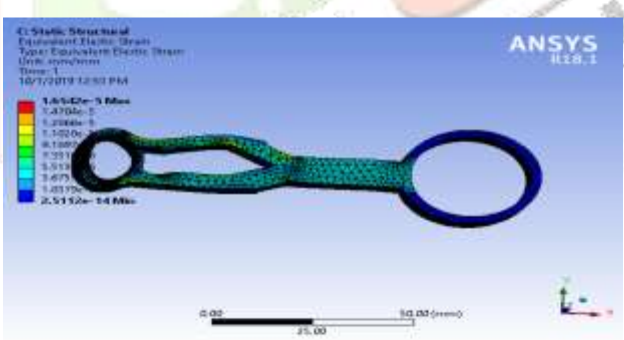


Fig 14 equivalent strain

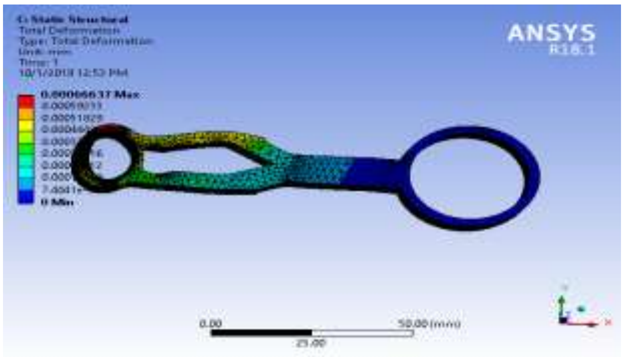


Fig 15 total deformation

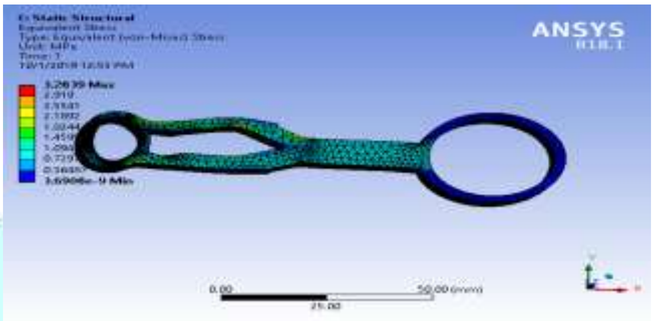


Fig 16 equivalent stress comparison of results before and after optimization stress

s.no	Stress	Minmum (MPA)	Max (MPA)
1	Before optimization	9.432E-6	2.4413
2	After optimization	3.6E-9	3.28

Deformation

s.no	Total deformation	Maximum (mm)
1	Before optimization	0.0006637
2	After optimization	0.00037548

Strain

s.no	strain	Minmum	Maximum
1	Before optimization	4.732E-11	1.22E-5
2	After optimization	2.5E-14	1.65E-5

By comprising results after optimization the results should be slight high but the connecting rod it’s optimized in weight up to 30%.

VI. CONCLUSIONS

Optimizing the manifold to meet performance and weight requirements has been proven to be possible, although quite tricky. The loads are of great magnitude and that puts limitations on how much weight actually is possible to reduce whilst still keeping the structural integrity. What kind of design space is assigned to a model is important for the final outcome from the TO; small tweaks can have a big impact, especially for interfacing surfaces. Working in 2D is hugely beneficial before working with 3D due to the speed in sim- ulation time. It is although quite hard and tedious to not be able to put the stress constraints on the axisymmetric model as it requires extra manual iterations. The 2D topology optimized geometries indicates that the connecting rod is more important than the casing for structural integrity, It is important to smooth out sharp edges and corners from the faceted data when post-processing before carrying out a design validation to mitigate local maxima. From structural analysis and design validations the pressure/force loading are the most prominent. Since topology optimization in ANSYS currently does not support thermal loads the optimized result risk to become unfeasible. This phenom- ena presents itself well.

Reducing the weight of the connecting rod seems to be possible, thus achieving the design requirement with margin whilst still keeping the global stress levels under the yield limit, where the weight was reduced by 30%. However, design validation is always necessary to confirm whether the results are feasible or not.

REFERENCE

1. M.P Bendsøe, O. Sigmund. Topology Optimization. Theory, Methods and Ap- plications, 2nd ed. Lyngby, Denmark.

2. Shiro Yoshida, Michell Truss: Shape Optimization vs Analytical Solution. SHONAN, R& D, Inc. #402 Shonan Homes, 3-19-17 Katase-Kaigan, Fujisawa, 251-0035 Japan

3. M.P Bendsøe, N Kikuchi. Generating Optimal Topologies in Structural Design Using A Homogenization Method. (North-Holland 1988).

4. M.Zhou, G.I.N Rozvany. The COC algortihm, part II: Topological, geometrical and generalized shape optimization.1991.

5. ISO/ASTM 52900:2015 (ASTM F2792) Additive manufacturing – General prin- ciples – Terminology

6. Dheeraj Gunwant & Anadi Misra Topology Optimization of Continuum Struc- tures using Optimality Criterion Approach in ANSYS. G. B. Pant University of Agriculture and Technology, Pantnagar, India

7. Shun Wang. Krylov Subspace MEthods for Topology Optimization on Adaptive Meshes. B.Eng, Tsinhua University, 2002.

8. Christo Dordlofva. Qualification of Metal Additive Manufacturing in Space In- dustry. Challenges for Product Development. Luleå University of Technology, 2018.