POLITECNICO DI TORINO

Collegio di Ingegneria Meccanica,
Aerospaziale, dell'Autoveicolo e della Produzione

Master's Degree Course in
Mechanical Engineering

Master of Science Thesis

Topology Optimization of Light Weight Gear

Supervisor:

Prof. Andrea Mura

Candiate:

Kharaghanian

Contents

1 Introduction

Gears are essential and rotating parts of transmission systems which usually are meshed with other gears to transmit torque.they are also able to to change the speed and direction of power by virtue of their gear ratio.when multiple number of gears work in sequence we have transmission . Since they are rotating parts they face vibration and have rotational inertia.most of the time the vibration is undesirable because can lead to fatigue failure ,noise and pollution.most of the time gears are over-designed , heavy and stronger than is needed. this excessive weight increase rotational ratio ,hence excessive vibration , noise an pollution .our objective in this these is to optimize a spur gear which has been design in CAD environment(SolidWorks) . we conduct a topology optimization in the Altair HyperWorks Software using optistruct solver. the solver goes through an iterative method by removing nodes from the body one by one by making sure that the safety factor is keep above . if by removing a node the safety factor goes below the criterion the node will be back at it's own place .by this method we decrease the weight of gear by removing unnecessary materials as result we have decrease the rotaional intertia of our gear.

2 The Optimum Design

2.1 Definition of the Optimum Design

You should, as a designer,always aim for optimum design. What is not so obvious is how we can recognize the "optimum" configuration precisely. Dictionary description is a strong starting point. An "optimum" is "the greatest degree or best result achieved or obtainable under particular circumstances," says the dictionary. It's the expression "unique requirements" that gives you flexibility in your design. You identify the requirements that allow you to evaluate your design alternatives as a designer. This means that you draw up mathematical equations in engineering terms that measure a design's efficiency. For example, the statement "good ride quality" will translate into a specification of the maximum values of the acceleration components that the passenger seat can experience. The quantitative parameter you are using to test a design is called objective. You might well have several goals, of course. It's very likely, for example, that a car designer will want great safety and low cost at the same time. Unfortunately, the objectives are conflicting in many situations, making it more difficult for the designer to arrive at the best solution. To make it harder for you, few designers follow their goals with the luxury of unlimited wealth. If the resources are the money you can afford to spend on materials, the quantity of fuel that the spacecraft can hold, or the maximum allowed drag coefficient for a sports car, you typically have to work between limits. These limitations, or restrictions, give rise to the topic called restricted optimization. A solution that satisfies the constraints is considered a feasible one, and an one that is not considered an unfeasible one. It's important to note that not all of the idea is done from scratch. In certain instances, we have to start from existing designs and upgrade them to the best extent possible. For various reasons , for instance, modifying a manufactured design that has failed a test might be acceptable. You should list the priorities and constraints and look for the right solution if you're starting from scratch. Typically things are a little tougher when you're working on improving an existing design, because you have less ability to alter things. One more criteria faces mechanical engineers. You have to assemble most of the components you build with other components. Together, they need to match. This means you need to deal with a package space into which your component needs to fit, and assembly points which can not be varied as other components decide on them.The package space is referred to in mathematics as the design space or the optimisation domain. Eventually, you might not be able to adjust any parameter you like. The material you can work with, for example, can be limited by factors beyond your control: working with sheet steel restricts you to commercially available thickness. The parameters you have the freedom to vary are called variables of design.

The objective's dependency on the variables of design is presented as an equation which is called the objective function. The declaration of the problem of design optimization, then, comprises of the

- package space
- design variables
- constraints
- objectives

If you miss any of these, your design ideas are more likely to be useless.[1]

2.2 Terminology of Optimization

DESIGN VARIABLES - The design variables are the structural parameters that during an optimization are free to be modified. Typical examples include material properties, a structure's topology and geometry, and the sizes of members. Depending on the type of optimization being done, design variables can be continuous or discrete.

DESIGN SPACE - The component or the section or a part that is choosen to be undergone optimization process. For instance, in our thesis internal sections of gear which is supposed to become lighter. Non-Design spaces are parameters that have been already specified and would not change in our optimization process. For example, Here the tooth of gear would not be taken into account in optimization. as an overall, any element that a force or constraint would be applied.

RESPONSE - The Response of optimization is exactly what you want to perfom in the optimization for example if you want to make a gear like it 's better to reduce you the mass

or volume of the gear, in other word is performance measuring of the system.
In OptiStruct (from the HyperWorks Help Documentation)
DRESP1 - Responses available in the software to be considered volume, volume fraction, Mass, mass fraction, compliance, weighted compliance, weighted frequency ,frequency, displacement, stress, strain, force, composite responses, , and compliance index, frequency response analysis responses
DRESP2 - Here you can have a function that is defined by the user as Response The responses can be the function of design variable design variables, grid location, table entries, responses, and generic properties

Example: Average displacement of two nodes:

$$F(x_1, x_2) = \frac{x_1 + x_2}{2}$$

DRESP3 - Response definition using a user defined external function, written in C (C++) or Fortran.

OBJECTIVE FUNCTION – The objective function is exactly what you want to do with the response, is the goal that you have defined , for instance if you want to make gear lighter you should define the volume or mass as response and then you decrease them by defining you objective as min(minimum) It represents the most significant single property of a design.

DESIGN CONSTRAINT FUNCTIONS - Sometimes you need to restrict some prameters, for example, you want to make a gear light but you don't surpass allowable stress or you want to keep you a number of elements lower than a specific number. therefore, the constraint function is A constraint imposed on a problem by restricting the values that can be taken by the system's selected response functions which must be met in order for the design to be appropriate.Usually are expressed by inequalities
Example:

$$\sigma(b,h) \leq 70MPa$$

$$t(b,h) \leq 15MPa$$

$$h \geq 2 * b$$

FEASIBLE DESIGN - If your optimization is feasible, it has met all constraints functions.

.

INFEASIBLE DESIGN - If your optimization is infeasible, it has not met all constraints functions.

OPTIMUM DESIGN - When the result of your optimization satisfies your objective function and constraints function simultaneously for example minimize your mass and met your constraints you have reached your minimum design

RESPONSE SURFACE – There is usually no continuous function that will relate the purpose to the variables of the design. Instead, a table of objective-function values versus design-variable values can be created by numerical experiments. We construct an Answer Surface by fitting a surface to this series of points, which is then used to find optimal locations.[1]

2.3 Optimum acquisition

In optimization theory, we aim for the minimum of the objective function by convention. This is not a restriction because maximizing an objective is equal to minimizing its reciprocity (it is often mentioned as minimizing the negative value of x, i.e. -x).

A function within the optimization domain which has only one minimum is called convex function. It's helpful to remember the fundamentals of differential calculus at that point. In calculus, a zero slope (or first derivative) is defined by the minimum (as well as every other 'turning point') of a curve. We are then ensured a global minimum if the objective function is a quadratic function of the design variables. The reason is that, there is only one turning point for a second order curve and thus only one minimum in the design space. .

In design space a higher order curve can have several critical points. If it is like that, then there could be multiple minimums. The critical point at which the objective function has the least value is the global minimum, whereas the other minimum values are called local minimums .

There may well be numerious design variables for a real life problem. And a non-convex function, with multiple local minima within the design space, might well be the objective function. .

Generally, the optimization is not linear even if the model for analysis is linear.Here we are going to the deflection of a cantilever beam that has a rectangular cross-section as and example. The deflection equation is

$$\delta = \frac{wL^3}{3EI}$$

Here the model is linear and our equation is a linear function . but Elasticity Modulus (E) is a function of the deflectin in plastic analysis ,Therefore, the analysis model is non-linear..

Assume we want the optimum depth (d) to be chosen for the cross-section. The Inertia Moment is :

$$I = \frac{bd^3}{12}$$

Design variable(d) is not a linear function..

The optimizer could have to look for the minimum of a non-convex function regarding the objective function chosen.

to achieve a better solution the in a fair period of time the software take the advantage of Iterative Solution.

To solve the optimization problem, OptiStruct uses an iterative procedure known as the local approximation method. This approach uses the following steps to evaluate the optimisation problem.

1. Analysis of the physical problem using finite elements

2. Convergence test; whether or not the convergence is achieved.

3. Response screening to retain potentially active responses for the current iteration.

4. Design sensitivity analysis for retained responses.

5. Optimization of an explicit approximate problem formulated using the sensitivity information. Back to 1.

After each iteration design variable adjustments are limited to a small range within their boundaries, called move limits, in order to achieve a consistent convergence. Within the first few iterations, the greatest design variable changes take place, and convergence for functional applications is usually achieved with just a limited number of FE analyses due to the advanced formulation and other stabilizing steps.
The design sensitivity analysis calculates derivatives of structural responses with respect to the design variables. This is one of the most important ingredients for taking FEA from a simple design validation tool to an automated design optimization framework.
Based on sensitivity information, the design update is created by solving the explicit approximate optimization problem. OptiStruct has introduced two groups of methods of optimization: dual method and primal method. The dual approach solves the problem of optimization in the dual Lagrange multiplier space corresponding with active constraints. For design problems involving a very large number of design variables but much fewer constraints (common for topology and topography optimization), it is highly efficient. In the original concept variable space the primal approach looks for the optimum. It is used for problems involving as many design constraints as the design variables which are typical for optimizing size and shape.[1]

2.4 Formulation of an Optimization Problem

Remember that The approch that is going to elaborated here is limited only to linear problems when there is a linear relation between responses and inputs.that is not useful for non-linear promblems.
To review, the design space, the design variables, the constraints, and the objective(s) must be defined to define a problem in design optimization.

The corresponding mathematical statement is:

Minimize$f(x) = f(x_1, x_2, x_3, \ldots .x_n)$

Subject to

$$g_j(x) \le 0 \qquad j = 0, 1, ...m$$
$$x_i^L \le x_i \le x_i^U$$

where f(x) is the objective function, g(x) are the constraint functions, and x is a vector of design variables.

An Example

We may be asked to build a light weight bracket which must fit into a volume of 300 mm x 300 mm x 600 mm. We need the steel bracket to hold a load of 100 Kg. The allowable maximum bracket deflection is 0.1 mm and the allowable maximum bracket pressure is 20 Kg / mm2. We can use sheet-steel with a thickness of 1 mm, 2 mm or 4 mm.

Our design space will be the volume of 300 mm x 300 mm x 600 mm. we want to reduce the mass therefore minimizing the mass is our objective. The constraints on optimisation will be the allowable stress and deflection. The design variables will be the steel thickness and the steel structure .

The optimizer will start with an initial structure or proposal to overcome a problem like this. The analysis program will be asked to measure the mass, stress and deformation of this structure, which are called responses to the values measured by the analysis package and monitored by the optimizer.

