ISTANBUL TECHNICAL UNIVERSITY ★ GRADUATE SCHOOL OF SCIENCE ENGINEERING AND TECHNOLOGY

TOPOLOGY OPTIMIZATION METHOD AND AUTOMOTIVE BRACKET OPTIMIZATION

M.Sc. THESIS

Melek HATİPOĞLU

Department of Mechanical Engineering

Mechanical Design Engineering Program

MAY, 2015

İSTANBUL TECHNICAL UNIVERSITY ★ GRADUATE SCHOOL OF SCIENCE ENGINEERING AND TECHNOLOGY

TOPOLOGY OPTIMIZATION METHOD AND AUTOMOTIV BRACKET OPTIMIZATION

M.Sc. THESIS

Melek HATİPOĞLU
(503121215)

Department of Mechanical Engineering

Mechanical Design Engineering Program

Thesis Advisor: Prof. Dr. Hikmet KOCABAŞ
Anabilim Dalı : Herhangi Mühendislik, Bilim
Programı : Herhangi Program

MAY, 2015

İSTANBUL TEKNİK ÜNİVERSİTESİ ★ FEN BİLİMLERİ ENSTİTÜSÜ

TOPOLOJİ OPTİMİZASYON METODU VE OTOMOTİV BRAKETİNİN OPTİMİZASYONU

YÜKSEK LİSANS TEZİ

Melek HATİPOĞLU
(503121215)

Makine Mühendisligi Anabilim Dalı

Konstrüksiyon Programı

Tez Danışmanı: Prof. Dr. Hikmet KOCABAŞ
Anabilim Dalı : Herhangi Mühendislik, Bilim
Programı : Herhangi Program

MAYIS, 2015

Melek Hatipoğlu, a **M.Sc.** student of ITU **Graduate School of Mechanical Engineering** student ID **503121215**, successfully defended the **thesis** entitled "**TOPOLOGY OPTIMIZATION METHOD AND OPTIMIZATION OF AUTOMOTIVE BRACKET** ", which she prepared after fulfilling the requirements specified in the associated legislations, before the jury whose signatures are below.

Thesis Advisor :	**Prof. Dr. Hikmet KOCABAŞ** İstanbul Technical University	....
Jury Members	**Prof. Dr. Ferhat DİKMEN** Yildiz Technical University	....
	Yrd. Doc. Dr. Atakan ALTINKAYNAK İstanbul Technical University	....

Date of Submission : 4 May 2015
Date of Defense : 28 May 2015

FOREWORD

I would like to thank my thesis advisor Prof. Dr. Hikmet Kocabaş for his help, guidance and patience during this study.

May 2015

Melek Hatipoglu
(Mechanical Engineer)

TABLE OF CONTENTS

Page

FOREWORD .... **vii**
TABLE OF CONTENTS .... **ix**
ABBREVIATIONS .... **xi**
LIST OF TABLES .... **xiii**
LIST OF FIGURES .... **xv**
SUMMARY .... **xvii**
ÖZET .... **viii**
1. INTRODUCTION **1**
1.1 Thesis Purpose 1
1.2 Scope 1
1.3 Literature Survey 2
2. OPTIMIZATION THEORY **5**
2.1 Multiple Criteria Optimization 5
2.2 Structural Optimization 6
2.2.1 Sizing optimization 7
2.2.2 Shape optimization 7
2.2.3 Topology optimization 7
2.2.4 Density method 8
2.2.5 Homogenization method 8
2.3 Numerical Problems in Topology Optimization 10
3. SOFTWARE TOOL REVIEW **13**
3.1 OptiStruct 13
3.1.1 Topology optimization with OptiStruct 14
3.1.2 Software usage 15
4. ENGINE ACCESSORY DRIVE SYSTEM **17**
4.1 Engine Accessory Drive System Design Basics 17
4.1.1 Accessory design target 19
5. BRACKET TOPOLOGY OPTIMIZATION **21**
5.1 Natural Frequencies and Undamped Multiple d.o.f. Free Vibration 22
5.2 Bracket Topology Optimization Step 1: Natural Frequency Analysis 22
5.3 Bracket Topology Optimization Step 2: Desgin Space Creation of Bracket ... 23
5.4 Bracket Topology Optimization Step 3: FE Model Creation 24
5.5 Bracket Topology Optimization Step 4: Non-Design Space, Design Space 25
5.6 Bracket Topology Optimization Step 5: Property Collections Creation 26
5.7 Bracket Topology Optimization Step 6: Load Collector Creation 26
5.8 Bracket Topology Optimization Step 7: Optimization Response Function 27
5.9 Bracket Topology Optimization Step 8: Manufacturing Constraint 28
5.10 Bracket Topology Optimization Step 9: Optimization Results Monitoring .. 29
5.11 Bracket Topology Optimization Step 10: Optimizated Design Reformation 30
5.12 Bracket Topology Optimization Step 11: Natural Frequency Analysis 31
5.13 Bracket Topology Optimization Step 12: Stress Analysis of Latest Design.. 32
6.RESULTS AND CONCLUSION **35**

REFERENCES **37**
APPENDICES **39**
CURRICULUM VITAE **51**

ABBREVIATIONS

FEAD	**:** Front End Accessory Drive
FEM	**:** Finite Element Method
SIMP	**:** Single Isotropic Material with Penalization
2D	**:** Two Dimensional
3D	**:** Three Dimensional
NVH	**:** Noise Vibration Harshness
A/C	**:** Air Condition
HPDC	**:** High Pressure Die Casting
RPM	**:** Revolution per Minute
DOF	**:** Degree of Freedom

LIST OF TABLES

Page

Table 5.1: Material properties of AlSi9Cu. 21
Table 5.2: Bracket design targets. 21
Table 5.3: Compressor and alternator hub load values. 33
Table 6.1: Comparision of designs. 35

LIST OF FIGURES

Page

Figure 1.1: Design and non-design zone of driveline bracket. 2
Figure 1.2: Optimization results of the driveline bracket. 2
Figure 1.3: Design space of engine mounting bracket... 3
Figure 1.4: Optimization output and optimization based CAD model. 3
Figure 1.5: Alternator bracket and its fixing points to the engine. 4
Figure 1.6: Bracket desgin changes during the optimization process........................ 4
Figure 2.1: Model problem graphical sample. 6
Figure 2.2: Different types of optimization a) size optimization b) shape optimization c) topology optimization. 7
Figure 2.3 : Examples of microstructures with rotation in 2D: a) Microstructure with rectangular holes and b) Layered microstructure.. 9
Figure 2.4: Checkerboard patterns on optimized structure. 10
Figure 2.5: Solution by using 600 elements.. 10
Figure 2.6: Solution by using 5400 elements.. 10
Figure 2.7: Convex problem, global mininum solution. 11
Figure 2.8: Non-convex problem, local and global mininum solutions................... 11
Figure 3.1 : Optimization process workflow. 13
Figure 3.2 : Beam optimization problem. 16
Figure 4.1 : Engine accessory drive system (engine front view). 17
Figure 4.2 : Engine front end accessory drive system (side view). 18
Figure 4.3 : The graph is the average of all driving styles. (shows that city drivers will spend high percentage of their time at idle speed)............................... 18
Figure 4.4 : Front end accessory drive system parts. 19
Figure 4.5 : High pressure die casting (Learneasy, 2015). 20
Figure 5.1 : Baseline model first natural frequency is 167 Hz. 23
Figure 5.2 : Bracket position and its fixing points. 24
Figure 5.3: a) Base model front side view (b) design space front side view (c) base model rear side view (d) design space rear side view.......................... 24
Figure 5.4: Hyperworks screenshot- rbe2 element degree of freedom contraints (dof1,2,3 tranlsational dof 4,5,6 rotational motion)............................. 24
Figure 5.5: Second order 3D tetraheral element (TETRA10)................................. 24
Figure 5.6: Bolt connections are modelled with rbe2 and cbar elements. Design space modelled with 140416 nodes and 710480 elements. 25
Figure 5.7: FE model of design space... 25
Figure 5.8: HyperMesh screenshot to creat design and non-design space................ 25
Figure 5.9: Non-design space and design space areas are seperated. 26
Figure 5.10: HyperMesh screenshot to define material properties.......................... 26
Figure 5.11: Engine is constrained from cylinder upper surface. 27
Figure 5.12: HyperMesh screenshot to define response function for frequency. 27
Figure 5.13: HyperMesh screenshot to creat load step. 27
Figure 5.14: HyperMesh screenshot for volume fraction response. 28

Figure 5.15: HyperMesh screenshot volume fraction response contsraint function. 28
Figure 5.16: Single die draw direction. 28
Figure 5.17: HyperMesh screenshot draw type definition. 28
Figure 5.18: Frequency response change for each iteration. 29
Figure 5.19: HyperView screenshot optimization results of design space. 29
Figure.5.20:.HyperView screenshot, elements which have density value 0.8 are removed. 30
Figure 5.21: Red part is optimization result, blue one is after modification (front side view). 30
Figure 5.22: Red part is optimization result, blue one is after modification (rear view). 31
Figure 5.23: HyperMesh screenshot, EIGRL card image. 31
Figure 5.24: Latest design bracket mesh model. 31
Figure 5.25: Latest design bracket natural frequency analysis result. 32
Figure 5.26: Compressor and alternator hub loads. 32
Figure 5.27: Stress analysis results. 33
Figure 6.1 : Design comparision of base model and latest design bracket front view. 35
Figure 6.2 : Design comparision of base model and latest design bracket rear view. 35

TOPOLOGY OPTIMIZATION METHOD AND AUTOMOTIV BRACKET OPTIMIZATION

SUMMARY

At today`s competitive automotive industry manufacturers and developpers need to find cost reduction ways without comprimising from quality. Optimization is being used widely in last decade for this purposes.

The objective of this thesis to understand topology optimization methods and to perform topology optimization for automotive engine bracket which is used to assemble air conditioner compressor to engine. According to design rules all the engine component`s first natural frequency must be higher than engine working frequency to prevent resonance. As a general NVH (noise, vibration, harshness) target to ensure no resonances within the driving range. Automotive companies get most of engine components from suppliers and this components are being used for past and future engine projects. Suppliers are educated with project design targets and produce parts accordingly. So components like compressor, alternator etc. are not new for every new project and not designable for the companies directly.

At first section thesis purpose and scope is clarified. Purpose is to understand optimization theory and optimize automotive bracket with gained knowledge.

At second section optimization theory is investigated. Mathematical algorithm is basically showed and then optimization types explained. There are mainly 3 types of optimization as size, shape and topology. Topology optimization produces an optimized shape and material distribution for a structure within a given design space.

At third section software tools which are used for topology optimization explained. HyperMesh and OptiStruct which are parts of HyperWorks that released by Altair Engineering are used.

At forth section the optimization process of engine bracket is explained. Some basic knowledge is given about engine front end accessory system parts and system design requirements. Bracket material properties, bracket production method are clarified.

Bracket optimization problem stated as to maximize first mode natural frequency. Design space is created by using Catia, finite element model is created by using HyperMesh. By mesh model design space is discretized to small elements and for each element OptiStruct calculated material properties. Material density of each element is directly used as a design variable. As a design constraints first natural frequency minimum value is defined and as a manufacturing constraint draw direction is defined. In the casting process cavities that are not open and lined up with the sliding direction of the die are not feasible. Draw direction constraint the material distribution in a structure so that the die can slide in that direction.

OptiStruct result is viewed with HyperView and Ossmooth tool is used to import conceptual design to CAD system which is part of OptiStruct package.

Aim to obtain smooth geometry and manufacturable design. The minimum requirements for performing a topology optimization are that a design variable to be set and an objective function to be defined. Mostly design variable is directly related with the design space which the objective function will be active. To know this basic but essential definitons will help us to describe our problem and our aim in other words what we are looking for and after that program will calculate according to our inputs and at the end engineering judment will have crucial impact.

TOPOLOJİ OPTİMİZASYON METODU VE OTOMOTİV BRAKETİNİN OPTİMİZASYONU

ÖZET

Son yıllarda otomotiv endüstrisinde rekabetin artmasıyla kaliteden taviz vermeden maliyet azaltabilme yöntemleri önem kazanmıştır. Maliyet azaltmaya en cok katkıyı da ürün dizaynı süresince harcanan zamanı ve kullanılan malzemeyi minimuma indirmek sağlar. Bilgisayar destekli analizle hızlı ve güvenilir sonuçlar elde edilir. İşte bu noktada optimizasyon yöntemi devreye girer. Modeli oluşturulmuş dizayn kriterleri tanımlanmış bir parçanın üretim aşamasına geçmeden bilgisayar analiz programları ile optimizasyonu yapılabilir.

Bu tezin amacı topoloji optimizasyon metodunu anlamak, yapılan çalışmaları incelemek ve otomotiv motor elemanı olan kompresör braketinin gerekli bilgisayar programlarını kullanarak optimizasyonunu yapmaktır.

Topoloji optimizasyonu genellikle dizayn sürecinin başlarında optimum malzeme dağılımı sağlamak amacıyla kullanılır. Amaç yer değiştirme, rijitlik, doğal frekans gibi kısıtları sağlayarak en uygun tasarımı bulabilmektir. Genellikle optimizasyon probleminin amacı çözüm fonksiyonunu maksimize ya da minimize etmektir. Örneğin ürün fiyatını minimize etmek ya da dayanıklılık fonksiyonunu maksimize etmek. Optimizasyon mühendislik problemlerinin çözümü haricinde günlük hayatta da kullandığımız bir araçtır. Yeni bir araç almaya karar verdiğimizde fiyat, motor performansı ve verimi gibi parametreler karşımıza çıkar. Her ne kadar ucuz, verimli ve güçlü motora sahip olan aracı seçmek istesek de her koşulda en iyiyi seçmek mümkün olmayacaktır. Böyle bir problemi çözerken birden çok kriter göz önünde bulundurulacak ve uygun olan seçilecektir (multiple criteria optimization).

Optimizasyon, boyut optimizasyonu, şekil optimizasyonu ve topolojik optimizasyon olmak üzere üç başlık altında incelenebilir. Boyut ve şekil optimizasyonunda genel amaç parça dayanımını artırmak ve gerilme yığılmasını azaltmaktır. Boyut optimizasyonunda yapının şekli bilinmektedir ve amaç parçaların boyutunu değiştirerek yapıyı optimize etmektir. Örnek olarak metal levhanın kalınlığı dizayn değişkeni olarak seçilebilir. Şekil optimizasyonunda ise yarıçap, eğiklik, kavisler değişebilir. Topoloji optimizasyonu ise belirlenen tasarım alanında en uygun elemanları seçerek sonuca ulaşmaktır. Topoloji optimizasyonundan çıkacak parçanın şekli önceden bilinmez. Bu tarz problemlerin çözümünde sonlu elemanlar yöntemiyle dizayn alanı küçük elemanlara ayrılır. Daha sonra bu elemanlardan hangilerinin dizayn için önemli olduğu belirlenir.

Topoloji optimizasyonunun amacı, esnekliği minimum (rijitliği maksimum) yapan ya da doğal frekansı maksimum yapan en iyi malzeme dağılımını bulmaktır. Topoloji optimizasyon problemlerinin çözümünde kullanılan homojenleştirme metodu 1988 yılında Bendsoe ve Kikuchi tarafından geliştirilmiştir.

