Available online at www.sciencedirect.com

ScienceDirect

Procedia Structural Integrity 26 (2020) 211–218

Structural Integrity Procedia

www.elsevier.com/locate/procedia

The 1st Mediterranean Conference on Fracture and Structural Integrity, MedFract1

Design for NVH: topology optimization of an engine bracket support

Enrico Armentani[a]*, Venanzio Giannella[b], Antonio Parente[c], Mauro Pirelli[c]

[a]Department of Chemical, Materials and Production Engineering, University of Naples Federico II, P.le V. Tecchio, 80, 80125 Napoli, Italy
[b]Deptartment of Industrial Engineering, University of Salerno, via Giovanni Paolo II, 132, Fisciano (SA), Italy
[a]Fiat Chrysler Automobiles (FCA) Powertrain S.p.A., via ex Aeroporto, 80038 Pomigliano D'Arco, Italy

Abstract

Noise Vibration and Harshness (NVH) issues are proven to be the main drivers for customer dissatisfaction in the latest years. This work relies on the framework of Design For X (DFX), specifically, Design for NVH. Main goal of this work was to perform a Topology Optimization (TO) of an engine bracket based on its vibrational behavior, in order to reduce the vibrations transmitted from the engine to the chassis and, consequently, improving the comfort for passengers. In particular, the target function was defined with the aim of increasing the first natural frequency of the bracket, whereas the bracket mass reduction was considered as a constraint function for the TO process. The vibrational characterization of the bracket was based on Frequency Response Function (FRF) analyses which, conducted via FEM (Finite Element Method), allowed to identify the resonant frequencies of the different bracket configurations built up during the TO. The FEM models included the cylinder head, with the related engine bracket support under optimization; the latter is connected to the bracket on which the simulation load was applied. The TO turned out to be effective in lowering the mass of engine bracket support of nearly 20% and, at the same time, increasing the first natural frequency of nearly 10%, this latter result was sufficient to guarantee an improvement of the comfort for passengers.

Keywords: Design For X; NVH; topology optmization; engine; FRF

* Corresponding author. Tel.: +39-081-768-2450; fax: +39-081-768-2450.
E-mail address: enrico.armentani@unina.it

10.1016/j.prostr.2020.06.024

1. Introduction

The reduction of acoustic emissions and the improvement of car cabin interior comfort (Citarella et al. 2018) are on the path of all major industries of the transport system, having a direct impact on customer satisfaction and, consequently, on the commercial success of new car models. Topics to be tackled deal with computational, instrumentation and data analysis of noise and vibration of engine components, allowing for structural and airborne noise, sound absorption, cabin acoustic treatments, duct acoustics (Citarella et al. 2011), active noise control and vibroacoustic properties of materials.

In the last decades Topology Optimization (TO) has established itself as an important tool for optimizing engineering components, especially when linear analyses are sufficient to characterize component performances like strength, mass, natural frequencies and/or stiffness.

Most of the parts in the chassis area, such as the engine brackets, are generally cast parts. Especially for these components, TO helps in finding the optimal features of components, such as the optimal cross-section or, for that concern ribs, their optimal number and arrangement. It has been found that the determination of these features gives the largest amount of the optimization potential, whereas the fine-tuning using sizing or shape optimization typically gives only a minor contribution (Harzheim et al., 1995, 1996, 1999). Usually, strength requirements are the major issues in practice, but there are indications that the targets of strength or maximum stiffness are the same or at least very close together (Pederson, 1998, 2000).

TO has been applied successfully to cast parts for several years (Harzheim et al. 1996, 1999), by leveraging on codes based on empirical methods, e.g. see the code Soft Kill Option (SKO) (Harzheim et al., 1995). Such codes were based on the simulation of the adaptive biological growth rule of biological growth carriers like trees and bones. Experience has shown that the growth rule leads to designs with uniform surface stress, consequently, such codes were appropriate tools to deal with strength problems, common for cast parts. In contrast, the approach of OptiStruct (Bendsoe, 1995; Ma et al., 1992; Altair Engineering Inc., 2002) is more flexible because nearly every response, such as compliance, natural frequencies or displacements, can be used as objective functions and/or constraints. Consequently, OptiStruct seems to be a powerful code that can be used for a wide variety of problems. More generally speaking, specialized optimization codes, although equipped with fewer analysis capabilities than general Finite Element Method (FEM) based codes, offer more features and higher efficiency for optimization. The reasons for this are two-fold: (1) highly specialized codes are typically smaller and therefore more flexible for incorporating the latest developments than general codes; (2) for specialized codes, highest priority is devoted to its core technology of optimization.