The optimizer will determine the sensitivity of the responses to the different variables of the design and decide whether and how much to modify.

The responses vary as the design variables change too. The mass of the bracket varies if the steel thickness shifts. The displacement, as well as the stress, will probably shift too. So, to test the responses, the optimizer will again need to request the analysis package. This iterative process will proceed until the optimizer considers that the best possible design for the deal has been found.[1]

Evaluating Sensitivity

The response quantity, g, is calculated from the displacements as:

$$g = uTq$$

The sensitivity of this response with respect to the design variable x, or the gradient of the response, is:

$$\frac{\partial g}{\partial x} = \frac{\partial q^T}{\partial x} u + q^T \frac{\partial u}{\partial x}$$

There are more constraints than design variables in some design problems, whereas others have more design variables than constraints. OptiStruct uses different algorithms, i.e. (the direct and attached variable method) for each case, in order to achieve the optimal solution efficiently (e.g. HyperWorks Support Documentation — ¿ Sensitivity).

Direct-size and shape	**adjoint**-topology
-low number of DVs	-high number of DVs
-high number of constraint	-low number of constraint
$k\frac{\partial u}{\partial x} = \frac{\partial f}{\partial x} - \frac{\partial k}{\partial x}u$	$\frac{\partial g}{\partial x} = \frac{\partial q^T}{\partial x}u + a^T[\frac{\partial f}{\partial x} - \frac{\partial k}{\partial x}u]$

The Optimization Model

I could be very time-consuming computation to ask the package to analyze the responses and time a variable is modified. OptiStruct takes a different approach: inside this approximate model, the optimizer constructs an approximate model, and does most of its work, turning back to the analysis program only when appropriate. This approach would much faster. .
It has another ramification as well. An estimation of the product's physical behavior is the analysis model itself. While an optimization model is just an approximation, it is unlikely

that the responses tested by the optimizer will be very accurate.[1]

Convergence and Iteration Control

The optimizer should make sure that the suggestions by the solver are optimal or not by searching in the design space.
The **HyperWorks Help Documentation** provides the following information:

Global Search Option

A typical question that appears when an optimization issue is solved is whether the optimum achieved is a local or global optimum or not gradient-based optimization methods are likely to find a local optimum, whereas the response surface methods and genetic algorithms are more likely to find the global optimum. As an overall, these processes increase the likelihood of discovering a more global optimum. However, no algorithm can assure that the optimum that has been found is in a real sense the optimum global. Only when the optimization problem is convex can an optimum be proven to be the global optimum. The objective function and tenable domain must be convex for a convex optimization problem, Generally speaking, most of the engineering problems that are being resolved can not be seen to be convex in fact. Therefore, a global optimum for functional problems remains elusive. Various types of algorithms merely change the odds of obtaining a more global optimum, . With that in mind, it's essential to know that algorithms that increase the chances take a lot of computation. And this will most often be considerable .
The picture below shows the definition of a convex problem as explained previously. There is only one minimum in the convex curve. Point A is a minimum. .
when we have a non-convex problem and if we use the gradient techniques the result depends on the initial point. This increases the likelihood of finding the local optimum. With the release of OptiStruct version 11.0, a new global search algorithm was made available – an extension to the gradient-based optimization approach. The technique is called the Optimization of multiple starting points. This global search algorithm conducts an extensive

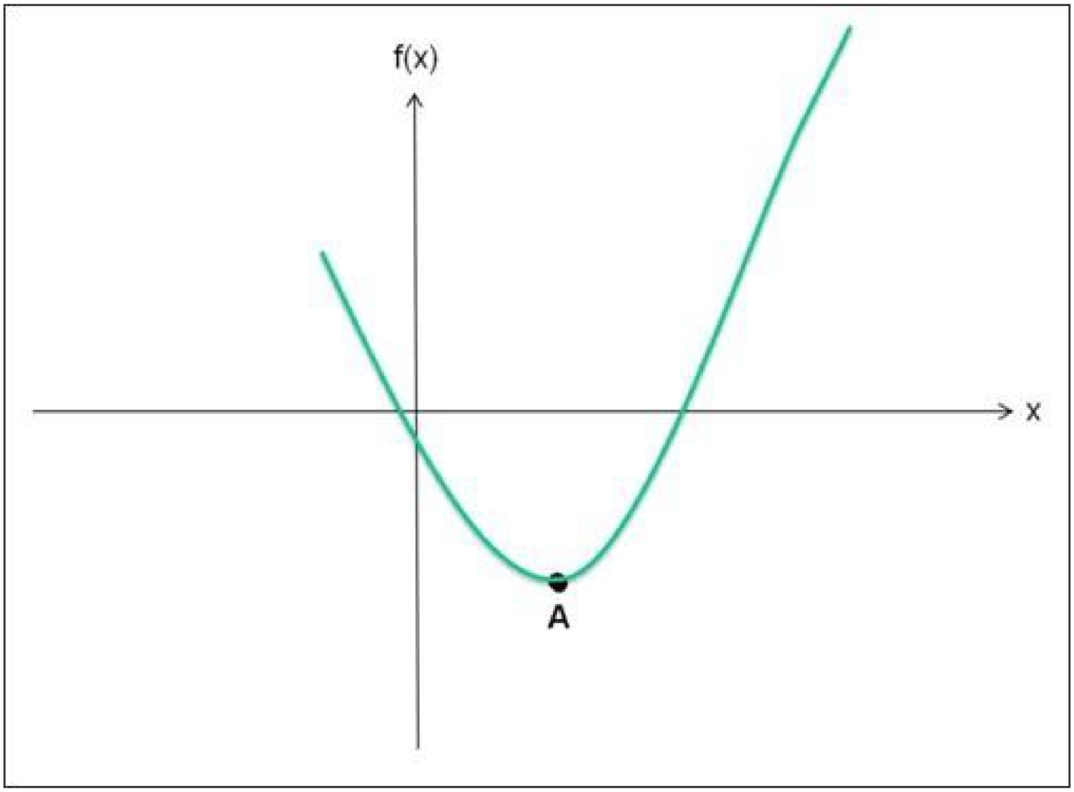

Figure 1: Convex Function,f(x)[1]

search for multiple starting points for the design space to enhance the likelihood of finding a more optimal global. N different design starting points may theoretically lead to n different optimal solutions, depending on the initial design starting point. probably by using different starting points, you can reach the same optimum result. By the way, it can not be guaranteed that the result is the global optimum. .
The following picture demonstrates this concept.
In the image, we can see three different cases of solutions with different starting points. , f(x), bounded by –a $< x <$b . starting point A and B have reached same solution P but , on the contrary , the starting point C has reached to different result Q . can be concluded that a global optimum can not be promised by multiple starting point technique. But the probability will increase.

We should be aware of the fact that our optimization is an iterative procedure. we should put a limit to the solver by defining a maximum number of iteration to specify how far the

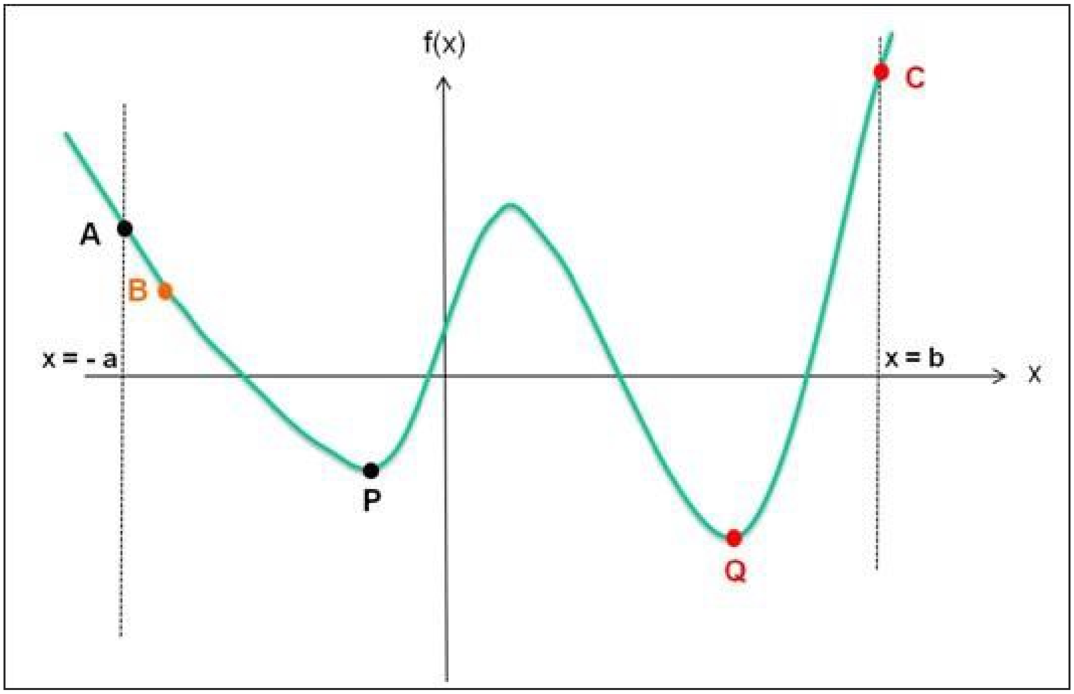

Figure 2: Non-convex functione,f(x)[1]

optimizer can go to search the solution. If the gap between two consecutive solutions is less than a convergence tolerance, the optimizer can be asked to infer that this is appropriate to us.[1]

Regular or Soft Convergence

This explanation is taken from the HyperWorks Help Documentation.

Two convergence tests are used in OptiStruct and satisfaction of only one of these tests is required. When the convergence conditions are met for two consecutive iterations, regular convergence (the design is feasible) is achieved. This indicates that for two successive iterations, the variation in the objective function is less than the objective tolerance, and constraint breaches are less than 1 For regular convergence, as a conservative estimate, three analyses are needed, because as convergence is focused on the comparison of true objective values (values derived from the analysis at the latest design point). the design occurs infeasible where the restrictions remain exceeded by more than 1%, and for three successive iterations the variation in the objective function is less than less than the objective tolerance and the alteration in the constraint violations is less than 0.2%. In this situation, the iterative process will come to an end with the assertion that no successful concept can be accomplished.