Topoloji optimizasyon problemlerinin çözümünde yaygın olarak kullanılan diğer bir yöntem olan yoğunluk metodu (density method) ise R.J. Yang ve C.H. Cuhang tarafından 1993 yılında geliştirilmiştir. Bu metod literatürde malzeme dağılım metodu (material distribution method) olarak da adlandırılmaktadır.

Yoğunluk metodunda malzeme yoğunluğu 0 ile 1 arasında değişir ve optimizasyon probleminde tasarım değişkeni her elemanın yoğunluğudur.

İki boyutlu elemanlarda yoğunluk malzeme kalınlığına göre belirlenir. Üc boyutlu elemanlarda ise yoğunluk metodunu uygulamak o kadar kolay değildir. Yoğunluğu 0 olarak kabul edilen eleman tasarım alanından boşaltılır ve 1 olan elemanlar tasarım alanında kalır. Yoğunluğu 0 ile 1 arasında kalan elemanlara ise penalizasyon (cezalandırma) yöntemi uygulanır ve yoğunluk 0 veya 1 olmaya zorlanır.

Homojenleştirme metodunda parça mikroyapısal olarak kabul edilmekte ve bu yapı homojenleştirilmektedir. Mikroyapı malzeme içermeyen boşluklu, malzeme içeren dolu ve bir kısmı malzeme içeren bir kısmı içermeyen olmak üzere üç gruba ayrılır. Optimizasyon sürecinde mikroyapılar, katı ve boşluk arasında değişir. Eğer mevcut malzeme miktarı belli ise malzeme yapının bir kısmından diğer bir kısmına hareket edebilir. Bu nedenle yapıların topoloji tasarımı, belirlenen uygun yapısal alan içinde optimal malzeme dağılımının bulunması olarak ele alınabilir.

Optimizasyon yöntemleri hakkında bilgi verildikten sonra tezin ikinci aşamasında kullanılan bilgisayar programı ile ilgili bilgi verilmiştir. Bu tezde Altair Mühendislik firmasının yayınladığı HyperWorks programı kullanılmıştır. Parçanın sonlu elemanlar yöntemi ile mesh modelini oluşturmak için bu programın arayüzü olan HyperMesh, optimizasyon çalışmasını yapmak için de yine bu programın aracı olan OptiStruct kullanılmıştır. Optimizasyon sonucu ise HyperView'de görüntülenmiştir. OptiStruct programıyla statik, modal, termal olmak üzere birçok farklı analiz yapılabilir.

Optimizasyon terminolojisini kavramak tasarım problemini doğru tanımlayabilmek için gereklidir. Optimizasyon süresince dizayn edilebilir kısım ya da parçalar tasarım alanı olarak nitelendirilir. Tasarım alanı belirlenirken sistemin paket kısıtlamaları esas alınır. Tasarım değişkeni optimize edilecek parametrelerden oluşur. Tasarım alanı ve tasarım değişkenleri belirlendikten sonra amaç fonksiyonunun tanımlanması gerekir. Amaç fonksiyonu rijitliği artırmak, bir noktadaki gerilme yığılmasını azaltmak ya da doğal frekansı artırmak olabilir. Tasarımın kabul edilebilir olması için sağlanması gereken koşullara da tasarım kısıtları denir. Örneğin amaç fonsiyonumuz ağırlığı azaltmak ise kısıt emniyet gerilmesi olabilir. Ağırlık ancak emniyet gerilmesinin izin verdiği miktarda azaltılabilir.

Bu tezde optimizasyonu yapılan parça klima kompresörünü motora bağlamaya yarayan brakettir. Amaç braketin ilk doğal frekansını motorun çalışma frekansıyla çakışmaması için artırmaktır. Bu braket motor kayış kasnak sistemine ait bir parçadır. Kayış kasnak sisteminin amacı bütün sürüş koşullarında elemanlara (alternatör, kompresör..) güç iletimidir. Optimizasyonu yapılan braket dinamik yüklere maruz kaldığından sistemin rezonans frekansında titreşmesine sebep olabilir. Buna engel olmak için sistemin ilk doğal frekansının motor maksimum devrine göre belirlenen frekans değerinin üzerinde olması gerekir. Ele aldığımız motor maksimum devir sayısı 4200 devir olan dizel bir motordur, 4200/60'dan 70Hz, 4 silindirli bir motorda patlamalar 2 çevrimde bir olacağından 70x2=140 olur. Bu değere %30 emniyet eklediğimizde hedef değer 182 Hz olarak elde edilir. Bu demektir ki, motorun çalışma frekansıyla sistemlerin doğal freakansının çakışmaması için, motora bağlanan doğal frekansı 182 Hz'den büyük olmalıdır.

Yukarıda bahsedilen optimizasyon adımları brakete uygulanırsa öncelikle varolan tasarımın sorunu ifade edilmelidir. Bu ilk tasarımın bir önceki motor ailesinden olduğu gibi alınarak yeni motor tasarımının hedeflerini sağlaması amaçlanmıştır. İlk tasarımdan tez boyunca "base model" olarak bahsedilmiştir.

Base modelin frekans analizi yapıldığında ilk doğal frekansi 167 Hz çıkmıştır. Sonuç olarak 182 Hz olan hedefin altındadır. Bu duruma göre optimizasyon problemi ilk doğal frekansı 182 Hz'den büyük yapmak, bunu yaparken de kütle azaltmak olarak ifade edilmiştir. Base modelin ağırlığı 1000,5 gramdır.

Optimizasyonun amacı belirlendikten sonra tasarım alanı berlirlenir. Braketin bağlantı noktaları, parça eklenip çıkarılmasını engellemek amacıyla tasarım alanının dışında bırakılmalıdır. Tasarım alanı sonlu elemanlar yöntemiyle elemanlara ayrılır, yani mesh modeli oluşturulur. Mesh modelde ikinci dereceden tetrahedral elemanlar kullanılmıştır. Optimizasyon amaç fonksiyonu frekansı 182 Hz'in üzerine çıkarmaktır.

Braketin malzemesi AlSi9Cu'dur ve braket yüksek basınçla kalıba döküm yöntemiyle üretilir. Bu üretim yöntemiyle daha hafif alaşımlardan parça üretilerek ağırlık azaltılmasıyla araç perfomans ve yakıt tüketimine pozitif etki sağlanır. Optimize edilen braketin üretilebilir olması için OptiStruct programında kalıptan çıkış yönü üretim kısıtı olarak tanımlanmıştır.

HyperMesh programında yukarıda belirtilen amaç fonksiyonu ve kısıtlar belirlendikten sonra model OptiStruct'da çözdürülür. Sonuç HyperView'de değerlendirilir. Çıkan şekil frekans koşulunu sağlamaktadır. Ancak OptiStruct'tan çıkan parçanın bir design programı yardımıyla şeklinin düzeltilmesi gerekir. Konsept dizaynı CAD sistemine aktarabilmek için OptiStruct'da yer alan OssMooth aracı kullanılır. Buradaki amaç düzgün ve üretilebilir bir geometri elde etmektir. Parçanın yeniden dizayni için CATIA kullanılmıştır.

Optimizasyondan çıkan parçaya üretilebilecek bir şekil verildikten sonra bu en son tasarımın da hedefleri sağlayıp sağlamadığına bakılması gerekir. Yine en başta yapıldığı gibi bu en son tasarımın mesh modeli oluşturulur ve frekans analizi yapılır. Analiz sonucunda parçanın ilk doğal frekansı 186 Hz'e çıkmıştır. Aynı zamanda parçanın kütlesi de 972 gramdır ve ilk dizayna göre 28 gr azalmıştır.

Sonuç olarak hedef frekans değeri sağlanmış ve bunun yanında kütle azaltılmıştır. Bu çalışmada braket modeli oluşturulurken sistem etkileşimlerinin de analize katılarak gerçeğe yakın sonuçların elde edilebilmesi için motorun mesh modeli oluşturulmuş ve braket motora bağlandığı gibi pozisyonlanarak analize sokulmuştur. Optimizasyon brakette yapılacağı için braket dışındaki diğer motor parçaları tasarım alanının dışında tanımlanmıştır (non-design space). En son elde edilen tasarım, projenin ilerleyen safhasında araç dayanıklılık testinde kullanılacak, testi tamamladığında seri üretim onayı almış olacaktır.

Elde edilen son tasarımın kullanılabilirliğini görmek amacıyla yine HyperWorks programı kullanılarak gerilme analizi yapılmıştır. Gerilme analizinde kuvvet girdisi olarak brakete maksimum yükleme durumunda gelen kuvvetler ve yerçekimi kuvveti dikkate alınmıştır. Maksimum gerilme değeri maksimum yükleme durumunda malzeme akma değerinden %60 küçük olmalıdır. Malzeme AlSi9Cu'nun akma dayanımı 130 Mpa olduğundan analiz sonucunda gerilmeler 78 Mpa'dan küçük olmalıdır. Analiz sonucunda bulunan maksimum değer 51 Mpa'dır, hedefi sağlamaktadır.

1. INTRODUCTION

In today´s market engineering business is getting more and more competitive industry. Manufacturers need to find solutions to reduce cost without compromising quality and performance. With computer aided design faster and reliable solutions can be obtained. Using optimization software results efficient design of products especially where we need high performance and cost saving study such as automobile, aeronautic, electronics industry.

Topology optimization is often used early in the design process to define the optimum part layout and this method provide optimal design for given design spaces, given loads and boundary conditions.

In this study topology optimization methods will be described and such a trial case automotive bracket will be optimized by using software applications.

1.1 Thesis Purpose

The purpose of this thesis is to understand what optimization is and provide information about performing optimization tool software by optimization of FEAD (Front End Accessory Drive) compressor bracket.

1.2 Scope

The scope of thesis will be basic theory of topology optimization and application by using HyperWorks OptiStruct tool. The mathematics behind the different optimization method is not investigated in detail. System model and material conditions are obtained from FEAD system engineering team. Derivation of them are not scope of this study.

1.3 Literature Survey

There are numerous resources related with optimization. In this section similar samples with this study will be summarized.

Optimization of Front Axle Mounts Vibration Characteristics for Driveline Integration with Chassis (GM,2012) is done by Vasudeva Rao and Soundararajan.

A driveline is used to transfer torque and speed from engine to wheel and bracket attachments designed to get the dynamic stiffness which can provide vibration isolation form vehicle structure. Modal analysis is carried out to determine the natural frequencies of bracket. First mode natural frequency is 520 Hz which must be higher than 600 Hz. Objective function is defined as to maximize the frequency by introducing stiffeners at more strain energy location to the bracket.

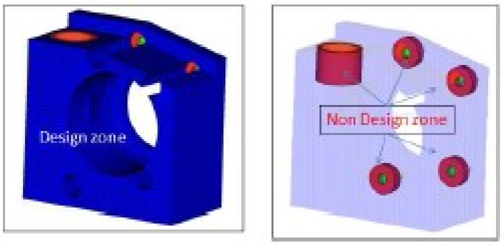


Figure 1.1: Design and non-design zone of driveline bracket.

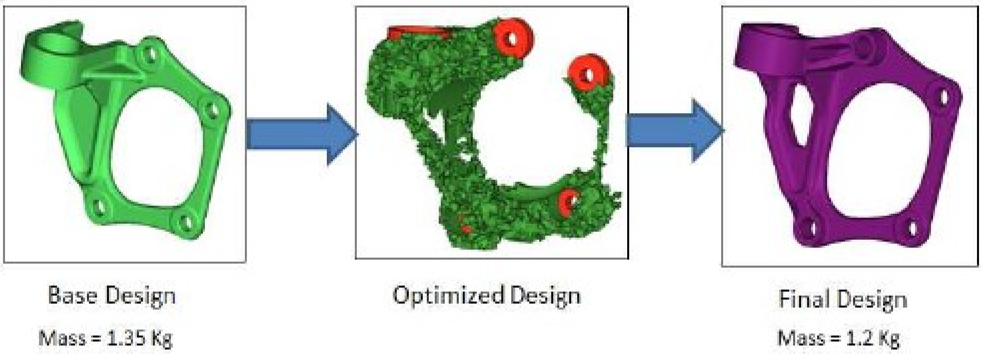


Figure 1.2: Optimization results of the driveline bracket.

First bending mode is improved %21 which is above 600 Hz. (Rao & Saundarajan, 2012)

B. Sreedhar and U.Naga Sasidhar (Hyundai,2012) studied on topology optimization of engine mounting bracket.

The engine mount sistem includes an engine, three to four engine mounts and a foundation. Engine need to be properly constrained and isolated due to the vibration issues.

The design space of the mounting bracket is available space which does not interfere with any surrounding components when being held in steady or work condition. The aim of the topology optimization is to minimize the volume without affecting the bracket stiffness and strength compared to base bracket.

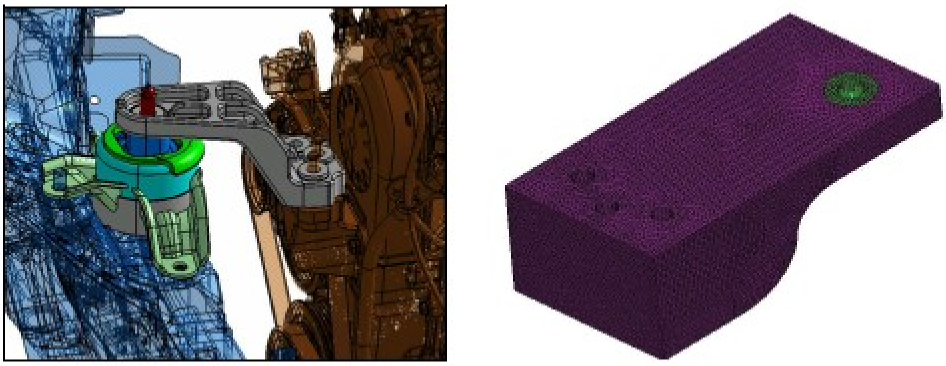


Figure 1.3: Design space of engine mounting bracket.

Constraints are from durability point of view is stress limit value and from manufacturing point of view is single draw direction. At the end of the process unnecessary material removed from the design space.

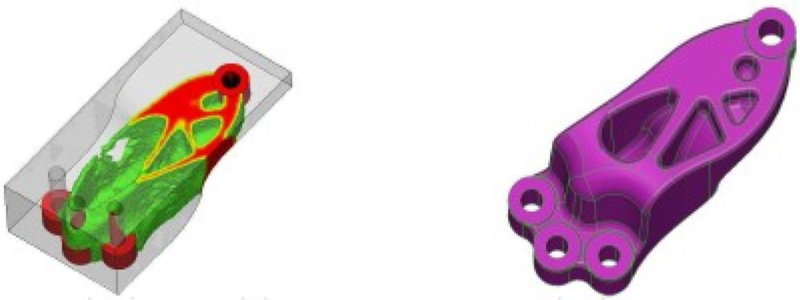


Figure 1.4: Optimization output and optimization based CAD model.

Comparing with original design of mounting bracket, the mass on the final design is reduced by 20% and it is strong enough in fatigue and strength with improved frequency and stiffness values (Sreedhar & Sasidhar, 2012).

Claudia Raffaldini`s (University of Parma) thesis is structural optimization in the automotive industry. She examined engine alternator bracket and optimized according to system design targets. This bracket used to connect the alternator to the engine.

Bracket is subjected to dynamic loading that could make the system oscillate at its resonance frequencies. To avoid system`s natural frequency of the normal mode must be increased over 250 Hz. This is the aim of the optimization. First of all modal analysis need to be conducted to see existing bracket`s natural frequency value.