Finite Element Method (FEM) based topology, sizing and shape optimization tools are typically used as part of a two-phase design process. Firstly, TO is performed to obtain a first view on an optimal configuration for the structure, namely an initially optimized design with optimal load paths. Next, the suggested configuration is interpreted by user to form a feasible engineering design and this design is then optimized using detailed sizing and shape optimization methods with real design requirements.

This paper studies the use of Altair's FEM based topology, sizing and shape optimisation tool Optistruct to design a cast part of a vehicle. In particular, a bracket support of a 4-cylinder, 4-stroke, petrol engine was considered as the component undergoing the optimization process. The main goal was to perform a TO of the bracket based on its vibrational behavior, in such a way to limit the vibrations coming from the engine with consequent improvement of the passengers' comfort. In particular, the target was focused on the increase of the first natural frequency of the bracket support whereas its mass reduction was considered as a constraint function for the optimization process. The vibrational characterization of the bracket was based on FRF analyses conducted via FEM, as widely used for such kind of problems (Siano et al. 2014, 2018; Armentani et al. 2013, 2016, 2017, 2018). These analyses allowed to identify the resonant frequencies of the different FEM models built up during the optimization process. Such FEM models comprised the cylinder head, useful to properly constrain the bracket, the bracket support under optimization and a second bracket on which the load was applied. In this work, Optistruct was used as optimisation tool whereas Altair Hypermesh was used as pre- and post-processor tool.

Nomenclature

E	Young's modulus
υ	Poisson' ratio
ρ	Mass density

2. FEM modelling

A FEM model was built up (Fig. 1) in order to perform the TO process of the bracket support. The FEM model comprised three components, namely the support bracket under analysis, the bracket on which the load was applied, and the cylinder head in order to properly constrain the previous components. It is worth noting that the analyses were conducted on just three components in order to reduce the computational burden, whereas the dynamic contributions from alternator, exhaust system, etc., were neglected.

The CAD model of the bracket support under analysis is shown in Fig. 2. All the considered parts were made in aluminum whereas the bolts to interconnect them were made of steel. Main material data are listed in Tab. 1.

Boundary conditions considered in the FEM analyses are shown in Fig. 1; clamped boundary conditions were applied to the bolt holes and in particular at the connections between the bracket under analysis and the engine head, whereas a unitary force was applied to the bracket that connects the engine to the chassis.

Table 1. Main mechanical properties for the materials considered numerically.

Parameter	Aluminum	Steel
Young's modulus E	72.6 GPa	210 GPa
Poisson' ratio υ	0.27	0.3
Mass density ρ	2740 kg/m^3	7850 kg/m^3

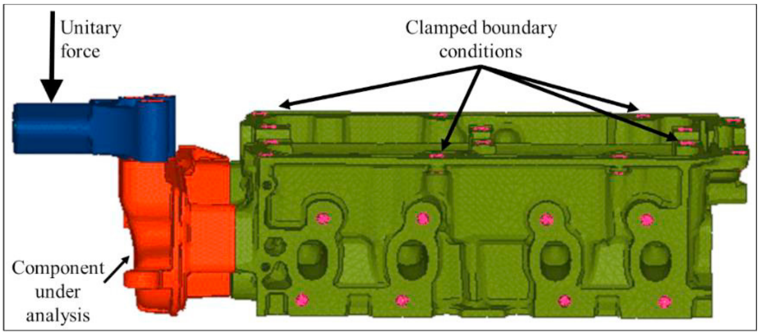


Fig. 1. FEM model with related boundary conditions considered in the analyses.

3. Topology Optimization process

To formulate an optimization problem, an objective function, together with constraint functions and design variables, need to be introduced:

- The objective function represents the characteristics that the user wants to minimize/maximize;
- The constraint functions represent the constraints that have to be taken into account to steer the optimization to a sought solution;
- The design variables are the parameters (e.g. geometric, materials related…) to be fine-tuned in a given range in order to configure an "optimal" structure.

From the mathematical standpoint, the problem can be described by a set of design variables $p_j = (p_1, p_2, \dots, p_n)$ that varies in a given range $p^l \leq p_j \leq p^u$, and by a function $\psi_i(p)$ representing the target value of the response function of the system $\psi_0(p)$. The aim of the optimization process is then to minimize/maximize the response of the system $\psi_0(p)$ considering the constraint functions to restraint the optimization solution.

More specifically, TO is a mathematical method that optimizes material layout within a given design space, for a given set of loads, boundary conditions and constraints with the goal of improving the performance of the system. The design variables are represented by the normalized density of each finite element in the mesh which, varying between 0 and 1, allows to define, in a given design space, which elements are required and which one can be removed.