Soft convergence is attained for two successive iterations if there is little to no alteration in the design variables. . The objective (or constraints) for the final design point need not be evaluated, as the model is unchanged from the subsequent iteration. Soft convergence thus demands one iteration less than regular convergence.[1]

Gradient Search Methods

As can be seen in the figure, this technique uses the curve slope to estimate the direction in which the initial guess should be changed to increase or decrease it. The gradient is also computed using a system of finite element method. One of the several techniques used by the optimizer to shift from the initial configuration the final solution is the gradient search method, also called the steepest descent method.
Gareth Lee The explanation in the following is provided by Gareth Lee:

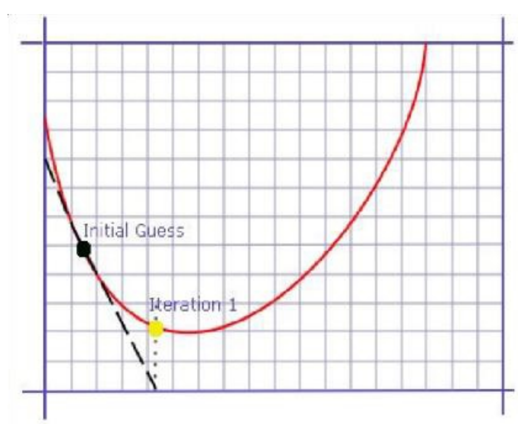

Figure 3: Gradient Search Mode[1]

For the optimum design, it all starts with an estimate. From this point, based on the gradient of the objective function, the direction in which the objective function decreases most quickly is determined. we must therefore travel in this direction to the extent that possible prior to redoing the procedure. Convergence is attained whenever the objective function gradient is 0 by iterating over, This is an algorithm for optimization that can be called the Gradient Descent Method, . this method is employed to identify the minimum of a function by the implication of gradient value as which can be defined:

1. Start from a X0 point
2. Evaluate the function F(Xi) and the gradient of the function ÑF(Xi) at the Xi.
3. Determine the next point using the negative gradient direction: Xi+1 = Xi - g ÑF(Xi).
4. Repeat the step 2 to 3 until the function converged to the minimum:

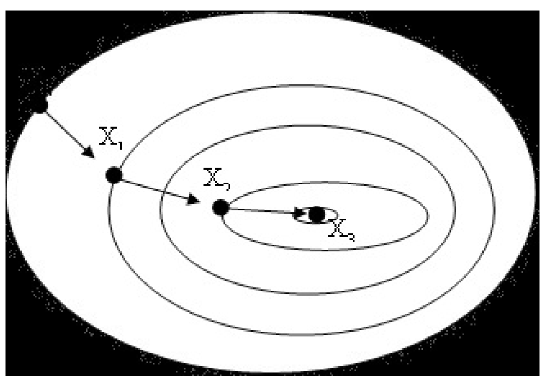

Figure 4: Gradient based method[1]

. Gradient methods are effectual whenever the sensitivities (derivatives) of the system responses can be computed effortlessly and inexpensively.

The technique of local approximation is appropriate to circumstances where :

- Design Sensitivity Analysis (DSA) is available.
- The method is applied to linear static and dynamic problems integrated mostly with FEA Solvers (i.e. OptiStruct).

Gradient techniques rely on the sensitivity of system modifications in design variables to comprehend the impact of design changes and optimize the system responses to changes in design variables .
You may use either finite-difference or empirical methods (such as the Adjoint Method) to adopt derivatives of the structural responses for linear structural analysis codes. the explicit algebraic responses with its requirements are written.[1]

Constraint Screening

The optimization model uses Constraint Screening, Constraint Linking, and Constraint Deletion to accelerate the optimization process. this method recognizes the critical constraints for the Iteration The optimizer uses one or more standards to select a subset of all variables to every iteration I to decrease the number of variables. As the optimizer progresses through the design space, this the subset is likely to shift from one iteration to another.
Constraint linking can not be always usable. for instance, if you have symmetry you can use it as a factor to reduce the number of constraints. assuming you have all beams in a structure and with the identical cross-section due to facility of purchase. In this example, by connecting all the beams you can decrease the load on the optimizer.
During the process optimizer may fail to comply with constraints for 2 or 3 times, however, the third constraint can be neglected and will be omitted for that iteration.
In the HyperWorks Help Documentation Constraint, Screening is explained as: by assessing all the objectives and constraints for every iteration there are two possible drawbacks to keeping all of these responses in the optimization problem:

1. having plenty of responses and design variables at the same cause problem for the

optimizer which is can not be acceptable.

2. It is necessary to compute the design sensitivities of these responses for the gradient technique. when there is a large number of responses and variables computation would burdensome

Constraint screening is the mechanism by which the group of responses is going to be reduced to a particular set. This collection of preserved responses retains the nature of the original design problem whilst retaining the scale of the optimization problem at an appropriate level. In this method, it is well known that the constrained response which is away from their limits or is not very critical in the same region and the same subcase will not have an impact on the direction of the optimization problem and hence can be omitted from the iteration.
imagine an optimization at which the goal is to minimize the mass of a model of a 100000 element at the same time maintaining the stresses below it's yield stress100,000 sensitivity measurements for each subcase, at each iteration must be carried out for each design variable. . Since design variable changes are constrained by movement constraints, it is not anticipated that stresses will shift significantly from one iteration to the next. It is therefore inefficient to quantify the sensitivities for those elements whose stresses are substantially lower than the yield stress of their related material. In addition, the optimization path will be driven mainly by the highest stresses. And hence, by taking into account only a random number of the highest stresses, the amount of necessary computation can be a lot diminished.
.

there must be a compromise in using constraint screening. if we do not take into account the constrained responses then plenty of iterations may be needed to attain convergence. also even if we take into account a large number of constrained responses, it takes a lot of time to attain convergence. worst we do not find a solution if we do not have enough responses comparing to the active constraints.
it has been proved by many experiments that using constraint screening can reduce the time and cost of calculation of many problems. For each response type, for each region, for each sub-case, the default settings consider only the 20 most important constraints that come

within 50 percent of their bound value.
Before the terminology “move limit” was used. This is what the HyperWorks Help Documentation says about Move Limit Adjustments:
The approximate values become less precise as the design travels away from its initial point in the approximate optimization problem. . as a consequence, the convergence speed would very low also because the estimated optimal designs are not similar to the real optimal design. To safeguard the precision of the approximations, movement limits on the design variables and/or intermediate design variables are used.
can be demonstrated as:

$$\underline{x} \leq \underline{x}_m \leq x \leq \bar{x}_m \leq \bar{x}$$

By using the small move limits smooth convergence would be obtained. with the sake of plenty of iteration, because there are small changes between iterations. By using large move limits fluctuation appears because critical constraints are incorrectly computedLarge move limits could be used if the approximations themselves are precise and correct. . Usually, in the optimization problem move limits are 20 percent of the design variable value .but if we are having an advance approximation then it a can be increased to 50 percent.
even if you have an advanced approximation you may get inadequate approximations of response according to the design variables. for precise approximations, it is more adequate to use large move limits and for the approximations which are not exact is more suitable to use move limits..
you should be aware of the fact that if you have a set of design variable which are the same, you should use move limits for all of the response approximations. it is necessary always to check the approximation of the responses which are guiding the design. These are the objective function and the most critical constraints. It is an indication that the approximations are not correct if the objective function travels in the incorrect direction or critical constraints are breached. . in this situation, all of the move limits will be smaller and reduced. nevertheless, if by the very small limits the convergence process takes so long, as a result, the design variables have to change little by little. Thense, The limits on the individual design variables that proceed to exceed the same upper or lower movement limit

are then increased. Move limits are automatically adjusted by OptiStruct.[1]

3 Topology Optimization with OptiStruct

Topology optimization was introduced as a technique to facilitate the production and development of the lightweight design. Via direct optimization of material distributions, topology optimization actually works: it can be defined as a "free shape" optimization approach. Usually, the material distribution issue is established On a model of finite element analysis comprising a design space. Each finite element will be a possible material point or void in this definition and topology optimization helps one to decide at the same time. Both the structure's external limits and the number, location, size, and shape of holes in the structure.[3]

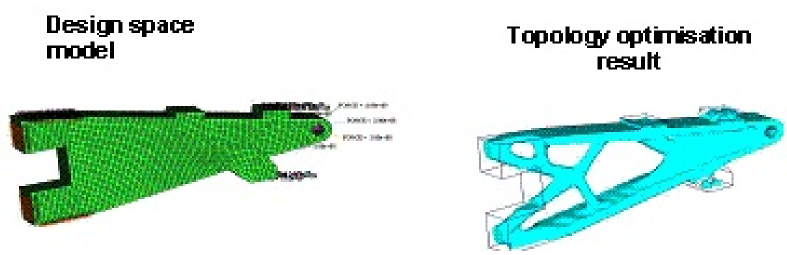

Figure 5: The topology optimisation method illustrated on an A380 Aileron Bracket[3]

The topology optimization method incorporates complete design independence by formulating structural optimization problems as problems of material distribution and all at the same provides a systematic and mathematically based method to assist in designing optimized ideas for the design. since for material distribution, plenty of design variables would be needed, this method would be computationally expensive. therefore the mathematics of the problem should be very specific to be efficient. These formulations contain formulations of energy and stiffness, which can be used to obtain the optimal shape. there is a possibility to put a stress constraint on the topology optimizations. to avoid surpassing the safe stress.[2][3] The distribution of the material along the main load direction to attain a structure with a minimum total elastic strain energy is considered a typical topology optimization problem. in other words, we also decrease the stress and strain in the structure and increase the structural stiffness by reducing the strain energy. posterior to obtaining the results of topology optimization we can carry out the eventual finite element analysis on the optimized model by considering the stresses. Obviously, optimization of topology from this definition should not be regarded as a standalone process, but a multi-step optimization process needing engineering feedback must be regarded.[3]

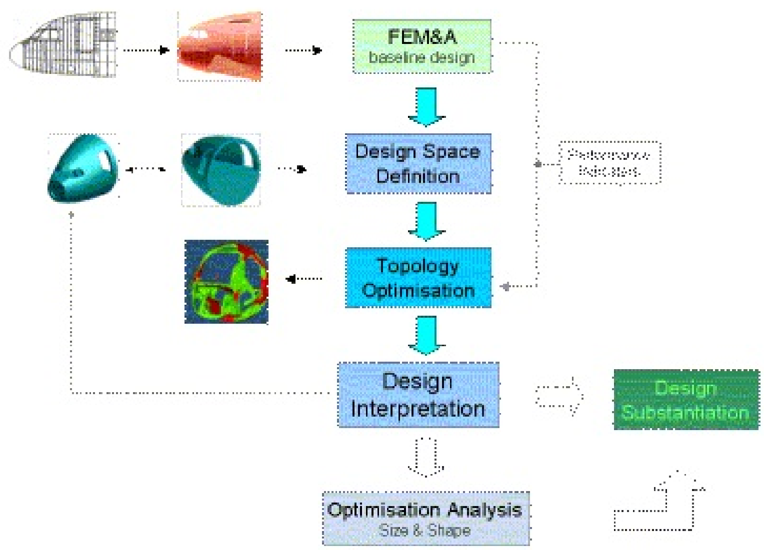

Figure 6: Numerical structure optimisation process at AIRBUS[3]

Topology Optimization

Optimizing topology is dealing with material distribution and how the members are related within a structure. It considers each element's "corresponding density" as a design variable,

. For each element, the solver computes an equivalent density at which 1 corresponds to 100% of the material, while 0 does not correspond to any material in the element. The solver then attempts to allocate a lower equivalent density to elements with a low-stress value before evaluating the impact on the residual structure. accordingly, the elements which are away from the center or are at the surface get closer to the 0 density, and the optimal design would be 1. afterward, You will need to conduct your decision. for instance, you can choose that from all (finite) elements whose density is less than 0.3 (or 30), you will exclude content. The use of an iso-plot of element densities helps to visualize the "residual" structure as it is possible to filter elements with a density below this limit, having left behind the optimal configuration. You would then need to return this configuration to the CAD environment, regulate it and reassess the template for stresses, displacements, frequencies, etc.

by using the plot of density we can see force flow and load path and take the advantages for designing.
in other words, in finite element analysis, we can see the loads and the can test the product. But in topology optimization, we can see the configuration that is able to tolerate the loads..