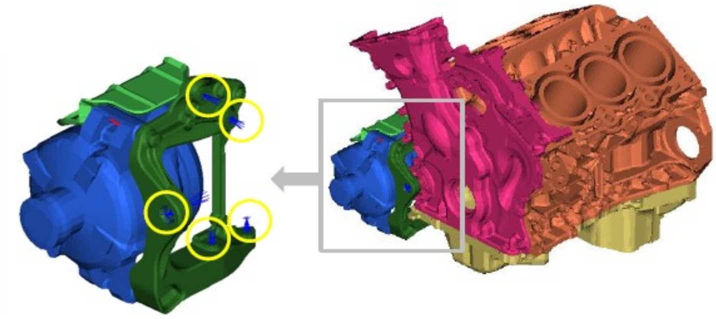


Figure 1.5: Alternator bracket and its fixing points to the engine.

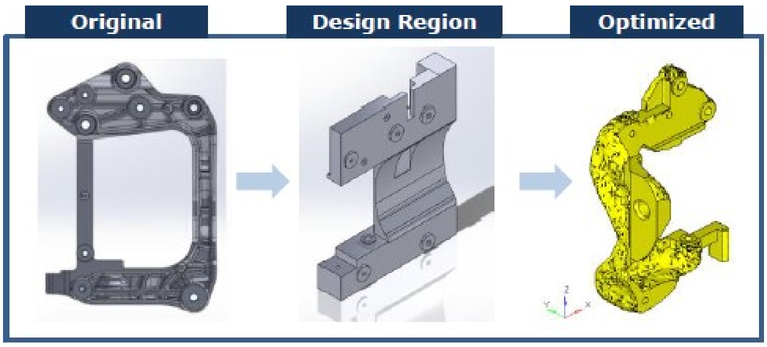


Figure 1.6: Bracket design changes during the optimization process.

Optimization result is reconstructed with OSSmooth tool and modified by CAD program. System natural frequency increased from 244 Hz to 273 Hz (Raffaldini, 2011).

From above 3 samples it can be seen that topology optmization process is need to be well explained before any analysis run. At first the issued part need to be considered as design space where optimization will be realized. In addition it can be seen that analysis result need to be re-shaped to have manufacturable design and then analysis need to be re-run to see if the final design is at the safe side from frequency, durability or stress targets point of view.

2. OPTIMIZATION THEORY

Optimization is automatic process to make a system or component as good as possible based on an objective function, given design space and constraints. Topology optimization aims to find best use of material by achieving requirements such as stiffness, eigenvalues, displacement (Bendsøe, Optimization of Structural Topology, Shape and Material, 1995). The objective of optimization problem is generally maximization or minimization of function. For example minimization of cost, time or maximization of eigenvalues. As an example cost function can be considered. Minimization of cost function can be shown as: (Rao., 1996)

$$\text{Find x } \boldsymbol{x} = \begin{Bmatrix} x1 \\ x2 \\ . \\ . \\ . \\ xn \end{Bmatrix} \text{ which minimizes } \boldsymbol{f}(\boldsymbol{x}) \text{ subject to } \begin{cases} gi(\boldsymbol{x}) \le 0 \;, i = 1,2,\ldots.,m \\ hj(\mathbf{x}) = 0 \;, j = 1,2,\ldots..,n \end{cases} \quad \textbf{(2.1)}$$

where x is the vector of design parameters and f(x) is the cost function. The functions gi(x) and hj(x) are called the inequality constraint function and the equality constraint function respectively and they define the constraints of the problem. This is called as a constrained optimization problem.

2.1 Multiple Criteria Optimization

For most of studies we need to consider multiple objectives. Ehrgott says optimization problem is to choose among set of alternatives an optimal one where optimality refers to certain criteria, according to which the quality of the alternatives is measured. When man decide to buy a new car decision will be made according to price, engine efficiency and power (Ehrgott, 2005). Cheap, efficient and powerful model will be choosen. But it is not possible to find the best from every aspect.

As an example we consider mathematical problem with two criteria and one decision variable.

Example 2.1 Funtions are

$$f_1(x) = \sqrt{(x+1)}$$
$$f_2(x) = x^2 - 4x + 5 = (x-2)^2 + 1 \qquad \textbf{(2.2)}$$

And we want to minimize both over the nonnegative real line :

$$\min_{x \geq 0} \left(f_1(x), f_2(x) \right). \qquad \textbf{(2.3)}$$

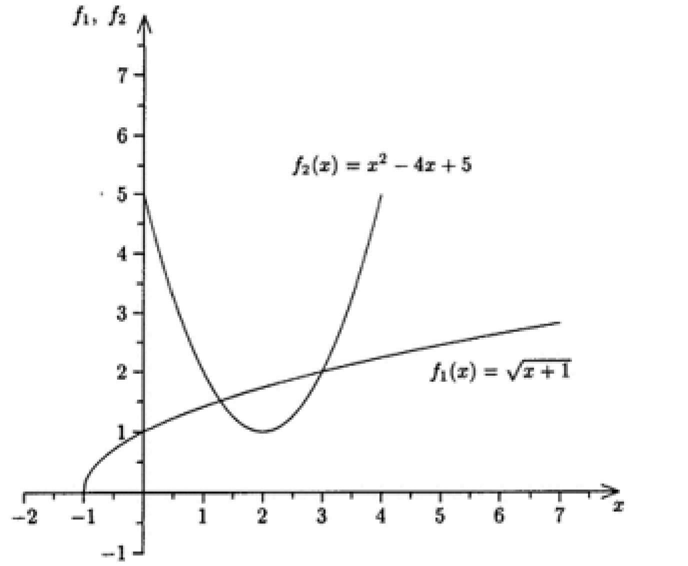


Figure 2.1: Model problem graphical sample.

The question is what are the minima and the minimizers in this situation? Note that again, for each function individually the corresponding optimization problem is easy : x_1=0 and x_2=2 are the unique minimizers for f_1 and f_2.

2.2 Structural Optimization

For structural optimization aim is to find optimal material distribution according to given design domain or space. Some common functions are to minimize mass, displacement or maximize eigenvalues. By structural optimization new holes, voids can be added to structure. Structural optimization can be divided to three: size optimization, shape optimization and topology optimization (Klarbring & Christensen, 2009).

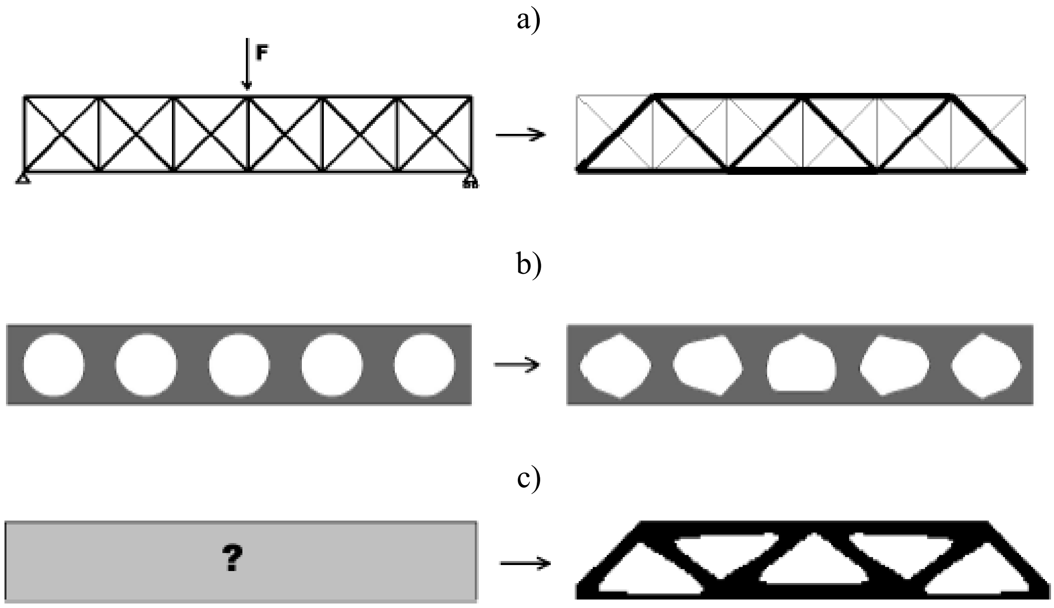


Figure 2.2: Different types of optimization a) size optimization b) shape optimization c) topology optimization.

2.2.1 Sizing optimization

For sizing and shape optimization generally main concern is increasing strength and reducing stress concentrations. For sizing optimization shape of the structure is known and the aim is to optimize structure by adjusting sizes of the components. For example, thickness of sheet metal can be design variable in order to find optimum solution with regards to weight, stress, displacement (Klarbring & Christensen, 2009).

2.2.2 Shape optimization

With sizing optimization topology of the structure is known. In shape optimization fillets, chamfers, radius can be changed. The optimization algorithms will not include removed holes, it is to adjust the ones specified in the analysis.

2.2.3 Topology optimization

Topology optimization is selecting the best elements in a given design domain or space that maximize the use of material. By assigning a valid design space and the boundary conditions like loads computer aided software are able to predict the optimal structure. From a given design space the purpose is to find optimum distribution of materials.

With topology optimization resulting shape or topology is not known. To solve this kind of optimization problem finite element method (FEM) is used by dividing design space to discrete elements.

Then to find which are these elements are substantial which are not. This method called SIMP (Single Isotropic Material with Penalization) (Rozvany, 2000). There are two main solution ways to solve optimization problem with SIMP, density and the homogenization method.

2.2.4 Density method

For density method material density is between zero and one. The design variable of the optimization study is density. If we consider elements in 2D it is possible to represent density as a changing thickness of plate. But for the 3D it is not easy to determine design variable from density point of view. Elements should be seperated as solid or empty. For intermediate densities penalization need to be implemented.

Without penalty the relation stiffness–material cost is linear, $E = \rho E^0$, where E is the elasticity tensor and ρ is the density, $0 \leq \rho \leq 1$. One popular method to achieve penalized intermediate densities is by letting the stiffness of the material be expressed as $E = \rho^p E^0$ (Bendsøe, Optimization of Structural Topology, Shape and Material, 1995).

If element´s density is 0 this element is removed and if it is 1 this element will stay in design domain. If it is neither 0 nor 1 density is penalized and forced to be 0 or 1.

Where p is a penalization factor greater than zero, typically 2 – 5. In literature the density method together with this penalization is often called the SIMP method (Solid Isotropic Microstructures with Penalization) (Rozvany, 2000).

2.2.5 Homogenization method

This method first introduced by Kikuchi and Bendsoe. The idea is to assume material density as a microstructure. The microstructure is a composite material with an infinite number of small voids (Bendsøe, Optimization of Structural Topology, Shape and Material, 1995). This leads to composite that can have a density between %0 and %100. Microstructure type can be square holes as showed in example below.

For a layered microstructure the elasticity can be found analytically, but for most other types of microstructures the elasticity needs to be calculated numerically by using the finite element method for different sizes and then interpolating between these values (Bendsøe & Kikuchi, Generating optimal topologies in structural design using a homogenization method, 1988).

The microstructures do by themselves provide some penalization on intermediate densities but this is most often not enough and some additional penalization need to be introduced (Rozvany, 2000).

The optimization is then carried out similarly to the density method. The problem is discretized into finite elements with the design variables (hole sizes and rotation) assumed to be constant over each element.

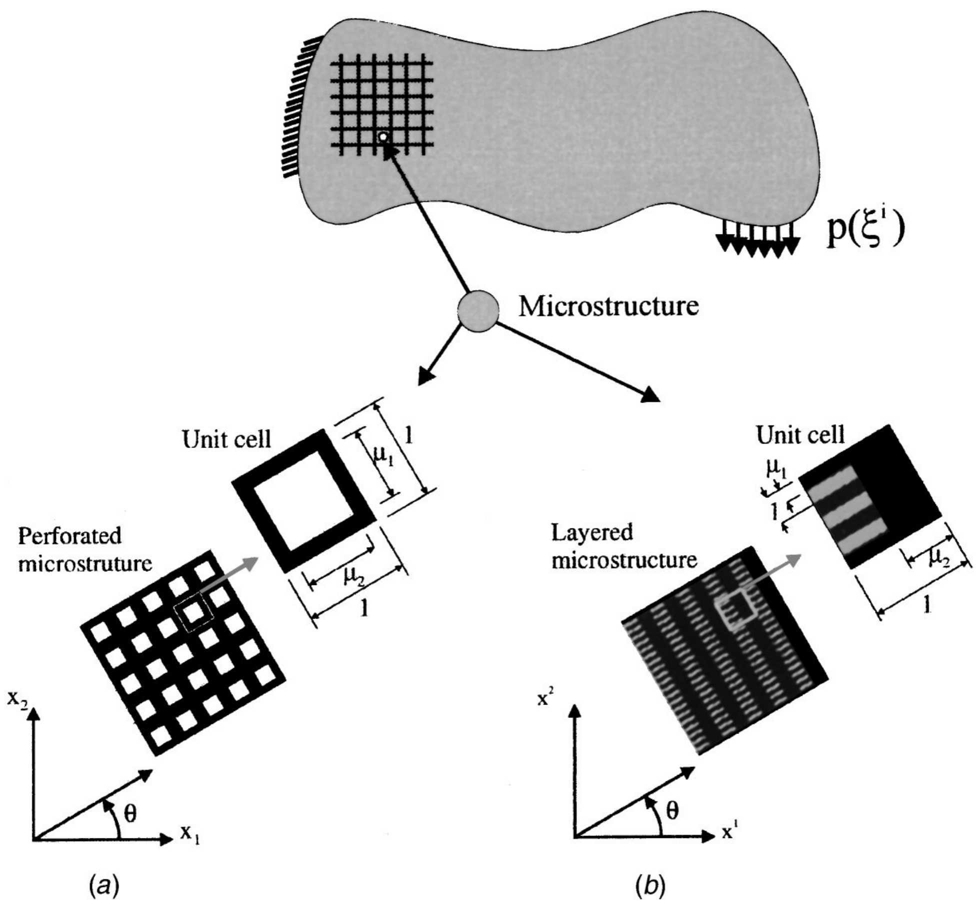


Figure 2.3 : Examples of microstructures with rotation in 2D: a) Microstructure with rectangular holes and b) Layered microstructure. (Hans, 2001)

Layered microstructures consists of unit cells. This unit cells` position angle, shape and its perforated structure are important to define elasticity module.
In this thesis subjected bracket material isotopic. OptiStruc uses density method for isotropic materials and homogenization method for anisotropic materials.

2.3 Numerical Problems in Topology Optimization

The topology optimization method has reached a level of maturity where it is being applied to many industrial problems and it has widespread academic use. Despite its level of matureness, there still exist a number of problems concernig checkerboard, mesh-dependence and local minumum issues. (O.Sigmund, J.Petersson , 1998)
Checkerboard consists of optimized results with black and white patterns. At first it was believed that this kind of regions are optimal microstructure but then it was seen that checkerboard patterns are shows unrealistic stiffness value.

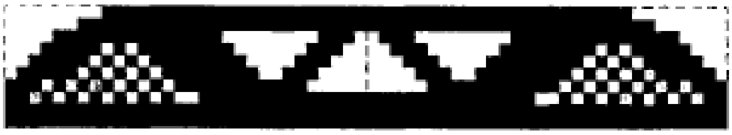


Figure 2.4: Checkerboard patterns on optimized structure.

By using higher order elements for finite element modelling checkerboard effect can be prevented.
Mesh-dependence is can be seen from below figure. First problem is solved by using 600 finite elements and then with 5400 elements. With higher element number much more detailed structure is obtained.

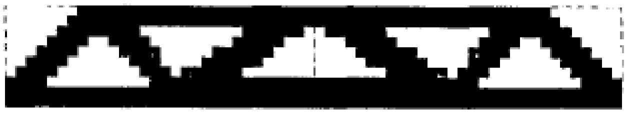


Figure 2.5: Solution by using 600 elements.

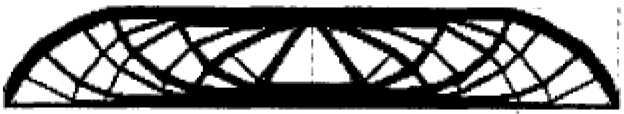


Figure 2.6: Solution by using 5400 elements.

In OptiStruct checkerboarding is controlled by the methods applied for the future when MINDIM command is activated. MINDIM is should be at least three times greater than the average element size. And if manufacturing containts (chapter 5) are applied, minimum member size control is always activated.
In OptiStruct gradient-based optimization method is used. When the problem is convex there will a one solution which is global minimum.

But in reality most of the engineering problems are consist of non-convex issues. When solving non-convex issue local minimum solutions can be obtained while seeking for global optimum solution.

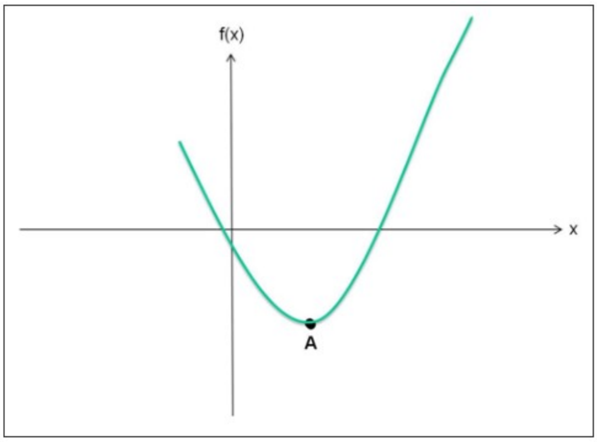


Figure 2.7: Convex problem, global mininum solution.

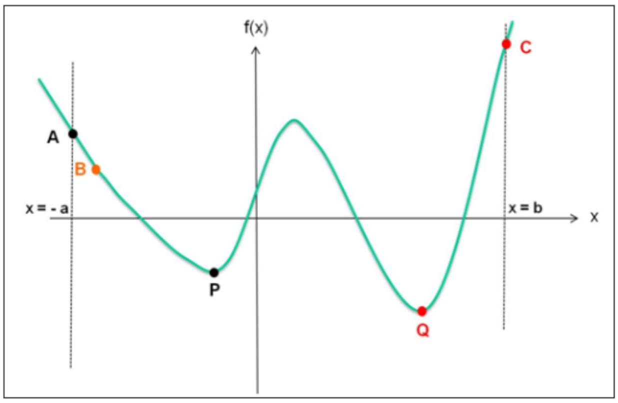


Figure 2.8: Non-convex problem, local and global mininum solutions.

For non-convex problem iteration beginning points will effect the solution. If iteration starts from point A solution will be point P which is local minimum solution. With this, penalization factor will increase step by step for each iteration thus 0-1 result is not acquired in one step. This kind of approach will increase the chance to get gloabal minimum solutions.

In OptiStruct with multiple starting point optimization choice convergence to global optimization can be increased but it does not guarantee that obtained solution is 100% gloabal minimum. However, it has strong effect to find global solution.

3. SOFTWARE TOOL REVIEW

For this thesis OptiStruct 12.0 from Altair Engineering software is used to solve optimization problem. To set up the model HyperMesh is used which is also released by Altair Engineering. HyperMesh is used to mesh design space and set up the problem to be solved by setting boundary conditions. Then data file from HyperMesh to OptiStruct is exported. And the result is evaluated by using HyperView.

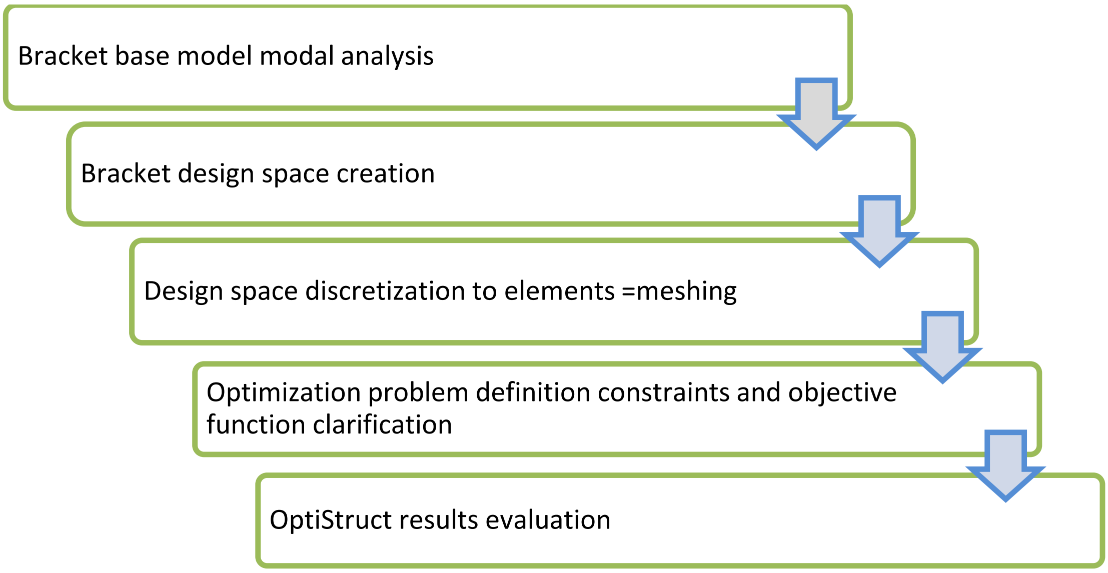


Figure 3.1 : Optimization process workflow.

3.1 OptiStruct

OptiStruct is the solver for structural optimization. It is started out as a research code at university in 1991. The only problem solved was the minimization of weighted compliance and eigenfrequencies using the homogenization method. OptiStruct do not have any graphical interface, the full problem formulation is made using HyperMesh and any other options are supplied via the command line. OptiStruct is capable of performing a range of different finite element analysis including static, modal, buckling and thermal analysis.

Different types of loads such as point forces, pressure, gravitational loads, thermal loads, etc., can be applied (Inc A.E., 2013).

OptiStruct capabilities are sorted below.

Analysis Types Include:

- Linear and Non-static Analysis
- Nonlinear Implicit Quasi-static Analysis
- Normal Modes Analysis for Real and Complex Eigenvalues
- Linear Buckling Analysis
- Direct and Modal Frequency Response Analysis
- Random Response Analysis
- Linear Direct and Modal Transient Analysis
- Coupled Fluid-structure (NVH) Analysis
- Linear Steady-state and Transient Heat Transfer Coupled with Static Analysis

Optistruct provides comprehensive solutions to simulate stiffness, strength and stability, noise and vibration, powertrain durability, thermal analysis, kinematics and dynamics, and more. Optistruct's award-winning design-synthesis technology uses the topology optimization approach to generate innovative concept-design proposals. In the initial phase of the development process, the user enters the package space information, design targets and manufacturing process parameters. OptiStruct then generates a manufacturable design proposal that is optimized for the given design targets. The manufacturing process parameters are important in achieving interpretable, feasible designs. In sheet metal parts, beads are often used to reinforce structures (Inc. A.E., 2013).

3.1.1 Topology optimization with OptiStruct

As indicated before topology optimization generates optimized material distribution for a set of loads and contraints within a given design space. The classical topology optimization set up solving the minimum compliance problem, as well as the dual formulation with multiple constraints are available. Constraints on Von Mises stress and buckling factor are available with limitations. Manufacturing constraints can be imposed using a minimum member size constraints, draw direction constrints, extrusion constraints, symmetry planes. (Inc. A.E., 2013)

OptiStruct uses an iterative procedure known as the local approximation method to solve the optimization problem.

This method determines the solution of the optimization problem using convergence test, response screening to retain potentially active responses, design sensivity analysis for retained responses. To achieve a stable convergence, desgin variable changes during each iteration are limited to a narrow range within their bounds, called move limits. The biggest design variable changes occur within the first few iterations and, due to an advanced formulation and other stabilizing measures, convergence for practical application is typically reached with only a small number of FE analysis.

3.1.2 Software usage

As a first step design problem need to be defined. Optimization statement can be find by asking: what is our criterion for ``best`` design , what are the available means, how do we describe different designs. To be familiar with optimization terminology design space, design variables, objective function, constraint function need to be defined.

Design space : Selected parts which are designable during optimization process. It can be based on the existing system packaging constraints.

Design variables : System parameters that are varied to optimize system performance that affects the value of the objective function. Design variables defines which elements subject for optimization. The design variable is created by using dialog box in HyperMesh. Solid or shell elements can be used as design elements.

Objective function : Any response function of the system to be optimized. The response is a function of the design variables.The function that will be minimized, maximized or that will reach a target value. Examples of objective functions can be to maximize the stiffness in a structure or to minimize the stress at a certain point in a structure.

Constraint functions: Bounds on response functions of the system that need to be satisfied for the design to be acceptable.

Constraint function is derivated from one of the response functions. Manufacturing constraint will be defined in section 5 in detailed.

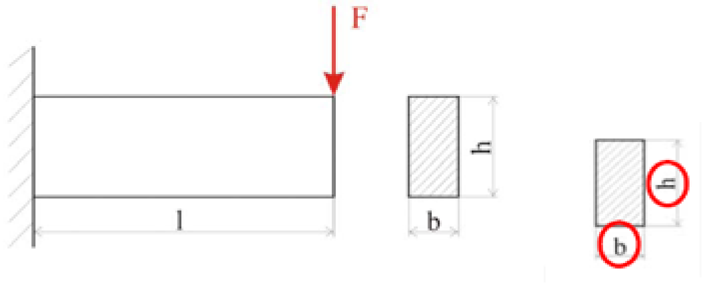


Figure 3.2 : Beam optimization problem.

20< b < 40   30 < h < 90

$\min Weight(b,h) \rightarrow ObjectiveFunction$

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

$\tau(b,h) \leq 15\,Mpa...h \geq 2b \rightarrow ConstraintFunction$

To perform optimization with OptiStruct, FE model need to be created. HyperMesh is used to meshed the model. In FE modeling boundary conditions, load cases need to be set up.

Different type of meshes should be used to get more reliable model. If the objective function is to maximize the eigenfrequencies a load step that calculates the desired eigenfrequencies must be defined.

For size and shape optimization it is sufficient to mesh existing component only, for topology optimization whole design space need to be meshed.

Optimization objective: Contsraints and objective defined similar way in OptiStruct. At first response need to be created. Response can be used either as an objective function or as a constraint. In the following sections optimization terminology will be deeply presented by on the basis of engine accessory bracket case study. Two responses will be defined on the program such as frequency and volume fraction restriction. One of them will be defined as a objective function and the other as a constraint fuction.

At section 1.3 Literature Survey samples for objective function and constraint fucntion can be reviewed which were gathered from similar studies with this thesis.

Before we start optimization we ensure that FE model is available and design and non-design space of elements are defined.

4. ENGINE ACCESSORY DRIVE SYSTEM

In this section A/C (air condition) compressor bracket which is part of engine accessory drive system will be optimized to achieve first natural frequency target and mass reduction. At first some basic knowledge about engine accessory drive system will be explained.

4.1 Engine Accessory Drive System Design Basics

Front End Accessory Drive system objective is to transmit power to turn the various accessories with maximum efficiency under all driving conditions without the driver knowing the job is being done.

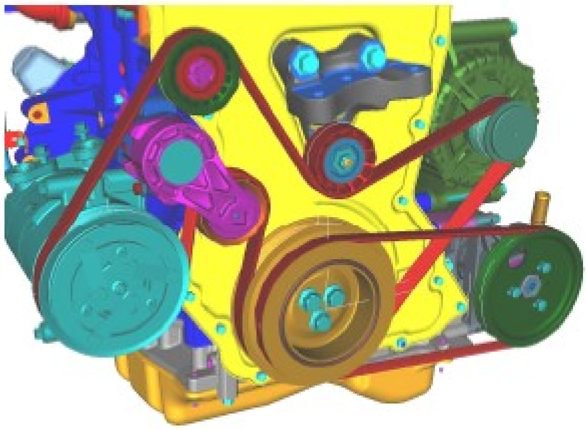


Figure 4.1 : Engine accessory drive system (engine front view).

System should have minimum friction and capable of operating in extreme environments (temperature, dust, oil). No NVH (noise, vibration, harshness) issues due to belt noise should be seen and naturally it must be durable. Requiring minimum maintenance.

FEAD system parts are : Tensioner, Belt, Pulleys, Brackets, Hardware and they have interaction with Alternator, AC (Air Conditioner) Compressor, Crank Pulley, Cylinder Head, Cylinder Block.

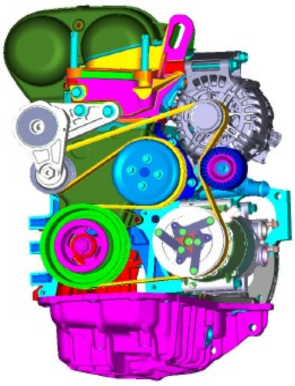


Figure 4.2 : Engine front end accessory drive system (side view).

To determine the natural frequencies and hub loads of accessory drive system some CAE analysis are required. To perform CAE analysis real test data is required such as customer usage profile. It means of quantifying customer frequency of use, customer imposed loads on the system. It covers expected product life and use for hub load histograms for accessory design and key life testing of components. (Data derived from in the Public Road Vehicle Database shows percentage time spent at different engine speeds over a number of driving cycles for a typical passenger car and percentage of time spent at different engine loads).