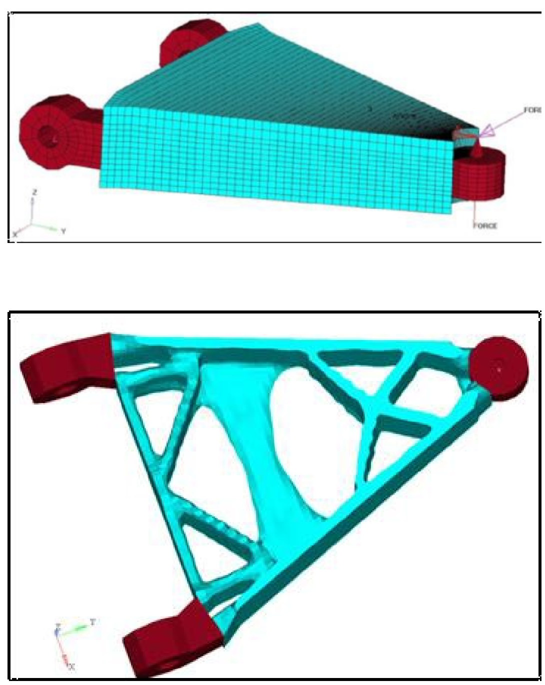

Figure 7: The remaining structure after omittig element blew threshold is caple of taking loads.[1]

About topology optimization with OptiStruct (from the HyperWorks Help Documentation).

The method that Optistruct uses for the topology optimization problems is called the SIMP

method that is a kind of density method.
in this method, the solver allocates a value of 0 or 1 to each element as a density that will represent the solid or void for each element.0 means void and 1 means solid since, plenty of discrete variables take a lot of computation, for material distribution problem we use continuous variables.
By the density process, each element's material density is explicitly used as the design variable and varies between 0 and 1 steadily; these reflect the void and solid-state. density values between 1 and 0 are not real material. There is a linear relationship between the density and stiffness of the material. This formulation of materials is in line with our understanding of traditional materials. for instance, steel is stronger than aluminum because it's density is much higher. The representation of fictional material at intermediate densities represents engineering intuitions, according to this reasoning.

generally speaking, large gray regions with intermediate densities in the structural domain are the optimal solution to problems. When we look for the topology of a given material, such solutions are not important and are not relevant when considering the use of various materials within the design space. hence, to correct intermediate densities and impose the final design to be represented by densities of 0 or 1 for each variable, techniques need to be implemented. The technique which is used to correct the intermediate density is the power law as elasticity properties. this method can be described for any 2D element and 3D solid as follow:

$$\underline{\mathrm{k}} = \rho^p k$$

There $\underline{\mathrm{K}}$ and K denote an element's penalized and real stiffness matrix, ρ is the density, respectively, and p is the penalization factor that is always greater than 1.
The parameter DISCRETE in OptiStruct corresponds to (p-1). A DOPTPRM bulk data entry can be described by DISCRETE The value of P is normally between 2.0 and 4.0.0. For instance, p=2 decreases the stiffness of the element from 0.3 to 0.09 times the stiffness of the completely dense element portion, compared to the non-penalized formulation (which is equal to p=1) at $\rho h\ddot{o}$=0.3. for shell structures, the DISCRETE is 1.0 and for solid structures is 2.0(by the proportion of a number of elements can be defined if structure is a shell or

solid there is another parameter named DISCRT1D, would be specified in the bulk data entry of DOOPTPRM. DISCRT1D requires 1D elements to use different 2D or 3D elements penalties.[1]

If the minimum size regulation is used, the penalty begins at 2 and is raised to 3 for the second and third iterative stages. This is required to obtain a more discreet approach. there are different constraints for manufacturing that will later be elaborated in details such as extrusion, draw direction, pattern grouping, and pattern repetition, in this constrains the penalty begins at 2 and rises to 3 and 4 for the second and third iterative stages. Clearly, due to the presence of semi-dense components, the results of the study which change significantly when the design process reaches a new phase using a different penalty factor.
In OptiStruct is possible to define three kinds of finite elements as topology design elements: 1D elements, shell elements, and Solid elements. the 1D elements comprise the ROD, BAR/BEAM, BUSH, and WELD elements.[1]

3.1 Responses

The following responses are currently available as the objective or as constraint functions:

Mass	Volume	Volume or Mass fraction
Center of Gravity	Moment of Inertia	Static Compliance
Static Displacement	Natural Frequency	Von-mises Stress on entire model (only as constraint)
Buckline Factor (special case)	Frequency Response Displacement, Velocity, Acceleration	Temperature
Weighted Compliance	Weighted Frequency	Combined Compliance Index
Function	Stress (DRESP1-based)	Composite Stresses
Composite Strains	Composite Failure Criteria	.....

Figure 8: Resposes[1]

The problem in the field of topology optimization is that the design concepts are usually impossible or difficult to manufactureA further problem is that if there is no suitable step taken, the solution of a topology optimization problem may be mesh dependent. When conducting topology optimization, OptiStruct provides several different methods to account for manufacturing:[1]

3.2 Member Size Control (MINDIM)

This parameter controls the smallest dimension to be maintained in the design of topology, along with minimizing the mesh-induced checker board effect and providing a more distinct design. Since the optimization pursues a discrete value of 1 or 0 for the elements, by penalizing intermediate elements that would otherwise form, this restriction typically improves the clarity of the design.

While regulation of the minimum member size (MINDIM) penalizes the creation of tiny members, it is still possible to obtain results containing members significantly below the defined minimum member size. This is because it may be very necessary for the load transmission to have a small participant in the framework and may not be eliminated by penalization. Minimum member size control acts more like a control for quality than a control for quantity. MINDIM is suggested to be at least 3 times the average element size and not more than 12 times the average element size. The average element size is measured for 2D elements as the average of the square root of the element area, and for 3D elements as the average of the cubic root of the element volume.[1]

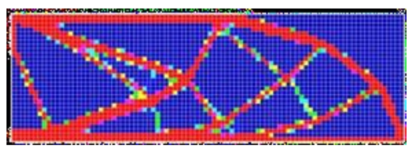

Figure 9: Without Minimum Member[1]

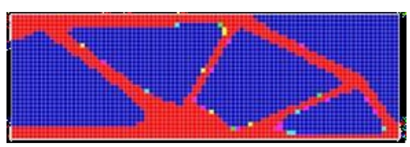

Figure 10: Minimum D=60[1]

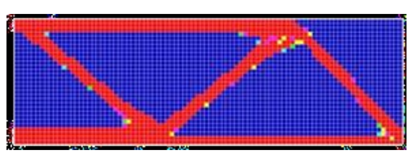

Figure 11: Minimum D=90[1]

3.3 Maximum Member (MAXDIM)

Maximum member size control avoids the creation of large members by a penalty. MAXDIM control is not directional, which means that if a member's thickness in either direction is less than MAXDIM, this restriction is met. This represents the need for the rib thickness of casting components to be monitored.

MAXDIM must be at least 2 times MINDIM, so the minimum mesh prerequisite is that for all elements referenced by that DTPL, MAXDIM must be at least 6 times the average element size. The constraint is strictly applied and if the constraints are not satisfied a termination error will appear. Moreover, MAXDIM has to be less than half the width of the thinnest portion of the design area. In order to achieve good results with this production constraint, a fine mesh is needed based on the constraints described above.

It should be remembered that the use of the maximum member size control allows the feasible design space to be more restricted and should consequently only be used if it is very suitable. Remember as well that this function is a recent research development, and the techniques are still being developed. An undesirable unintended consequence that has been found in some instances is that in the final solution it may result in more intermediate density. This function is consequently recommended to be used infrequently before the technology becomes more reliable.

The examples below show the influence of optimum control of member size on the outcome of the design.[1]

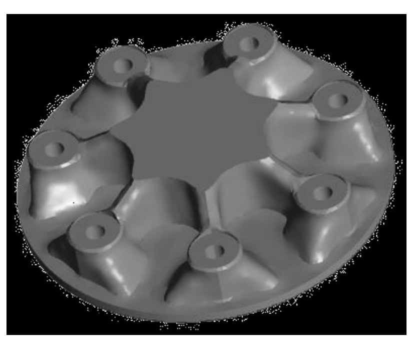

Figure 12: Without Maximum Member Size[1]

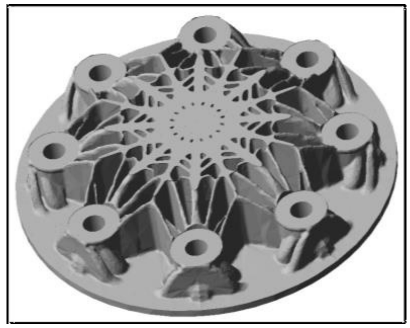

Figure 13: With Maximum Member Size[1]

3.4 Draw Direction Constraints

This feature is used for casting operation, in this process the cavities are not achievable if they are not fully open and lined up with sliding direction of the die. Topology optimisation designs also include cavities that are not feasible for casting. It may be extremely difficult, if not impossible, to transform such a design idea into a productive design.

OptiStruct provides you to enforce draw direction constraints to allow the die to slide in a given direction with the topology defined.

There are available two types of Draw options. The 'SINGLE' option implies that a single die will be used and it slides in the direction of the drawing given. The predefined contra portion for the die is the bottom surface of the considered casting portion. The 'SPLIT' option means that the part mentioned in this DTPL card will be cast with two dies splitting apart in the specified draw direction. During the optimization process, the splitting surface of the two dies is optimized.