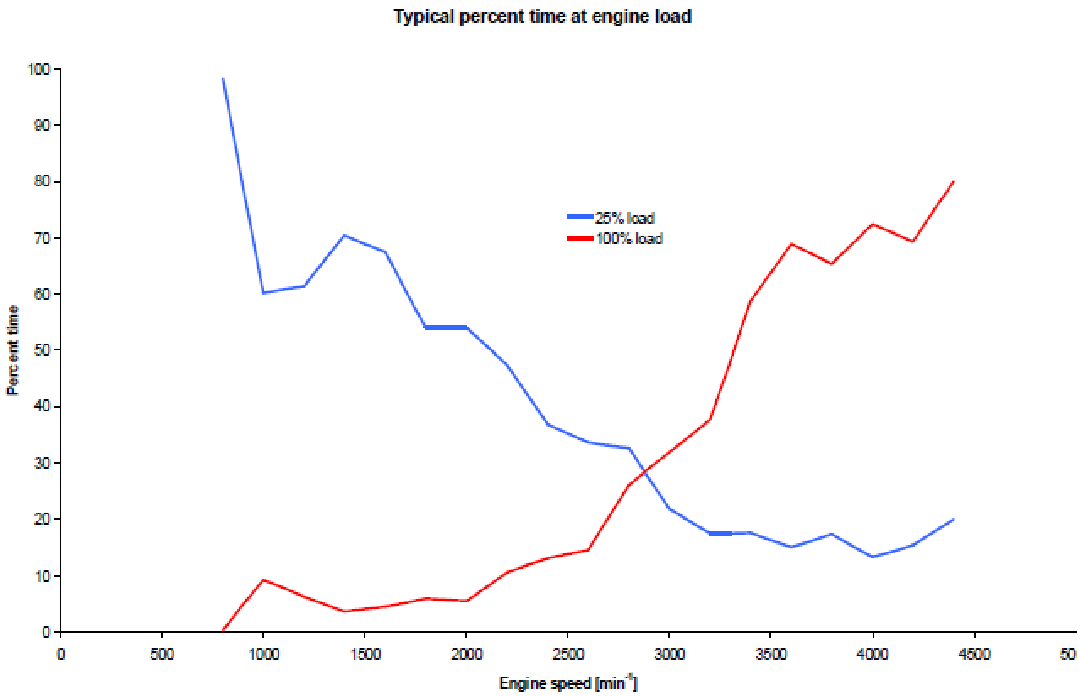


Figure 4.3 : The graph is the average of all driving styles. (shows that city drivers will spend high percentage of their time at idle speed).

4.1.1 Accessory design target

The design target requires that first natural frequency of the accessory assemble is 30% above the engine maximum firing order frequency. For example if the maximum engine rpm is 6000; 6000 rpm/ 60x2x1.2=240 Hz. Hence frequency target for first natural frequency should be higher than 240Hz.

Good bracket design uses a closed box structure which gives maximum stiffness plus good torsional stiffness.

Typical cast designs are hollowed from the rear, with cross-bracing ribs, usually to connect block attachment points to accessory mounting locations to provide a direct load path.

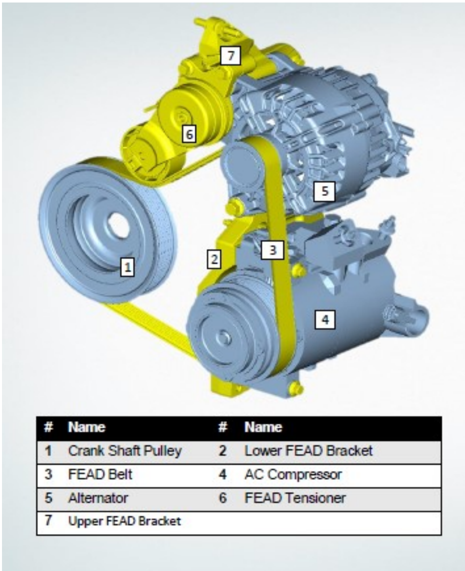


Figure 4.4 : Front end accessory drive system parts.

Bracket material is AlSi9Cu which is universal alloy with very good castability suitable for pressure die casting.

High pressure die-cast components reduces vehicle weight so it has positive impact on performance and fuel efficiency. Die casting is the process of forcing molten metal under high pressure into the cavities of reusable steel molds. Upon solidification, the liquid metal takes the shape of the inner surface of the die (Andresen, 2005). The high-pressure die casting (HPDC) process is widely used for manufacturing various automotive thin-walled components for structural applications with wall thicknesses typically ranging from 1 mm to 8 mm.

The ability to hold close dimensional control, cast complex shapes with fairly uniform microstructure, and minimize secondary machining operations, makes the HPDC process competitive for many high volume applications.

Advantages of cast components include reduction in vehicle weight, thin walls with intricate shapes, increased design flexibility, complex geometry, part integration, reduced tooling investment, and improved dimensional control and accuracy, and smooth surface finish (1—2.5 μm); in many cases, post-machining can be totally eliminated.

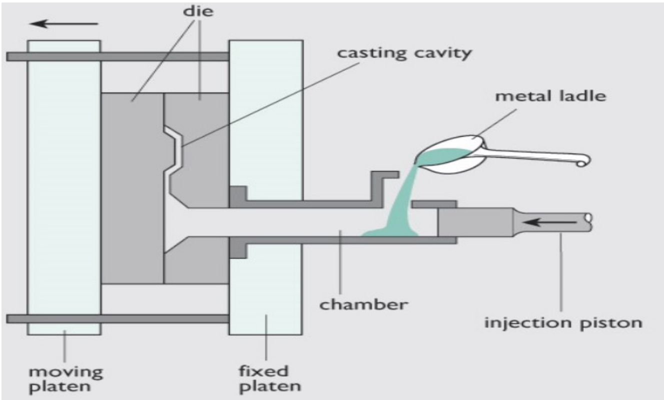


Figure 4.5 : High pressure die casting (Learneasy, 2015).

Aluminum casting temperature is 650 ºC. Pure Aluminum shrinks too much, and suffers hot cracking. It is alloyed with Silicon, which increases melt fluidity, reduces machinability. Silicon 9%, Copper 3.5%. Silicon increases the melt fluidity, reduces machinability, Copper increases hardness and reduces the ductility. High silicon alloy is used in automotive engines for cylinder castings - e.g.17% Silicon for high wear resistance.

5. BRACKET TOPOLOGY OPTIMIZATION

A/C compressor bracket subjected to dynamic loading that could make the system oscillate at its resonance frequencies. To avoid this, system´s natural frequency of the first mode must be increased over an threshold which is calculated according to maximum engine rpm and possible to reducing weight of the part by assigning volume fraction factor to design space.

Maximum engine rpm is 4200. 4200/60x2x1.3 = 182 Hz hence ;

Target > 182Hz

Constraint Mass ≤ Initial Mass (not more than defined volume fraction)

Table 5.1 : Material properties of AlSi9Cu.

Density	$2.7e^{-09}$ ton/mm^3
Elastic modulus	70000 Mpa
Poisson`s ratio	0.29

According to this objective function should be to reach maximum first natural frequency while decreasing the volume. Constraint is not to remove more than 50% of design space volume. While defining objective function frequency value is not defined as aim is to maximize it as much as until to reach constraints.

In addition to natural frequency target well designated bracket have to achieve maximum stress target. Maximum Von-misses stresses under maximum hub load or thirty times under gravity force (30g) loading must be lower than %60 of the material`s yield value. As yield values of AlSi9Cu is 130MPa,Von-Misses stresses should be lower than 78 Mpa. Design targets are summarized at below table.

Table 5.2: Bracket design targets.

First Natural Frequency	ωRI > 182 Hz
Maximum Von-Misses Stress	σmax < 78 MPa

In this section optimization process is presented step by step beginnig with desgin space creation and ending with latest design part .

5.1 Natural Frequencies and Undamped Multiple d.o.f. Free Vibration

Free vibration means that load which affects to part is zero. System`s natural frequency need to be seperated from its under dynamic loads frequency. For natural frequency analysis of automotive bracket damping effect is assumed as zero. If there is no damping motion will be harmonic and described by $u = u`sin\omega t$ where $u`$ is the amplitude of vibration and ω is its circular frequency. The cyclic frequency is $\mathbf{f = \omega/2\pi}$.

In general, natural frequencies are independent of amplitude if conditions remain linearly elastic, gaps do not open or close, and amplitudes are small in comparison with structural dimensions so that geometric nonlinearity does not appear. Without damping, all d.o.f. move in phase with one another and at the same frequency ω. (Rober D. Cook, David S. Malkus, Michael E. Plesha, Robert J. Witt , 2002).

Engine orders are simply the amplitudes of the frequency components which are multiples of the rotating frequency and determined by an order analysis are extensively used in the vibration work to identify source of excitation. In four cylindered in-line engine have its second order component as the dominant excitation.

In OptiStruct, normal modes analysis is performed by Lanczos method. Lanczos method has the advantage that eigenvalues and associated mode shapes are calculated exactly. This method is efficient for calculations in which the number of modes is small and the full shape of each mode is required. The disadvantage of the Lanczos method is that it is slow for large problems with millions of degrees of freedom for which hundreds of modes are required (Inc. A.E., 2013).

5.2 Bracket Topology Optimization Step 1: Natural Frequency Analysis with Base Part

The existing bracket which will be re-designed according to analysis results will be called as “base model”. In OptiStruct, to set up normal mode analysis EIGRL load collector card need to be defined. Details will be given in following steps. System is modelled with higher order 3D tetra elements, beams and rigids. Cylinder head upper surface is fully constrained.

When part of structue is very stiff relative to and lightly coupled to, the structue under investigation, it is candidate for application of rigid elemenents. In this study engine block constrained and it will be considered rigid as its reflections have little effect on bracket. To calculate natural frequency of bracket with the parts which bracket is fixed is important due to rigidity effect of whole these parts. If the analysis is conducted without engine block higher frequency would be acquired which is not realistic.

OptiStruct set up phase of natural frequency analysis are mentioned at the following steps as the aim of first step to see the current natural frequency of the bracket.

According to analysis results first natural frequency of bracket is 167 Hz which is below the frequency target. Thus first target is can be stated as optimize the bracket design to get more than 182 Hz frequency.

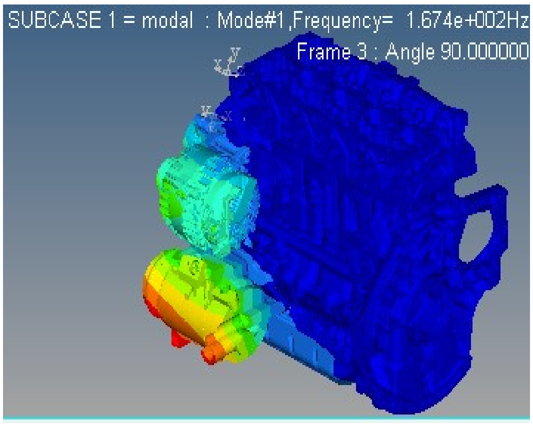


Figure 5.1 : Baseline model first natural frequency is 167 Hz.

5.3 Bracket Topology Optimization Step 2: Desgin Space Creation of Bracket Base Part

To define design space package restrictions, clearence requirements and bracket fixing points need to be considered.

Regarding to this considerations design space is created to have maximum volume. Cavities due to the base model`s rib patterned design is fillep up. Mass is increased from 1000.5 gramms to 1690 gramms. For design space creation CATIA V5 is used. To check package restrictions bracket position in the engine need to be considered. All parts around the bracket should be checked from clearence point of view.

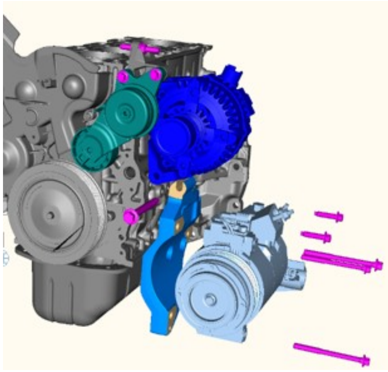
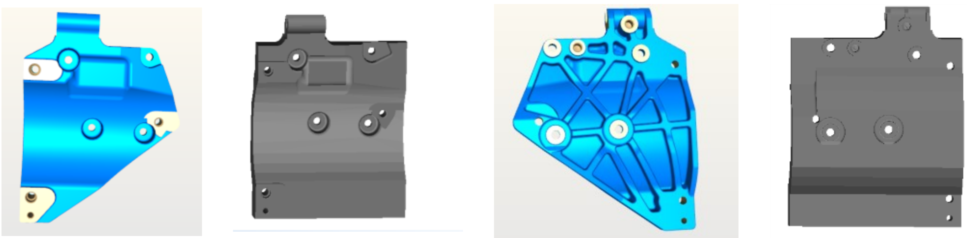


Figure 5.2 : Bracket position and its fixing points.



(a) (b) (c) (d)

Figure 5.3: a) Base model front side view (b) design space front side view (c) base model rear side view (d) design space rear side view.

5.4 Bracket Topology Optimization Step 3: FE Model Creation with HyperMesh

For finite element model of design space to minimize numerical problems (section 2.3) 3D tetrahedral elements are used. Bolt connections are modelled with rbe2 and cbar elements. Rbe2 elements provide rigid link to transfer motion from the independent node to dependent nodes.

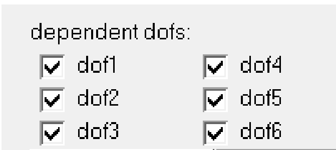


Figure 5.4: Hyperworks screenshot- rbe2 element degree of freedom contraints (dof1,2,3 tranlsational dof 4,5,6 rotational motion).

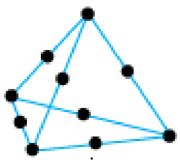


Figure 5.5: Second order 3D tetraheral element (TETRA10).

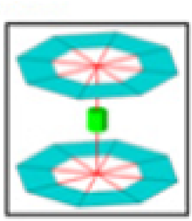


Figure 5.6: Bolt connections are modelled with rbe2 anc cbar elements. Design space modelled with 140416 nodes and 710480 elements.

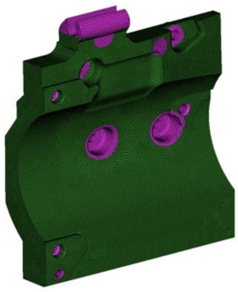


Figure 5.7: FE model of design space.

5.5 Bracket Topology Optimization Step 4: Non-Design Space, Design Space and Design Variable Definition

The bracket fixing connections to engine and to compressor need to be removed from design space and defined as a different component with same material properties. The aim of non-design space creation is to prevent fixing connections being lost or re-designed. As non-design space is new component it has been shown with different colour. Each element of design space area is defined as design variations.

As mentioned before desgin variations will have direct effect on response functions which are objective and constraints functions.

Figure 5.8: HyperMesh screenshot to creat design and non-design space.

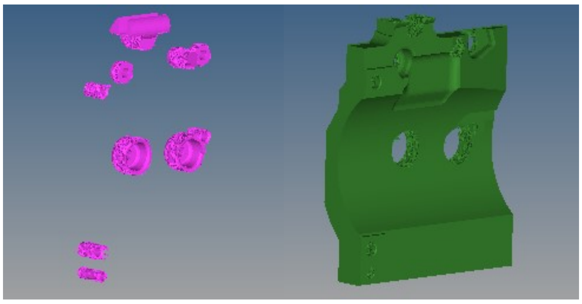


Figure 5.9: Non-design space and desgin space areas are seperated.

5.6 Bracket Topology Optimization Step 5: Property Collections Creation for Each Component

Material properties are defined by ISOTROPIC and card image MAT1. MAT1 defines the material properties for linear, temperature-independent, and isotropic materials. E is for Young`s modulus , RHO is for mass density and NU is for Poisson`s ratio. The most important point while entering material properties is unit system and its interchange. It is frequently a source of error. Bracket material properties are given at table 5.1.

As a property collector 3D, PSOLID are defined and it is linked with material. For non-design pace area same definitons are created under different component name.

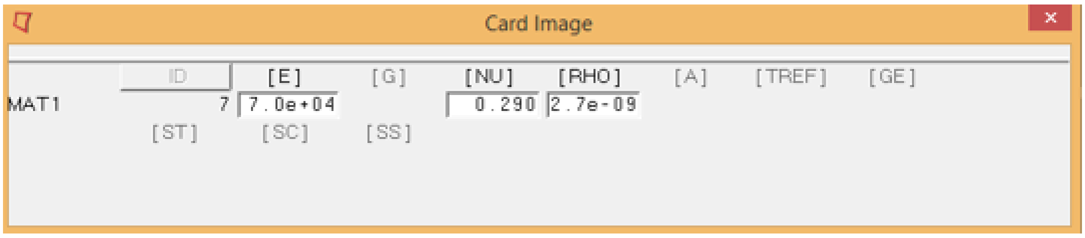


Figure 5.10: HyperMesh screenshot to define material properties.

5.7 Bracket Topology Optimization Step 6: Load Collector Creation

Since the engine itself is largest concentrated mass in this system it is constrainted from cylinder head upper surface from x, y, z translational motions. 3D solid parts do not have rotational degree of freedoms.

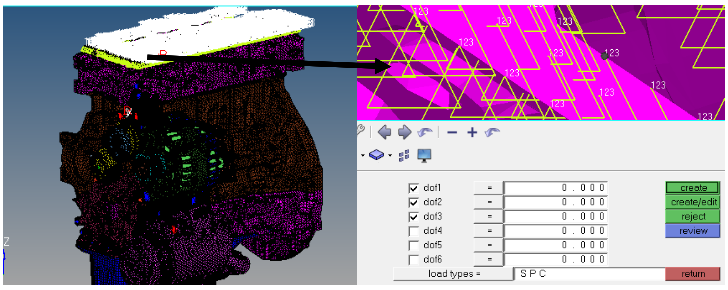


Figure 5.11: Engine is constrained from cylinder upper surface.

5.8 Bracket Topology Optimization Step 7: Optimization Response Function

Two response functions need to be defined, each for objective and contraint functions. Objective is to maximize first natural frequency (as possible much as over the threshold) with constrained volume fraction. Volume fraction is the material fraction of the designable volume. The difference between the mass fraction and the volume fraction is that the mass fraction includes the non-design mass in the fraction calculation but volume fraction is only for the design spce volume.

Objective function is (target) $\omega_{RI} > 182$ Hz and constraint function is volume fraction which aims to maximize frequency by using 50% of design space volume.

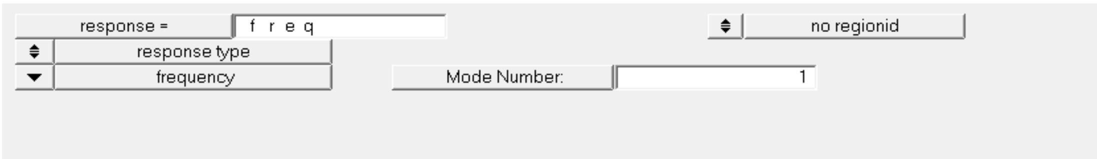


Figure 5.12: HyperMesh screenshot to define response function for frequency.

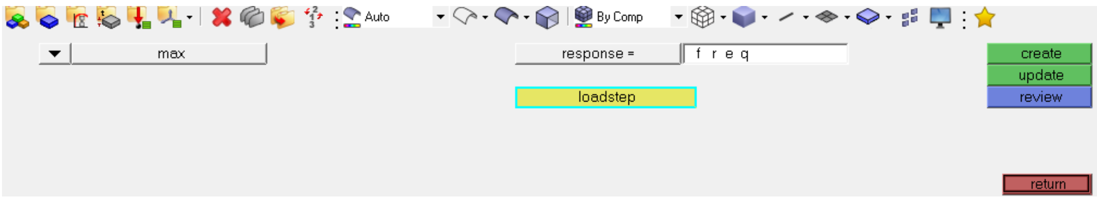


Figure 5.13: HyperMesh screenshot to create load step.

As aim is to maximize first natural frequency as a mode number value 1 is entered. But it does not mean that higher mode values will not be increased. OptiStruct can give results until more than 20 modes. For this study normal mode value will be increased.

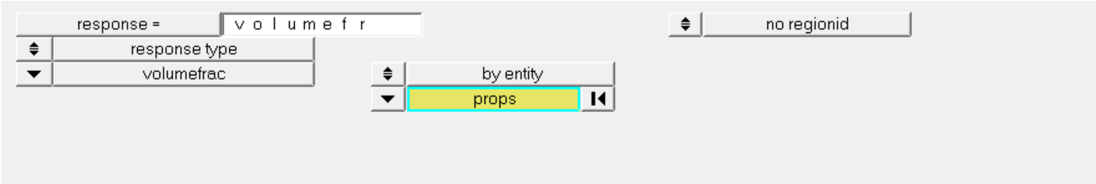


Figure 5.14: HyperMesh screenshot for volume fraction response.

Figure 5.15: HyperMesh screenshot volume fraction response contsraint function.

5.9 Bracket Topology Optimization Step 8: Manufacturing Constraint Definition

There are types of constraints to use in OptiStruct to control manufacturability of the final results such as constraints on minimum and maximum member, symmetry, draw direction. In this study draw direction constraint is considered as it is important for casting parts. (As indicated before bracket manufacturing method is high pressure die cast). The use of die-direction constraints in topology optimization has been implemented in OptiStruct. In the casting process cavities that are not open and lined up with the sliding direction of the die are not feasible. Draw direction constrain the material distribution in a structure so that the die can slide in that direction.

For bracket draw direction as input will ensure easy withdrawal of part from casting and casting from the mould for the proposed design. Single die is being used for this part.

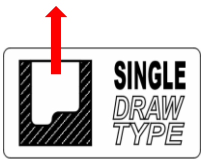


Figure 5.16: Single die draw direction.

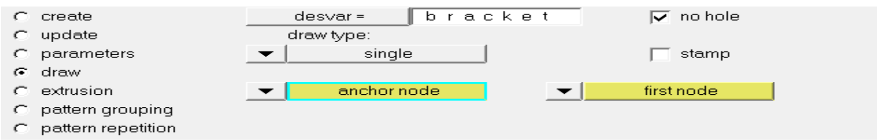


Figure 5.17: HyperMesh screenshot draw type definition.

5.10 Bracket Topology Optimization Step 9: Optimization Results Monitoring

Optimization solution file (*.out) shows response function change for each iteration. Analysis is completed by 8 iterations, all constraints are satisfied, design is feasible and optimization is converged. According to results first natural frequency is increased from 182 Hz to 191 Hz.

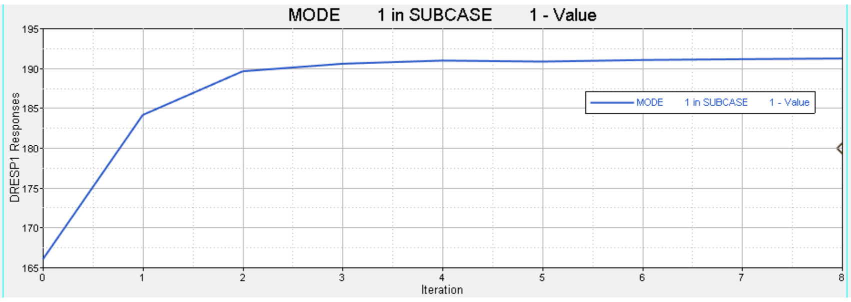


Figure 5.18: Frequency response change for each iteration.

Optimizated design can be monitored by using HyperView. Elements which have element density values as 1 are need to stay at design space. By creating iso-contour graph it is possbile to see design variation according to densities.

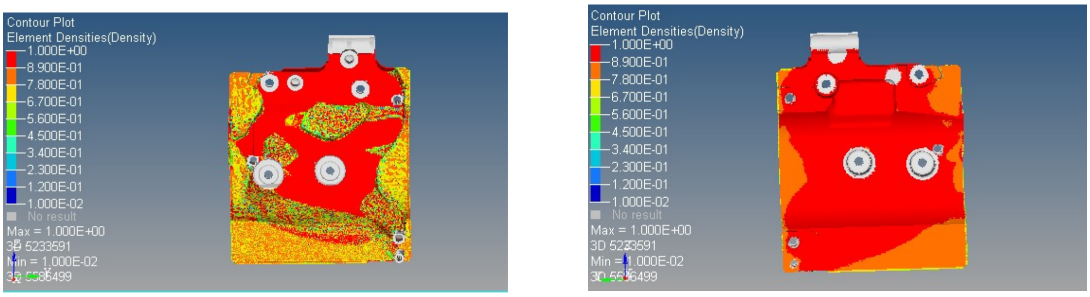


Figure 5.19: HyperView screenshot optimization results of design space.

Red areas shows the elements which has density value 1 that is required for design. If we move the elements which has density value lower than 0.8 part will be get shaped as shown in figure 5.19.

Until now optimization work is completed. Following steps are related with to create manufacturable design and re-analysis the new design if the targets are still being achieved.

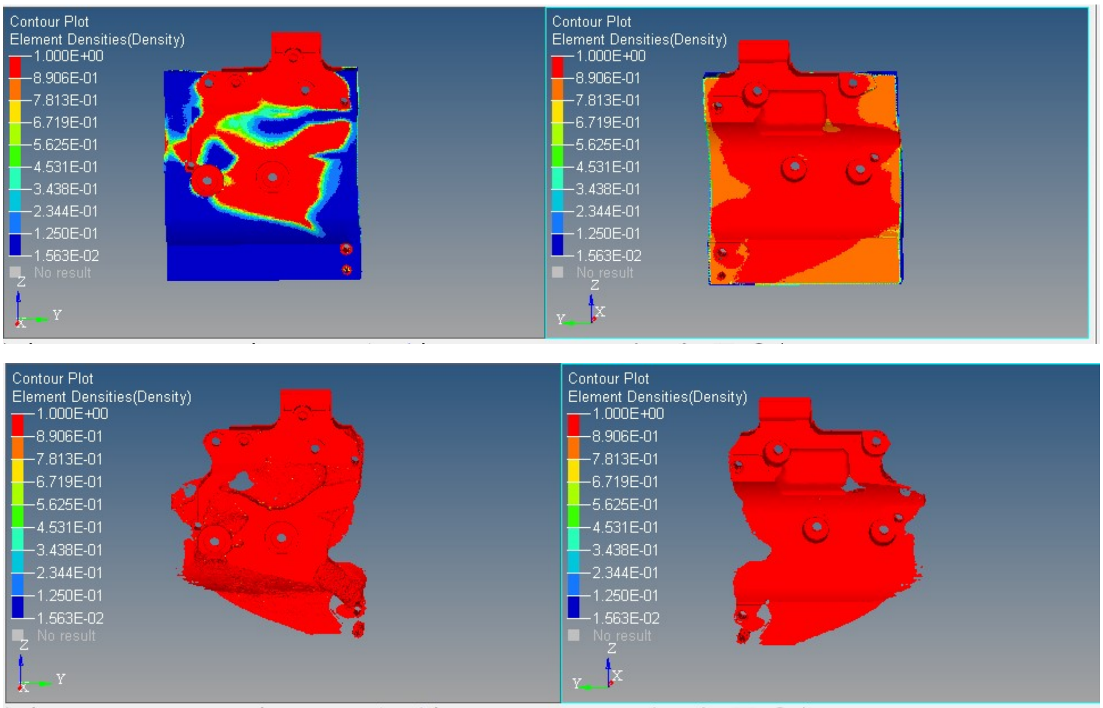


Figure 5.20: HyperView screenshot, below than 0.8 density values are removed.

5.11 Bracket Topology Optimization Step 10: Optimizated Design Reformation

Optimization result need to be reformated based on the HyperView output to get usable part. Sharp edges are rounded and fillets are added by using CATIA. Stiffeners added to material removed places to not to decrease strength of the bracket. To prevent natural frequency decrease due to design modifications minimum correction should be implemented.

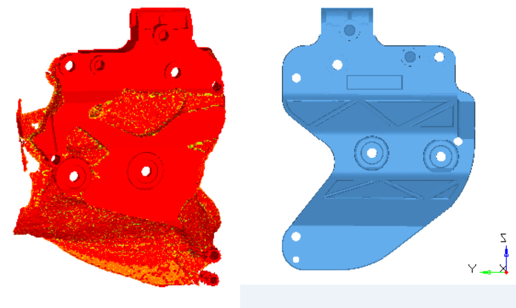


Figure 5.21: Red part is optimization result, blue one is after modification (front side).

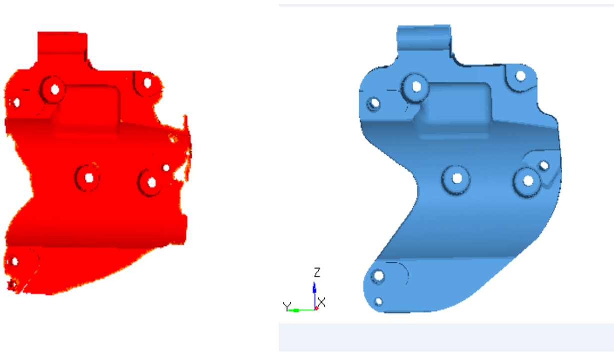


Figure 5.22: Red part is optimization result, blue one is after modification (rear view).

5.12 Bracket Topology Optimization Step 11: Natural Frequency Analysis of Latest Design

Natural frequency target was achieved with optimizated bracket but it is important to analysis redesignated bracket to see if the targets are still being achieved. Finite element model of latest design is created. Material and property allocation is performed same as previous HyperMesh study. EIGRL is choosen as a card image which defines data required to perform real eigenvalue analysis (vibration or buckling) with the Lanczos Method.

Figure 5.23: HyperMesh screenshot, EIGRL card image.

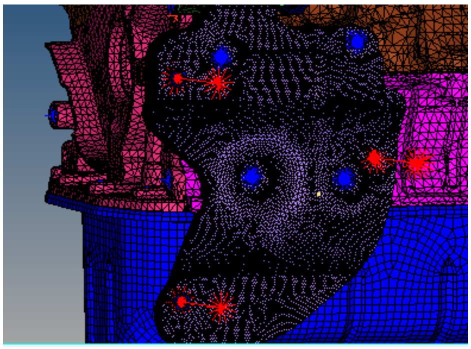


Figure 5.24: Latest design bracket mesh model.

According to analysis results first natural frequency of latest design bracket is 186 Hz which is higher than the target value 182 Hz.

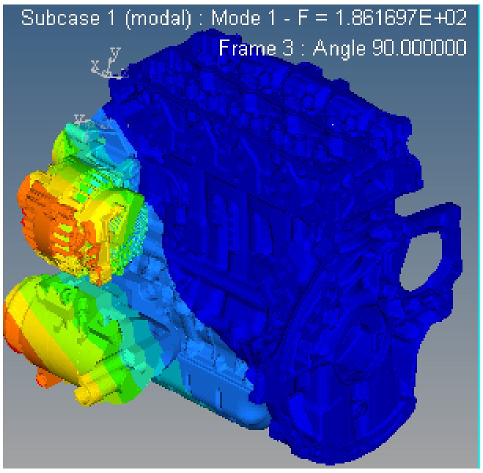


Figure 5.25: Latest design bracket natural frequency analysis result.

5.13 Bracket Topology Optimization Step 12: Stress Analysis of Latest Design

As stated at previous section bracket maximum Von-misses stress value under maximum hub load or thirty times under gravity force (30G) loading must be lower than %60 of the material`s yield value means not more than 78 Mpa.

Peak accessory hub loads are predicted by PowerMark according to engine load versus speed inputs. This calculations are beyond of this thesis scope. Below force inputs are used for stress analysis.