It is also a condition of certain designs that there are no holes throughBy using the 'NO HOLE' choice, these holes can be avoided from emerging in the direction of the draw. On the DTPL card, this parameter is also determined. With 'NO HOLE,' the topology can only progressively change one layer at a time from the boundary, and it requires several iterations in some cases to eliminate one layer.[1]

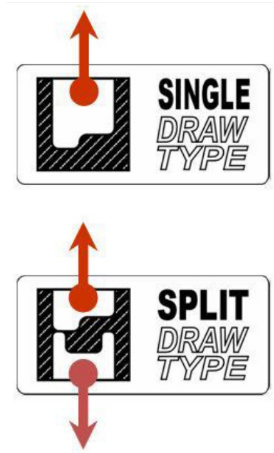

Figure 14: Single and Split Draw[1]

A stamping or sheet metal manufacturing constraint is accessible with the 'SINGLE' draw choice. This choice obliges the progression from a 3D modeling domain to a 3D shell comprehensible structure. This makes it possible to design 2D shells or stamped parts from a 3D design domain, enabling greater versatility in design.

In addition to a designable region, a cast can include a non-designable region. These non-designable regions should be described as barriers to the process of casting. The casting viability of the final structure is maintained by this

Notice also that for use with drawing direction constraints, there is a default minimum member size. Interiorly, this is estimated to be three times the average mesh size of the related pieces. The density of the mesh of the model and the intended volume fraction should also be selected such that adequate material is sure to serve members of the default minimum size. For each design component, the user may define a preferred minimum member size. This value must be higher than the default value, otherwise, the default value would be substituted.[1]

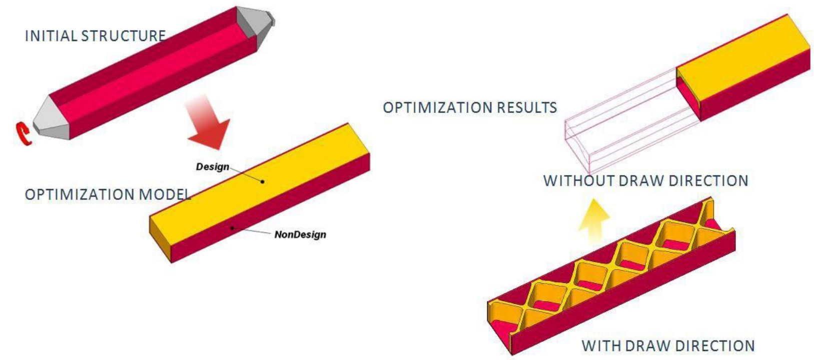

Figure 15: Draw direction[1]

3.5 Extrusion Constraints

In some instances, a design characterized by a constant cross-section along a given path is preferable, mainly in the presence of parts created by an extrusion process. Constant cross-section designs can be accomplished for solid models through utilizing extrusion manufacturing constraints in topology optimization, without paying attention to the initial mesh, boundary conditions or loads.

For the conceptual design analysis of structures that do not explicitly need to be generated using an extrusion process, extrusion constraints may also be used. These specifications can be considered as basic geometric constraints and can be used for any design requiring such features. For example, having ribs going through the entire depth of a solid domain may be ideal.

Extrusion constraints can be implemented at a part level, as with other manufacturing constraints, and can be specified in accordance with minimum member size regulation.[1]

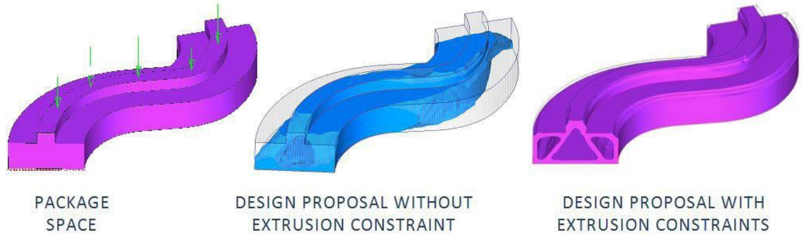

Figure 16: Beam with Extrusion constraint[1]

4 A Method to produce The optimized gear

Since the geometry of the optimized gear is very complex, producing that type of gear by the conventional methods such as milling, cutting, forging and machining, is not feasible in that case one of the best methods would be additive manufacturing (AM), advanced technology to produce complex 3d geometries by adding a layer of materials such polymer, plastic and metal or even ceramics and so on. this technology is derived from Rapid prototyping (RP) that is used widely to describe technologies which create physical prototypes directly from digital data. The basic principle of this technology is that a model, initially generated using a three-dimensional Computer-Aided Design (3D CAD) system, can be fabricated directly without the need for process planning. The model is built in a few hours, without the need for tools. It is possible to build virtually any shape. it is capable of producing of topology structures like cellular structure to reach to lightweight parts with high mechanical properties.there are many different AM methods like electron beam melting (EBM), direct metal laser sintering (DMLS), selective laser melting (SLM) and selective laser sintering (SLS). These methods all need to have 3d model digital model of lattice structure storing 3D geometric data and additive manufacturing processes use this model to slice it into each layer and translate into the trajectory tooling of the AM machine. The numerical model is usually created in computer-aided design systems.[2]

here is the topology optimization of gearing according to the reference[7]. It can be seen that this outcome can be produced conveniently by milling or other manufacturing methods and also, it is not able to attain a very light gear .

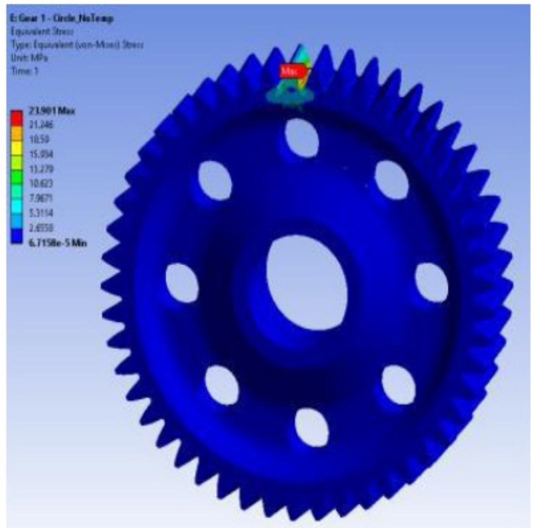

Figure 17: Plot of equivalent stress gear[7]

Figure 18: Gear with circular cut[7]

But we have a much lighter lattice structure below. The geometry, however, is very complex and can not be produced by normal production methods.in this case additive manufacturing method has been used.

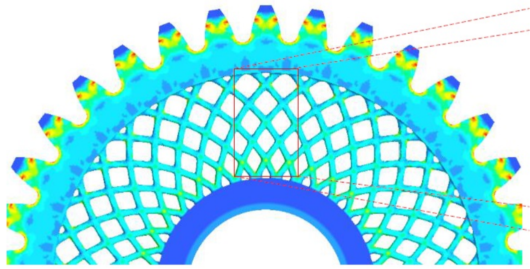

Figure 19: Lattice configuration fo spur gear [6]

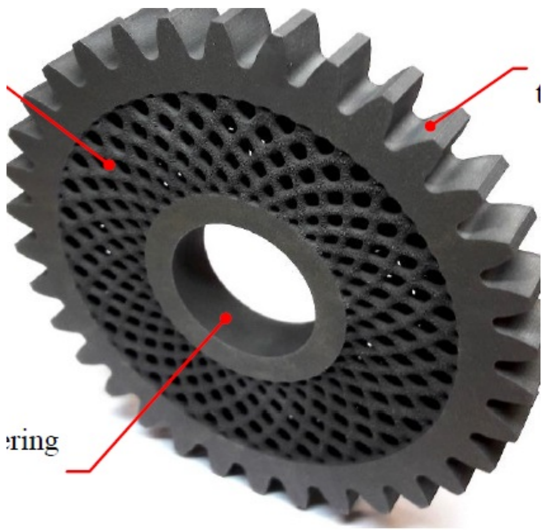

Figure 20: printed [6]

5 Topology Optimization of Gear

in this section, we demonstrate how The topology optimization is applied on gear. The purpose of this optimization is to reduce the mass of the gear to make it light and at the same time keep the stiffness of the gear high enough to satisfy all constraints.

5.1 Defining Geometry

A spur gear with the specification as below is created by SolidWorks. and has been loaded.

Table 2. Main data of gear specimen for bending tests.

Parameter	Symbol	Unit	Value
Module	m	mm	8
No. of teeth	Z	/	32
Pressure angle	A	deg	20
Face width	B	mm	20
Addendum modification coefficient	X	/	0.226
Basic rack	ISO 53.2 –B		
Tip diameter	d_a	mm	276

Figure 21: main data of gear specimen

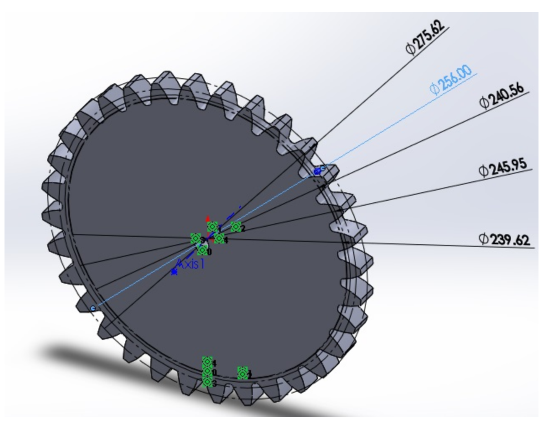

Figure 22: The initial geometry of gear

5.2 Defining Components

We must define the design space and non-design space. The design space is the one on which the optimization process and density reduction will be applied .parts are made separately and assembled in Solidworks.

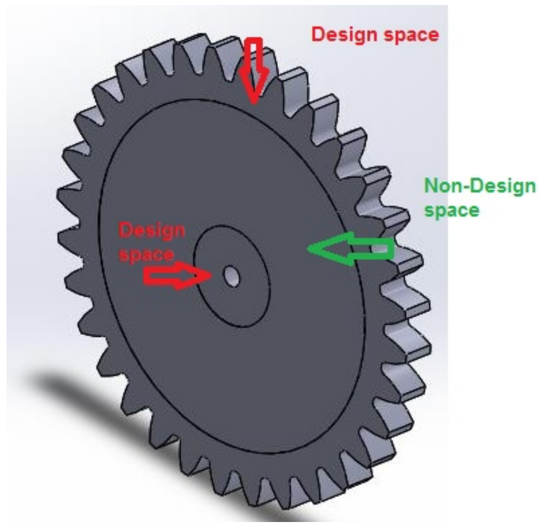

Figure 23: Design and Non-design spaces

The parts have to be loaded in Hyperworks from the menu bar, click File > Open > Geometry Model. And next step would be merging the parts by Using the Boolean feature of software.to that point we should choose solid edit from the Gemetry page and click the radio button boolean, add the solids A and B to merge, we select they operation A+B non and combine through as non.