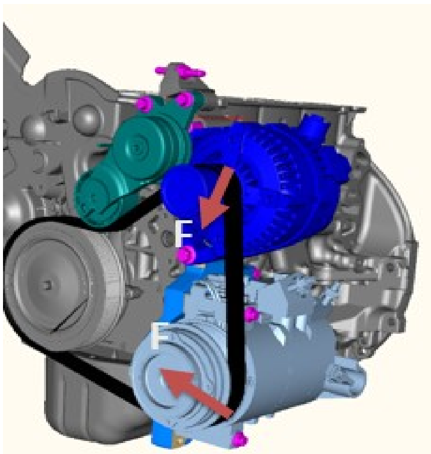


Figure 5.26: Compressor and alternator hub loads.

Table 5.3: Compressor and alternator hub load values.

	F_X	F_Y	F_Z
A/C compressor	1167.5 N	0	1051.2 N
Alternator	54.6 N	0	-1041.6 N

According to stress analysis results at maksimum loading conditions maksimum stress values is 51 Mpa which is less than 78 Mpa.

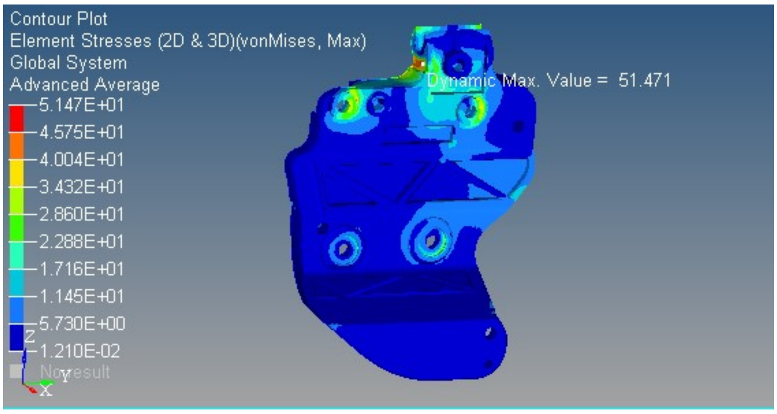


Figure 5.27: Stress analysis results.

Analysis preformed with 30G loading conditions due to the requirement statements. Maximum stress values for 30G X direction is 36 Mpa and fi the g is applied to x, y, z directions at the same time the highest stress value is 53 Mpa which is still lower than target values, 78 Mpa. Detailed screenshots can be seen at appendices section.

Optimization study is completed with 12 steps. First 11 steps are related with to have well design which satisfies optimization objective and constraint functions. Step 12 is to check how safe this desgin from stress point of view. If unreasonable data would be acquired from stress analysis the design would not be successfull. As stress constraint has not been defined while setting up optimization statement stress concentrations were unrecognised. Analysis resulst shows that highest stress concentrations occurs around fixing connections which is reasonable.

In addition assembly check of latest design of bracket is successful.

6. RESULTS AND CONCLUSION

Table 6.1: Comparision of designs.

Design	Base Model	Design Space	Latest Design
First Natural Frequency (Hz)	167	-	186
Weight (gr)	1000,5	1690	972

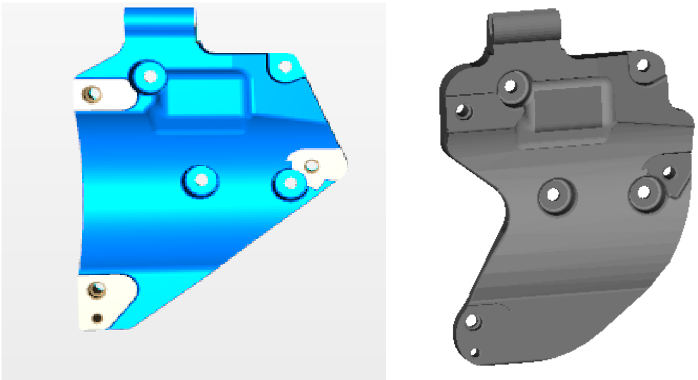


Figure 6.1: Design comparision of base model and latest design bracket front view.

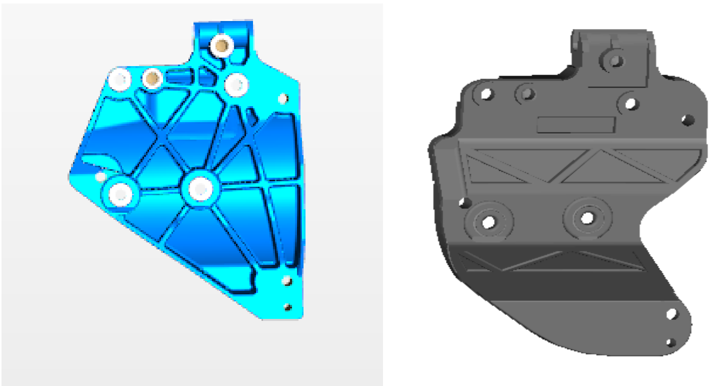


Figure 6.2: Design comparision of base model and latest design bracket rear view.

Topology optimization results that obtained from OptiStruct is not productable design so intensive study is required to reformate the design by using CAD program. As final design is quite different from OptiStruct result analysis should be repeated with last design.

At the beginnig of analysis just bracket model (not engine assembley) is created and modal analysis run but results were not logical (first natural frequency were around 800 Hz).

Then engine assembly is modelled (interacted parts with fead system) and results were as expected (base model frequency 167 Hz) Because of this for optimization tool engine system is used, bracket design space is defined as design space on the program and all other parts from engine defined as non-design space. As a lesson learned if the design that we work is part of a system whole system that is interaction with this part need to be considered to get realistic results.

First natural frequency of bracket can be increased by adding more ribs to current design or create thicker ribs. In this way compliance of design will be decreased hence required modal stiffness value achieved. As one the aims of this study to utilize optimization program and create design from the beginning this way is not suitable. But as a future work bracket natural frequency can be increased by design modifications on current one and compared with the design which created at this study.

REFERENCES

Andresen, B. (2005). *Die Casting Engineering.* New York, NY: Marcel Dekker.

Bendsøe, M. P. (1995). *Optimization of Structural Topology, Shape and Material.* Berlin Heidelberg: Springer-Verlag.

Bendsøe, M. P., & Kikuchi, N. (1988). Generating optimal topologies in structural design using a homogenization method. *Computer Methods in Applied Mechanics and Engineering, 71*, 197-224.

Ehrgott, M. (2005). *Multicriteria Optimization* (2. edition b.). Berlin: Springer Berlin Heidelberg.

Hans, A. (2001). Eschenauer and Niels Olhoff. Topology optimization of continuum structures: A review. *Appl. Mech. Rev., 54*(4), 331-390.

Klarbring, A., & Christensen, P. W. (2009). *An Introduction to Structural Optimization. Solid Mechanics and its Applications.* Berlin: Springer Science + Business Media B.V.

Raffaldini, C. (2011/2012). Structural Optimization in the automotive industry (University of Parma Master Degree Thesis in Mechanical Engineering)

Rao., S. S. (1996). *Engineering Optimization — Theory and Practice* (3. edition b.). John Wiley & Sons.

Rozvany, G. I. (2000). Aims, scope, methods, history and unified terminology of computeraided topology optimization in structural mechanics. *Struct. Multidisc. Optim., 21*, 90– 108.

Sreedhar, B. and Sasidhar, N. (2012). Optimization of Mounting Bracket (Hyundai Motor India , SAE Paper)

Vasudeva, R. and Soundararajan (2012). Optimization of Front Axle Mounts Vibration Characteristics for Driveline Integration with Chassis. (General Motors Technical Center India , SAE Paper)

Inc, A. E. (2013,1,1). *Optistruct*. date retrieved 01.02.2015 retrieved from <http://www.altairhyperworks.com/HWTemp3Product.aspx?product_id=19& item_name=Capabilities >

Learneasy.(2015,11) Learneasy. date retrieved 04.03.2015 retrieved from <http://www.learneasy.info/MDME/MEMmods/MEM30007A/processing/ pocessing.html>

APPENDICES

APPENDIX A: FE Model Figures

APPENDIX B: Optimization Iterations *.log file

APPENDIX A

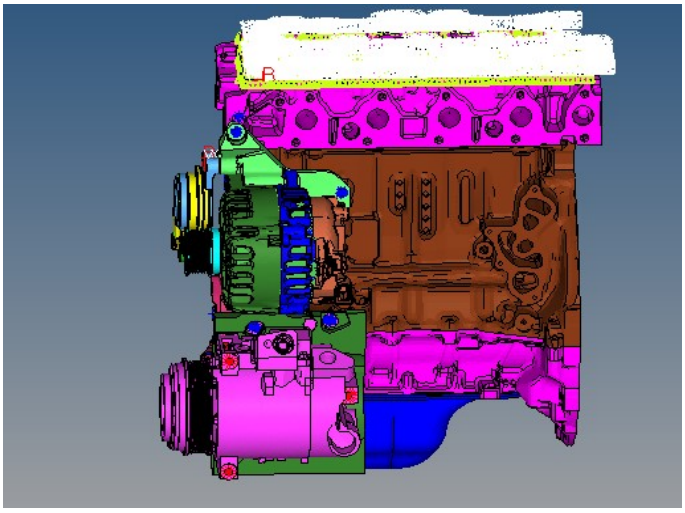


Figure A.1: Engine is fixed from gasket which is shown in white.

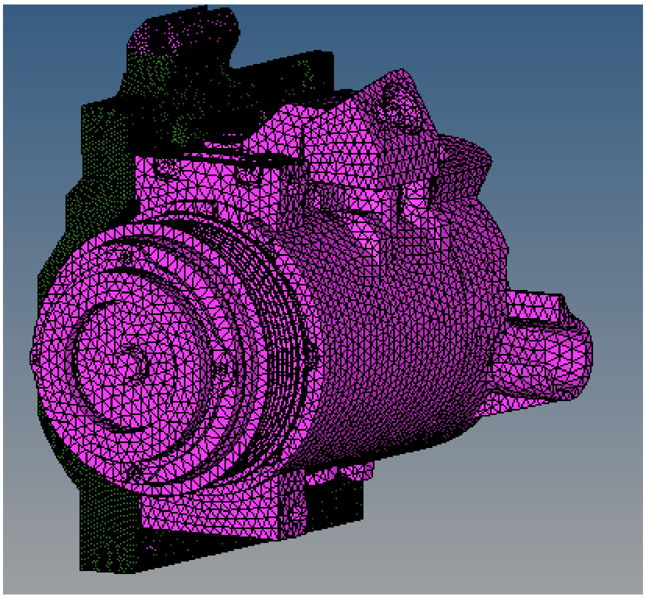


Figure A.2: Bracket design space and compressor assembly mesh model.

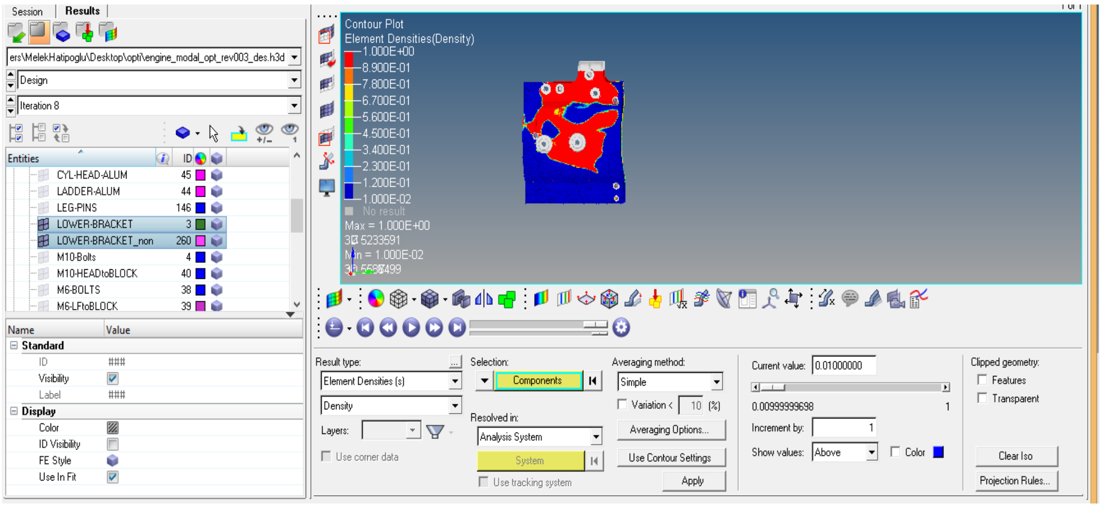


Figure A.3: Optimized bracket view when density values setted as 0.01

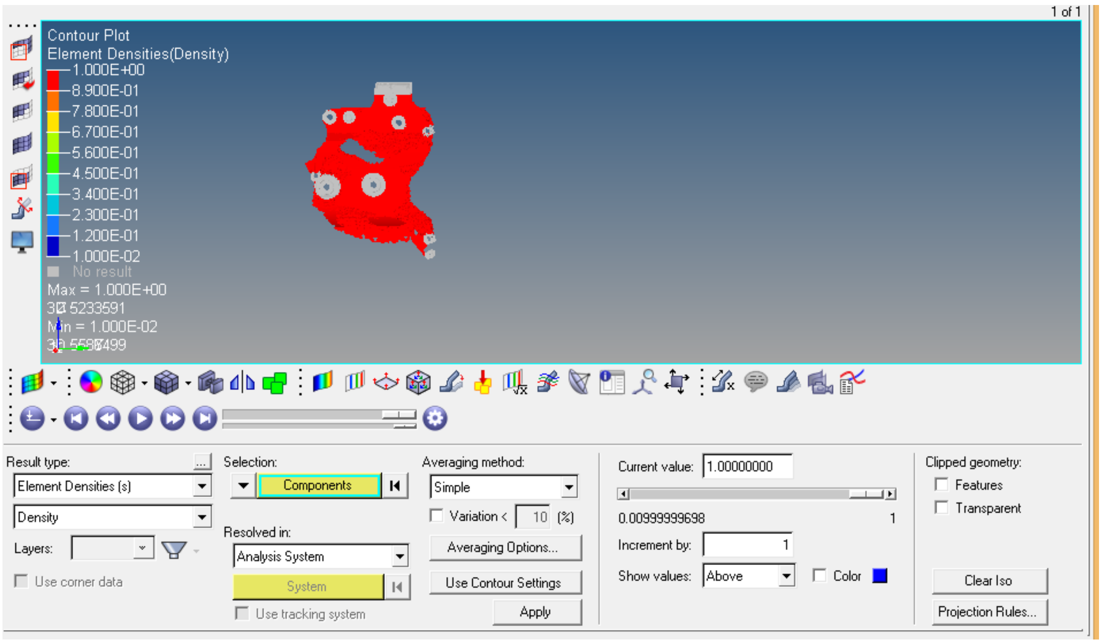


Figure A.4: Optimized bracket view when density values setted as 1. Densities between 0 and 1 are removed from design space.

APPENDIX B

Optimization result file. Frequency change can be seen for each iteration.