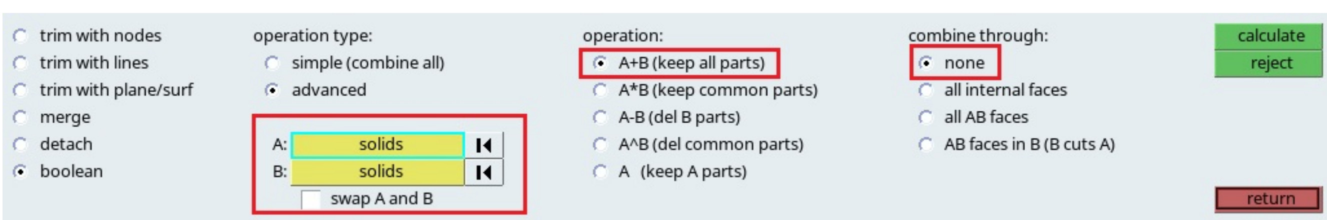

Figure 24: Boolean

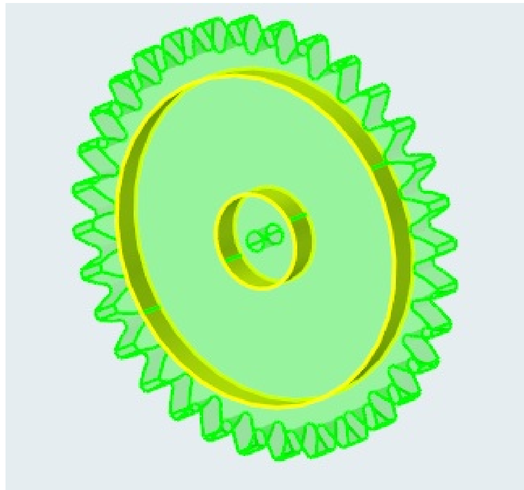

Figure 25: Merged components

5.3 Finite element model

5.3.1 Mesh

The finite element problem should be set, for this purpose first, we need to Mesh our model. By creating mesh we create a number of discrete and finite elements. Fine mesh is important for accurate calculations but we should have a compromise between coarse and fine meshes to avoid a very long analysis. Mesh the Solids using Solidmaps (multi solids) with the elements of 0.005 and source shell of mixed. We don't want to exceed the number of elements to reduce the analysis time elapses

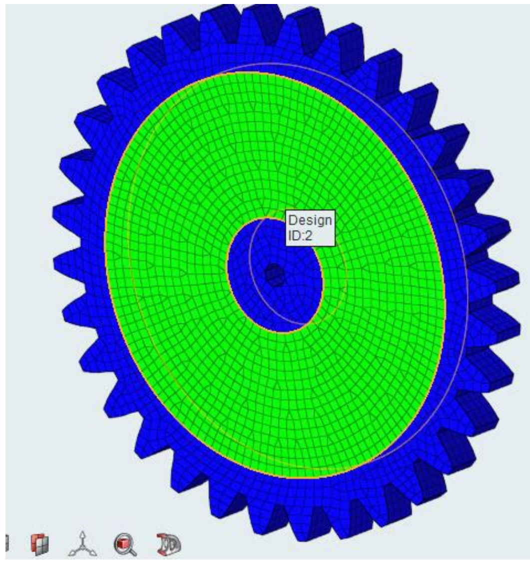

Figure 26: Meshed gear

5.3.2 Mesh quality

The mesh quality and the accuracy of the analysis are dependent on some parameters which must be checked to make sure That the analysis would be accurate.

- **warpage**

  This is the amount by which an element (or in the case of solid elements, an element face) deviates from being planar. Since three points define a plane, this check only applies to quads. The quad is divided into two trias along its diagonal, and the angle between the tria's normals is measured. Warpage of up to five degrees is generally acceptable.

  Ideal value = 0 (Acceptable <300)

- **Aspect ratio** This is the ratio of the longest edge of an element to either its shortest edge or the shortest distance from a corner node to the opposing edge. For 3-D elements, each face of the element is treated as a 2-D element and its aspect ratio determined. The largest aspect ratio among these faces is returned as the 3-D element's aspect ratio. Aspect ratios should rarely exceed 5:1.

  Ideal value =1.0 (Acceptable <5)

- **Skew**

  Skew of triangular elements is calculated by finding the minimum angle between the vector from each node to the opposing mid-side, and the vector between the two adjacent mid-sides at each node of the element. The minimum angle found is subtracted from ninety degrees and reported as the element's skew.
  Ideal value = 0° (Acceptable < 450)

- **Jacobian**

  This measures the deviation of an element from its ideal or "perfect" shape, such as a triangle's deviation from equilateral. The Jacobian value ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. The determinant of the Jacobian relates the local stretching of the parametric space which is required to fit it onto the global coordinate space. HyperMesh evaluates the determinant of the Jacobian matrix at each of the element's integration points (also called Gauss points) or at the element's corner nodes, and reports the ratio between the smallest and the largest. In the case of Jacobian evaluation at the Gauss points, values of 0.7 and above are generally acceptable. Ideal value = 1.0 (Acceptable ¿ 0.5)

The check these parameters we go to the **tools** page select the **check elems** then select the radio button of **3-d** and there we compare the parameters with the standard amount.

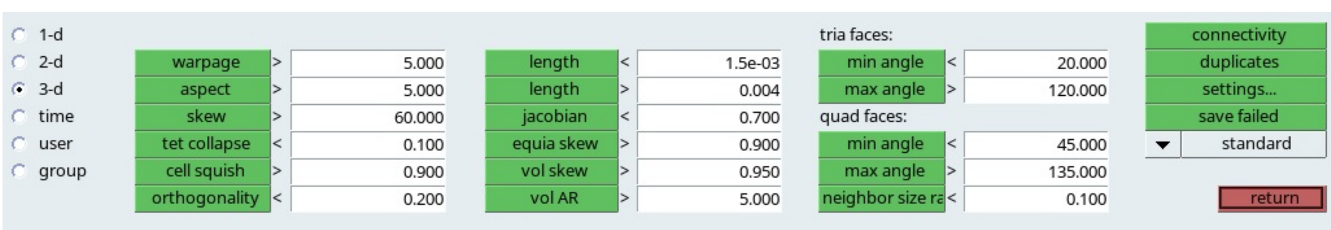

Figure 27: Element check

5.3.3 Definging Material

The material we defined for this problem is MAT1 Steel with Young Modulus 210000 and Poisson's ratio 0.3 and mass density of 7.85e-09. for defining the material. From the menu bar click the **model** and the select the **Creat material**. The Card image is set as Mat1.

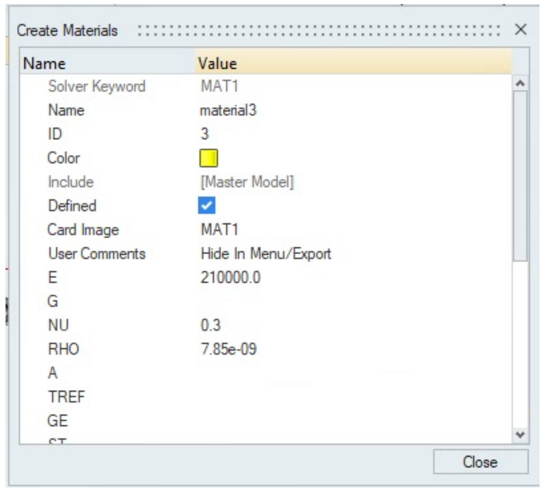

Figure 28: Create material

5.3.4 Property

To define the property we go to the Model from the menu bar select property. since we are working with solids the card image must be selected as PSOLID and we select the material as we have already defined.

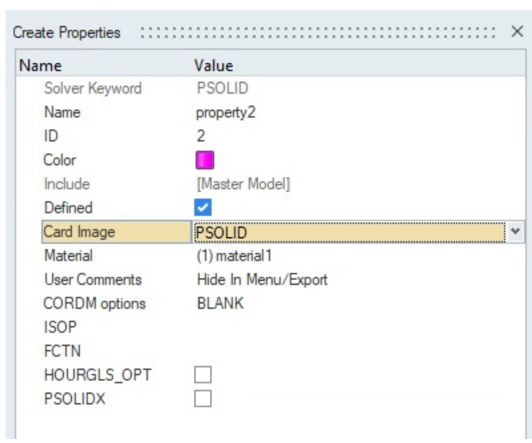

Figure 29: Property

in the end, the property and material are assigned to the Design and Non-design spaces. The constraint and Forces have to be created .constraints on all the point inside the hole shafts in all directions are applied .

The forces have to be applied on the pitch line , for the moment the nearest points to the pitch line are used . The magnitude is 3000 N , since there are five points ,is divided by 5 and inserted 600.

5.3.5 Constraints

The constraints must be applied on the Ring to that point we go on the **Anlysis** page select **constraints**. The ring must be constrained on all six degrees of freedom and then select the nodes with load type of **SPC**.

The force

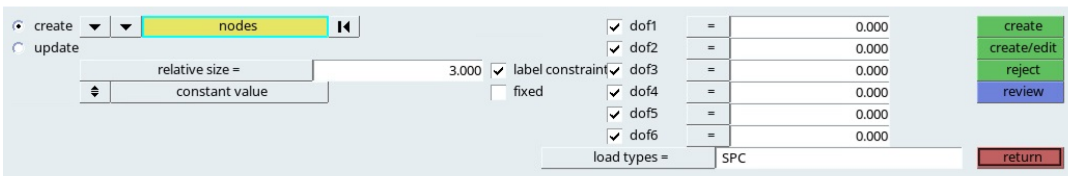

Figure 30: constraints on all directions

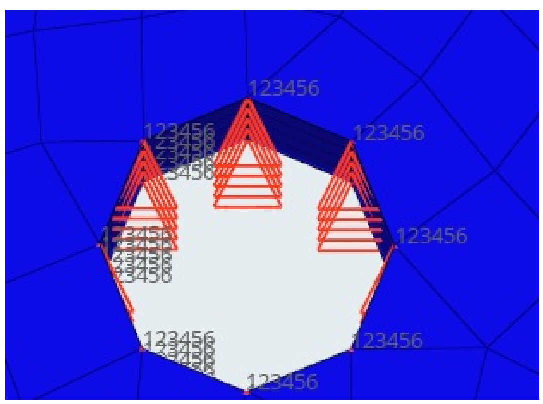

Figure 31: Constraints nodes

5.3.6 FORCE

The forces have to be applied on the pitch line, for the moment the nearest points to the pitch line are used. The magnitude is 9 kN , since there are 11 nodes, is divided by 11 and inserted 818.18.