OPTIMIZATION RESULT FILE (*out file)

```
OPTIMIZATION PROBLEM PARAMETERS :
--------------------------------

 Objective Function  : Maximize Frequency

 ------------------
 Response Summary :
 ------------------

 Number of volume fraction responses   :       1
 Number of frequency responses         :       1

  --------------------------
  Frequency Subcase Summary :
  --------------------------

  ----------   --------  --------
  Subcase ID    SPC ID    MODE #
  ----------   --------  --------
       1          2        4
  ----------   --------  --------

  --------------------------
  Design Parameters Summary :
  --------------------------

  Total # of topology design elements:    676109
  Total # of topology cards :1

  Total Volume of Design Material :   6.8860E+05
  Total Mass of Design Material :   1.8592E-03
  Volume of Non-Design Material :   1.4845E+07
  Mass of Non-Design Material :   5.0320E-02

  Solid Design Elements  :  PSOLID
  ---------------------    --------
                   900134


  --------------------------------
  Optimization Parameters Summary :
  --------------------------------

  Initial Material Fraction [0,1] :   0.5000
```

```
  Minimum Element Volume Fraction :   0.0100
  Discreteness Parameter          :   1.0000

  Topology Optimization Method    :   Density Method
  Maximum Number of Iterations    : 80

  Mode Scaling Parameter          :   1.0000E+00
  Convergence Tolerance           :   5.0000E-03
  Step Size (Topology)            :   0.5000

  Checkerboard Control            :   On (1 - Global Averaging)

  Run Type                        :   Topology Optimization

 ------------------------------
 Topology Optimization Summary :
 ------------------------------

 -------- ----------- ----------- -----------
 DTPL ID    Minimum     Maximum     Minimum
          Member Size Member Size  Gap Size
 -------- ----------- ----------- -----------
        1  0.2985E+01
 -------- ----------- ----------- -----------

 -------- ------- ------- ----------- ----------- ----------- -----------
 DTPL ID  Pattern Pattern  Draw Dir.   Stamping     No Hole    Extrusion
           Repet.  Group. Constraints Constraints Constraints Constraints
 -------- ------- ------- ----------- ----------- ----------- -----------
        1   NONE    NONE     SINGLE       NONE        YES        NONE
 -------- ------- ------- ----------- ----------- ----------- -----------

  Restart from previous solution  :   No
  Scratch file directory          : ./
                    Free space: 440138 MB
  Number of CPU processors        :        16


 MEMORY ESTIMATION INFORMATION :
 ------------------------------

 Solver Type is:  Sparse-Matrix Solver
                  Direct Method
                  Block Lanczos Eigenvalue Extraction

 Current Memory (RAM)                                    :   67459 MB
 Estimated Minimum Memory (RAM) for Out of Core Solution :   15871 MB
 Recommended Memory (RAM) for Out of Core Solution       :   17818 MB
 Recommended Memory (RAM) for In-Core Solution           :   67459 MB
```

DISK SPACE ESTIMATION INFORMATION :

Estimated Disk Space for Output Data Files : 5459 MB
Estimated Scratch Disk Space for In-Core Solution : 74784 MB
Estimated Scratch Disk Space for Out of Core Solution : 130149 MB

BEGINNING OPTIMIZATION SOLUTION ....

OPTIMIZATION HISTORY INFORMATION :

ITERATION 0

(Scratch disk space usage for starting iteration = 7717 MB)
(Running in-core solution)

Objective Function (Maximize FREQ) = 1.66037E+02
Maximum Constraint Violation % = 0.00000E+00
Design Volume Fraction = 5.00000E-01 Mass = 5.12498E-02

Subcase	Mode	Frequency	Eigenvalue	Generalized Stiffness	Generalized Mass
1	1	1.660372E+02	1.088355E+06	1.088355E+06	1.000000E+00
1	2	1.909813E+02	1.439930E+06	1.439930E+06	1.000000E+00
1	3	2.104793E+02	1.748954E+06	1.748954E+06	1.000000E+00
1	4	2.893978E+02	3.306361E+06	3.306361E+06	1.000000E+00

ITERATION 1

Objective Function (Maximize FREQ) = 1.84130E+02 % change = 10.90
Maximum Constraint Violation % = 0.62845E+01
Design Volume Fraction = 5.31423E-01 Mass = 5.13082E-02

Subcase	Mode	Frequency	Eigenvalue	Generalized Stiffness	Generalized Mass
1	1	1.841302E+02	1.338473E+06	1.338473E+06	1.000000E+00
1	2	2.095330E+02	1.733264E+06	1.733264E+06	1.000000E+00
1	3	2.171671E+02	1.861864E+06	1.861864E+06	1.000000E+00
1	4	3.093571E+02	3.778156E+06	3.778156E+06	1.000000E+00

RETAINED RESPONSES TABLE

```
--------------------------------------------------------------------------------
Response Type  Response    Subcase Grid/    DOF/   Response    Objective   Viol.
User-ID        Label     /RANDPS Element/  Comp    Value     Reference/    %
                  +Frqncy MID/PID/  /Reg            Constraint
                       Mode No.                   Bound
--------------------------------------------------------------------------------
     1 FREQ  freq            1       1    --   1.841E+02 MAX
     2 VOLFR volumefr        --   900134  SOLI   5.314E-01  <  5.000E-01    6.3 V
--------------------------------------------------------------------------------
```

MOST VIOLATED CONSTRAINTS TABLE

```
--------------------------------------------------------------------------------
Response Type  Response    Subcase Grid/    DOF/   Response    Constraint  Viol.
User-ID        Label     /RANDPS Element/  Comp    Value      Bound        %
                  +Frqncy MID/PID/  /Reg
                       Mode No.
--------------------------------------------------------------------------------
     2 VOLFR volumefr        --   900134  SOLI   5.314E-01  <  5.000E-01    6.3 V
--------------------------------------------------------------------------------
```

ITERATION 2

Objective Function (Maximize FREQ) = 1.89624E+02 % change = 2.98
Maximum Constraint Violation % = 0.00000E+00
Design Volume Fraction = 4.99991E-01 Mass = 5.12498E-02

Subcase	Mode	Frequency	Generalized Eigenvalue	Generalized Stiffness	Mass
1	1	1.896241E+02	1.419537E+06	1.419537E+06	1.000000E+00
1	2	2.098955E+02	1.739266E+06	1.739266E+06	1.000000E+00
1	3	2.245573E+02	1.990738E+06	1.990738E+06	1.000000E+00
1	4	3.122669E+02	3.849565E+06	3.849565E+06	1.000000E+00

RETAINED RESPONSES TABLE

```
--------------------------------------------------------------------------------
Response Type  Response    Subcase Grid/    DOF/   Response    Objective   Viol.
User-ID        Label     /RANDPS Element/  Comp    Value     Reference/    %
                  +Frqncy MID/PID/  /Reg            Constraint
                       Mode No.                   Bound
--------------------------------------------------------------------------------
     1 FREQ  freq            1       1    --   1.896E+02 MAX
     2 VOLFR volumefr        --   900134  SOLI   5.000E-01  <  5.000E-01    0.0 A
--------------------------------------------------------------------------------
```

ITERATION 3

The 1st satisfied convergence ratio = 4.8919E-03

Objective Function (Maximize FREQ) = 1.90556E+02 % change = 0.49
Maximum Constraint Violation % = 0.00000E+00
Design Volume Fraction = 4.99991E-01 Mass = 5.12498E-02

Subcase	Mode	Frequency	Generalized Eigenvalue	Generalized Stiffness	Mass
1	1	1.905563E+02	1.433528E+06	1.433528E+06	1.000000E+00
1	2	2.099259E+02	1.739769E+06	1.739769E+06	1.000000E+00
1	3	2.257870E+02	2.012601E+06	2.012601E+06	1.000000E+00
1	4	3.153130E+02	3.925034E+06	3.925034E+06	1.000000E+00

RETAINED RESPONSES TABLE

Response User-ID	Type	Response Label	Subcase /RANDPS +Frqncy Mode No.	Grid/ Element/ MID/PID/	DOF/ Comp /Reg	Response Value		Objective Reference/ Constraint Bound	Viol. %
1	FREQ	freq	1	1	--	1.906E+02		MAX	
2	VOLFR	volumefr	--	900134	SOLI	5.000E-01	<	5.000E-01	0.0 A

ITERATION 4

The 2nd satisfied convergence ratio = 2.1280E-03

Objective Function (Maximize FREQ) = 1.90963E+02 % change = 0.21
Maximum Constraint Violation % = 0.00000E+00
Design Volume Fraction = 4.99994E-01 Mass = 5.12498E-02

Subcase	Mode	Frequency	Generalized Eigenvalue	Generalized Stiffness	Mass
1	1	1.909626E+02	1.439648E+06	1.439648E+06	1.000000E+00
1	2	2.099358E+02	1.739934E+06	1.739934E+06	1.000000E+00
1	3	2.262035E+02	2.020033E+06	2.020033E+06	1.000000E+00
1	4	3.166676E+02	3.958832E+06	3.958832E+06	1.000000E+00

RETAINED RESPONSES TABLE

Response Type Response Subcase Grid/ DOF/ Response Objective Viol.
User-ID Label /RANDPS Element/ Comp Value Reference/ %

ITERATION 5

The 2nd satisfied convergence ratio = 1.4340E-03

Objective Function (Maximize FREQ) = 1.90830E+02 % change = -0.07

Maximum Constraint Violation % = 0.86079E+00
Design Volume Fraction = 5.04304E-01 Mass = 5.12578E-02

Subcase	Mode	Frequency	Eigenvalue	Generalized Stiffness	Generalized Mass
1	1	1.908299E+02	1.437648E+06	1.437648E+06	1.000000E+00
1	2	2.099161E+02	1.739608E+06	1.739608E+06	1.000000E+00
1	3	2.252570E+02	2.003163E+06	2.003163E+06	1.000000E+00
1	4	3.155168E+02	3.930111E+06	3.930111E+06	1.000000E+00

RETAINED RESPONSES TABLE

Response User-ID	Type	Response Label	Subcase /RANDPS +Frqncy Mode No.	Grid/ Element/ MID/PID/	DOF/ Comp /Reg	Response Value	Objective Reference/ Constraint Bound	Viol. %
1	FREQ	freq	1	1	--	1.908E+02	MAX	
2	VOLFR	volumefr	--	900134	SOLI	5.043E-01	< 5.000E-01	0.0 A

ITERATION 6

Objective Function (Maximize FREQ) = 1.91029E+02 % change = 0.10
Maximum Constraint Violation % = 0.13657E+01
Design Volume Fraction = 5.06829E-01 Mass = 5.12625E-02

Subcase	Mode	Frequency	Eigenvalue	Generalized Stiffness	Generalized Mass
1	1	1.910293E+02	1.440654E+06	1.440654E+06	1.000000E+00
1	2	2.099212E+02	1.739692E+06	1.739692E+06	1.000000E+00
1	3	2.254520E+02	2.006633E+06	2.006633E+06	1.000000E+00
1	4	3.164066E+02	3.952309E+06	3.952309E+06	1.000000E+00

RETAINED RESPONSES TABLE

Response User-ID	Type	Response Label	Subcase /RANDPS +Frqncy Mode No.	Grid/ Element/ MID/PID/	DOF/ Comp /Reg	Response Value	Objective Reference/ Constraint Bound	Viol. %
1	FREQ	freq	1	1	--	1.910E+02	MAX	
2	VOLFR	volumefr	--	900134	SOLI	5.068E-01	< 5.000E-01	

```
--------------------------------------------------------------------------------

                MOST VIOLATED CONSTRAINTS TABLE
--------------------------------------------------------------------------------
Response Type  Response     Subcase Grid/     DOF/   Response     Constraint   Viol.
User-ID          Label      /RANDPS Element/  Comp    Value       Bound        %
                       +Frqncy MID/PID/  /Reg
                             Mode No.
--------------------------------------------------------------------------------
     2 VOLFR volumefr        --   900134  SOLI   5.068E-01  <  5.000E-01    1.4 V
--------------------------------------------------------------------------------
```

ITERATION 7

The 1st satisfied convergence ratio = 5.4067E-04

```
Objective Function (Maximize FREQ ) =  1.91133E+02   % change =        0.05
Maximum Constraint Violation %      =  0.43396E+00
Design Volume Fraction              =  5.02170E-01   Mass    =  5.12538E-02
```

Subcase	Mode	Frequency	Generalized Eigenvalue	Generalized Stiffness	Mass
1	1	1.911326E+02	1.442213E+06	1.442213E+06	1.000000E+00
1	2	2.099244E+02	1.739745E+06	1.739745E+06	1.000000E+00
1	3	2.255870E+02	2.009036E+06	2.009036E+06	1.000000E+00
1	4	3.167952E+02	3.962022E+06	3.962022E+06	1.000000E+00

```
                   RETAINED RESPONSES TABLE
--------------------------------------------------------------------------------
Response Type  Response     Subcase Grid/     DOF/   Response     Objective    Viol.
User-ID          Label      /RANDPS Element/  Comp    Value       Reference/    %
                       +Frqncy MID/PID/  /Reg                 Constraint
                             Mode No.                    Bound
--------------------------------------------------------------------------------
     1 FREQ  freq            1       1    --   1.911E+02 MAX
     2 VOLFR volumefr        --   900134  SOLI   5.022E-01  <  5.000E-01    0.0 A
--------------------------------------------------------------------------------
```

ITERATION 8

The 2nd satisfied convergence ratio = 4.8800E-04

```
Objective Function (Maximize FREQ ) =  1.91226E+02   % change =        0.05
Maximum Constraint Violation %      =  0.39066E+00
Design Volume Fraction              =  5.01953E-01   Mass    =  5.12534E-02
```

Generalized Generalized

Subcase	Mode	Frequency	Eigenvalue	Stiffness	Mass
1	1	1.912260E+02	1.443622E+06	1.443622E+06	1.000000E+00
1	2	2.099284E+02	1.739811E+06	1.739811E+06	1.000000E+00
1	3	2.257655E+02	2.012217E+06	2.012217E+06	1.000000E+00
1	4	3.172224E+02	3.972715E+06	3.972715E+06	1.000000E+00

RETAINED RESPONSES TABLE

Response User-ID	Type	Response Label	Subcase /RANDPS +Frqncy Mode No.	Grid/ Element/ MID/PID/	DOF/ Comp /Reg	Response Value		Objective Reference/ Constraint Bound	Viol. %
1	FREQ	freq	1	1	--	1.912E+02		MAX	
2	VOLFR	volumefr	--	900134	SOLI	5.020E-01	<	5.000E-01	0.0 A

****

OPTIMIZATION HAS CONVERGED.

FEASIBLE DESIGN (ALL CONSTRAINTS SATISFIED).

****

RESOURCE USAGE INFORMATION

MAXIMUM MEMORY USED 67459 MB
MAXIMUM DISK SPACE USED 9954 MB

****

****

COMPUTE TIME INFORMATION

EXECUTION STARTED Tue Apr 07 20:25:34 2015
EXECUTION COMPLETED Wed Apr 08 01:52:45 2015
ELAPSED TIME 05:27:11
CPU TIME 2+21:48:45

CURRICULUM VITAE

Name Surname : Melek HATIPOLU

Place and Date of Birth : Kircaali, 04.03.1989

E-Mail : melek_hatipoglu@yahoo.com

EDUCATION

B.Sc. : Yildiz Technical University /Mechanical Engineering (2007-2011)

M.Sc. : Istanbul Technical University/ Mechanical Design Engineering (2012-2015)

PROFESSIONAL EXPERIENCE AND REWARDS:

01.2015 - current RLE International Group / Project Engineer / Cologne, Germany