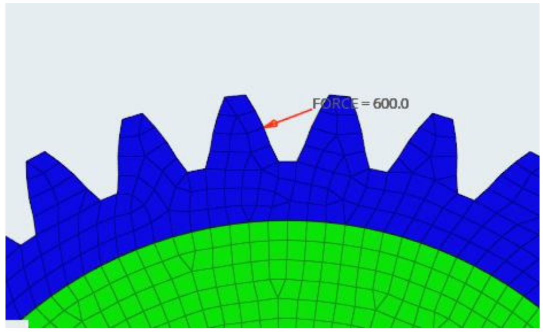

Figure 32: Force applied on Gear tooth

5.3.7 Load step

Creating the LoadSteps with the type of linear static and Then we run the simulation. Here is the result of the Finit analysis of this model.

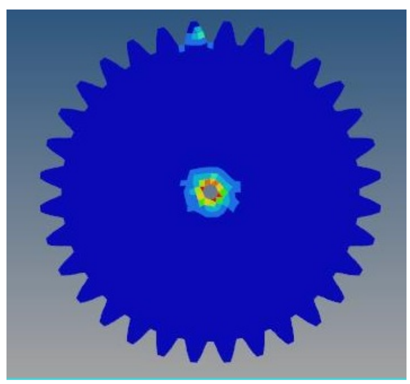

Figure 33: von mises stress plot

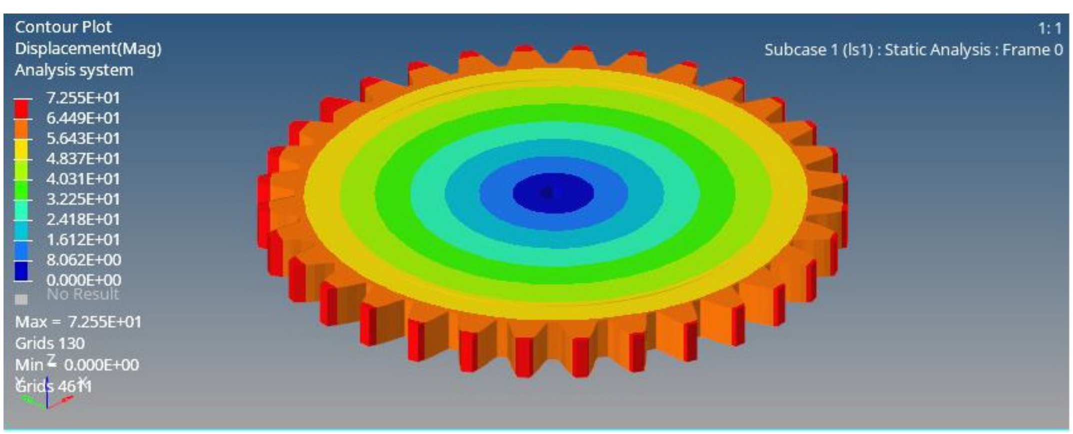

Figure 34: displacement magnitiude

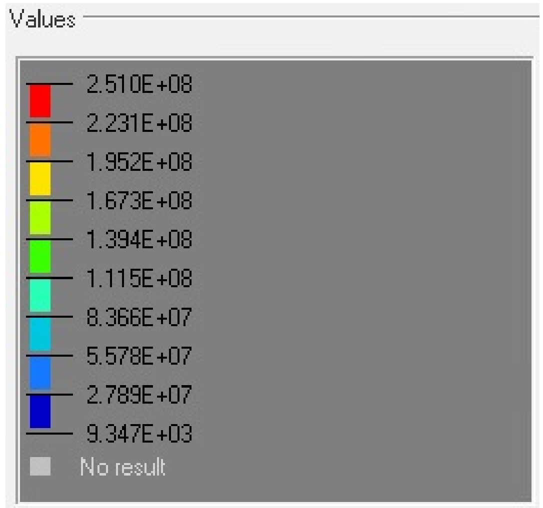

Figure 35: Stress values of the gear

5.4 set topology optimization

Defining the design parameters for topology optimization. it is defined as a minimum member with 1000. And use extrusion to avoid the reduction in the thickness of gear. Our Response would be volume fraction .the objective would be minimizing the compliance and it is put a constraint of 0.1 on the volumfrac to avoid the reduction more than 10 %.
and run the optimization. From the optimization page, we select the Topology to define the Design space as the property the design property must be set by this chose all the optimization process would be done on this space.

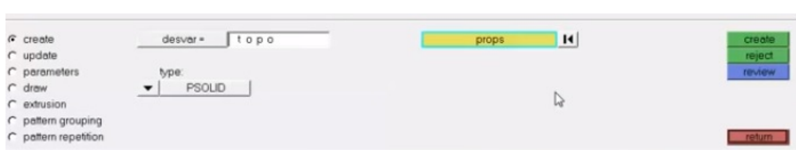

Figure 36: Create topology optimization

The type depends on the dimension of our model here should be PSOLID

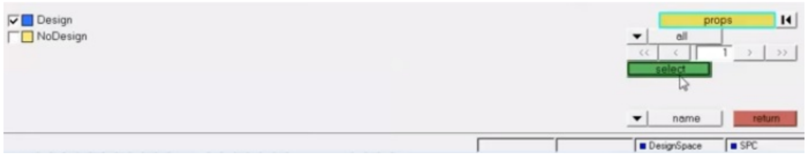

Figure 37: defining properties for topology optimization

we have to define the Response of the optimization, we define two response for our optimization one is the compliance and the other one is voluefrac.
The compliance is the opposite of stiffness by minimizing the compliance we keep stiffness high

Figure 38: compliance

by choosing the volumefrac we reduce the volume of our model according to the mathematical method of our solver.

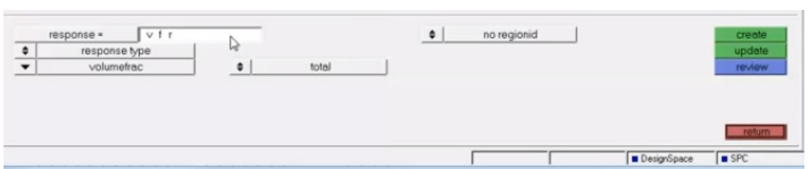

Figure 39: volume fraction

now we go to the deconstaint page here we constraint our Response, we give upper bound of 0.1 to the volume frac to avoid reduction of the volume more than 10 percent.

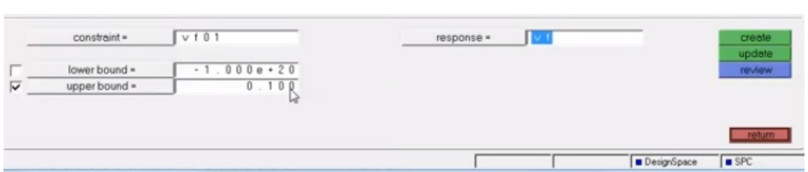

Figure 40: upper bound limit for volume reduction

the objective of optimization has to be set . that is minimizing the compliance which is going to keep the stiffness high.

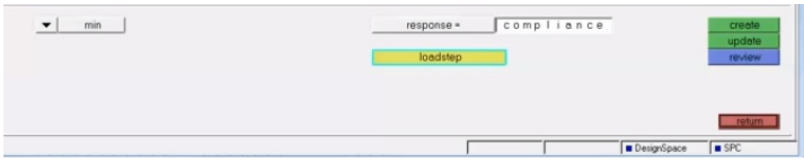

Figure 41: minimizing as objective

the optimization is run .here is the result of the optimization.

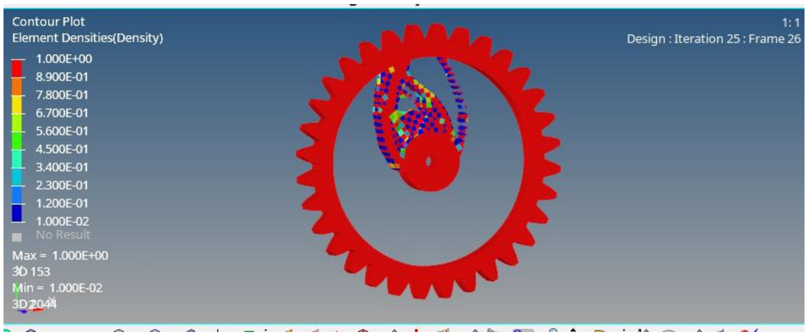

Figure 42: contour plot of the optimized gear

Now we are going to see the effect of the draw on our optimization. The draw option let the solver decide whether to extract material from one or two different directions. for instance, you have a milling process and you can have the milling from one or the other side .but not from inside to outside that would be quite helpful at the time of manufacturing.

5.4.1 Draw

At the topology page choosing the draw radio button, we are able to define draw direction. there are different types of draw which have been explained with details in previous chapters. we define two nodes to represent the direction.

SINGLE DRAW

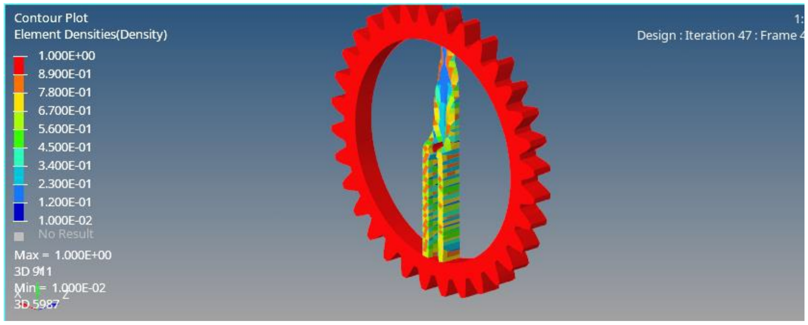

Figure 43: Single Draw

SPLIT DRAW

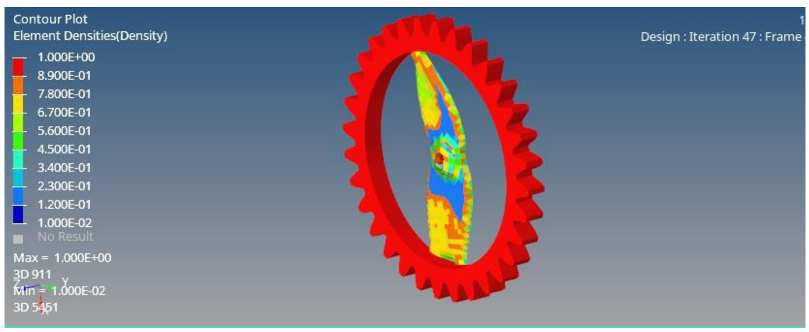

Figure 44: Split draw

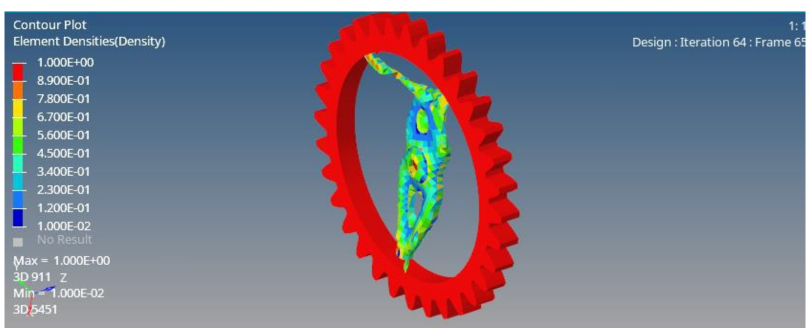

Figure 45: Split draw with different direction

RADIAL DRAW

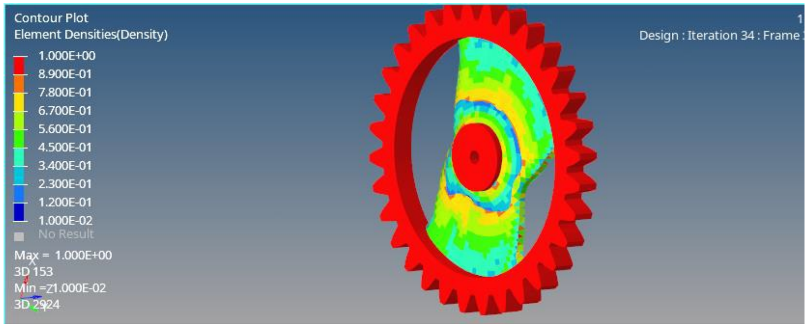

Figure 46: Radial draw

Finally, we have decided on the geometry of the gear. the gear is elaborated on the SolidWorks .and a final geometry has been obtained.

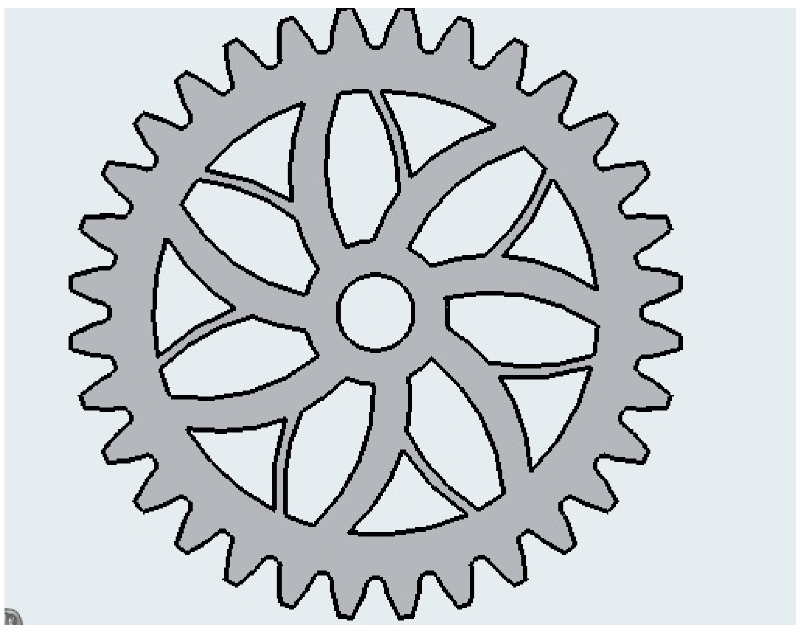

Figure 47: Optimized gear

the mass of gear has reduced from 1.05 to 0.63 which is about 40 percent reduction. for the last time, finite element analysis will be executed on the new gear to investigate that the new gear will pass the safety factors by von misses theory of stresses.

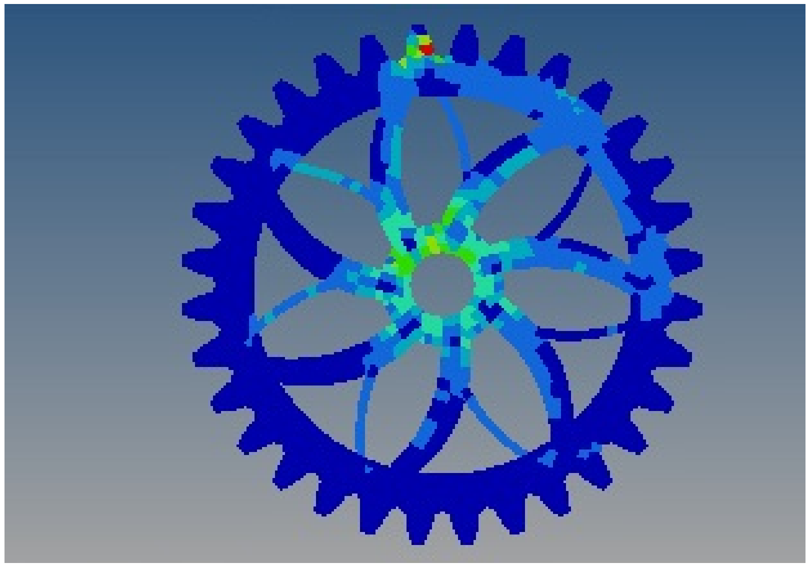

Figure 48: von mises plot of optimized gear

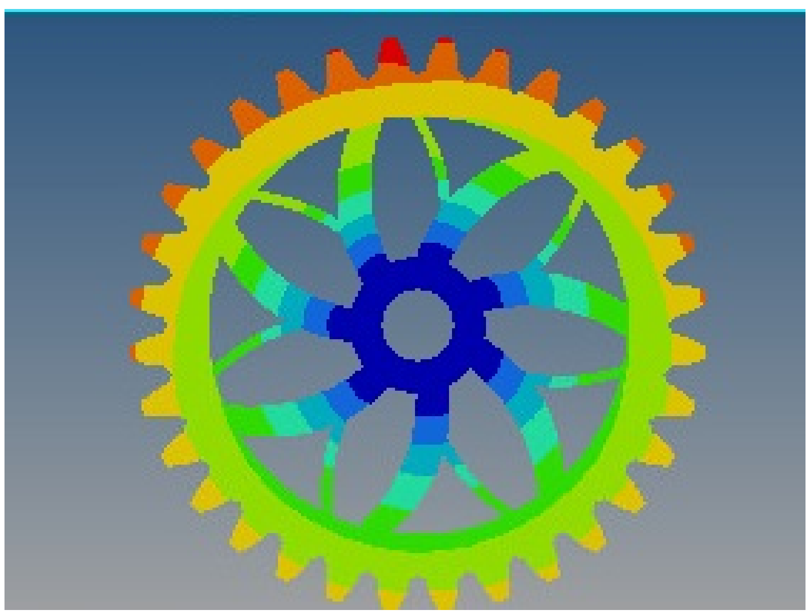

Figure 49: Displacement plot of optimized gear

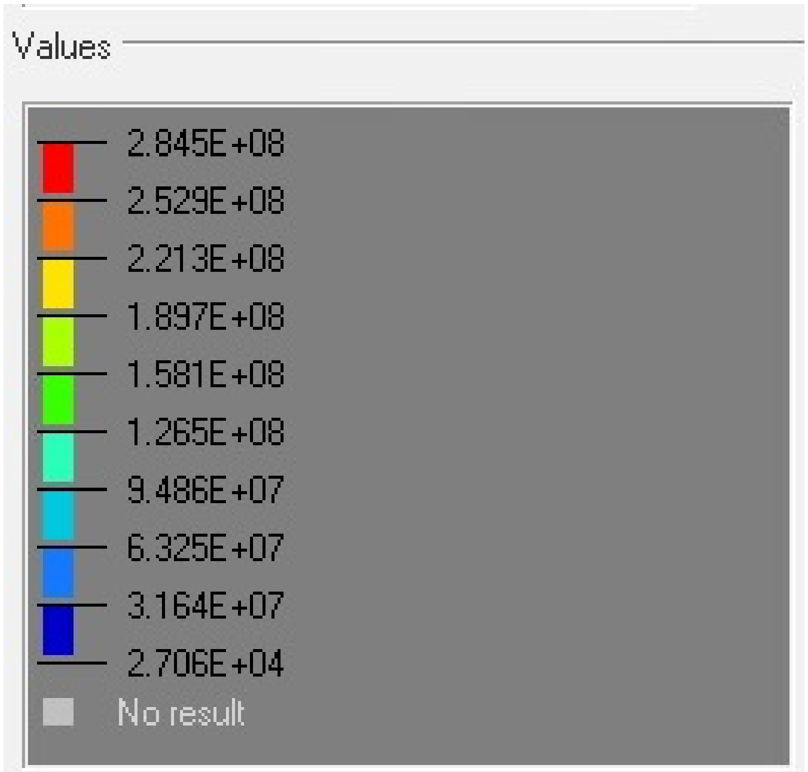

Figure 50: Stress value of optimized gear

Table 1: Performance comparision between gears

	original gear	optimized gear
weight	1.07e-12	0.68e-12
max stress(MPa)	251‘	284
Displacement	7.25	9.73
safety factor	1.39	1.23

6 Conclusion

Topology optimization with OptiStruct method has been conducted on a gear respecting constraints and correct load . The Results demonstrated a new and peculiar design. Considering that This desing is not the definitive desing. The component must be redesign with CAD software to reach the detailed and flawless gear . by taking into consideration of all these factors we have gained approximately 40% reduction in weight by keeping the safty factor greater than 1.2 .

References

[1] Altair ebook, *practical aspetcs of structural optimization*,2018

[2] Dinh Son Nguyen, Frédéric Vignat, *A Method to Generate Lattice Structure for Additive Manufacturing*,2016

[3] L.Krog1, S.Grihon2, A.Marasco3, *Smart design of structures through topology optimisation*,2009

[4] Altair ebook, *3D theory*,2012

[5] Mariangela Casiello, *Static and Dynamic Topology Optimization for Aeronautical Gears*,2018

[6] Riad Ramadani,Jožef Predan,Ales Belsak, *TOPOLOGY OPTIMIZATION BASED DESIGN OF LIGHTWEIGHT AND LOW VIBRATION GEAR BODIES Gears*,2018

[7] Mit Patel, Hadiya Valiulla, Vinay Khatod, Bhavin Chaudhary, Vikas Gondalia, *Topology Optimization of Automotive Gear using Fea*,2019

[8] Chinmay Shah, Swapnil Thigale, Rathin Shah, *Optimizing weight of a Gear using Topology Optimization*,2018

[9] Yuvraj P. Mali, Dr. E.R. Deore *A Review on Design Analysis with Weight Optimization of Two Wheeler Gear Set*,2017