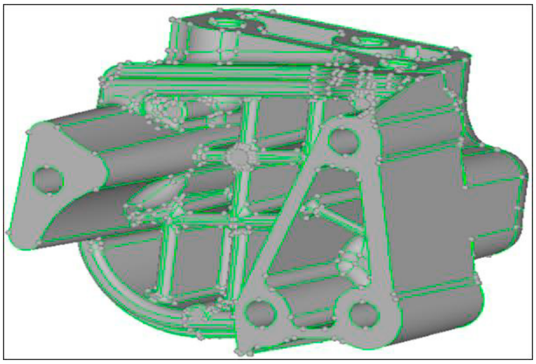


Fig. 2. CAD model of the bracket support under analysis.

The TO process is a step-by-step process that starts with an initial mesh of the structure, performs the FEM analysis, varies the elements' density based on the defined optimization algorithm, and re-iterates the process with a new mesh.

The bracket considered in the TO process was divided in two main volumes. A first volume represented the "design space", namely the volume that comprised the elements that the TO process was allowed to modify, whereas the second volume represented the "non-design space", namely the volume comprising the unmodifiable elements. Such volumes are shown in Fig. 3 for two different shapes of the design space that were envisaged for such TO process, i.e.,

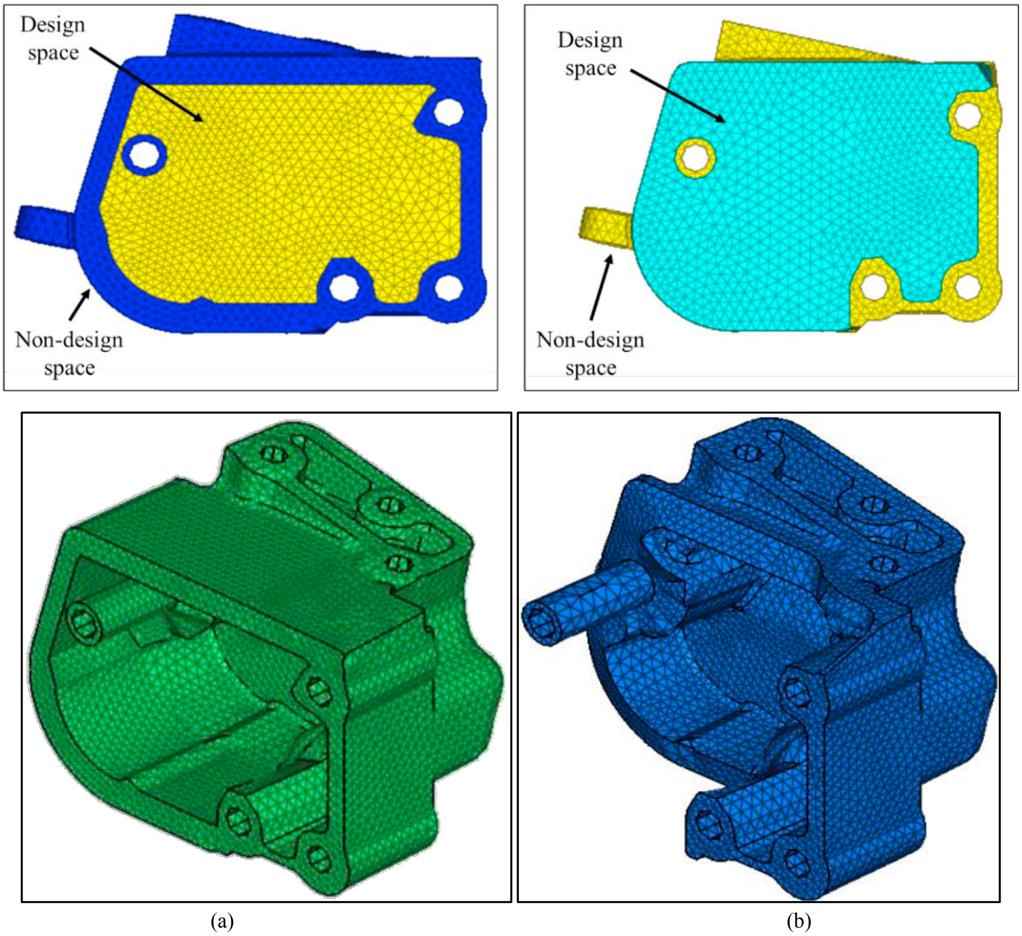


Fig. 3. Different volume subdivisions of the bracket: (a) small design space; (b) large design space.

a small and a large one. Two preliminary FRF analyses were conducted with the models shown in Fig. 3 in order to understand whether the small or the large design space was preferable for the current analyses.

From these preliminary calculations, the model with the larger design space (Fig. 3b) was selected since its mass was nearly 30% lower than the model with the smaller design space (822 g against 1150 g), whereas the first natural frequency was only 6% lower (915 Hz against 971 Hz).

Subsequently, the objective function was defined by maximization of the increment of the first natural frequency of the bracket support, whereas the constraint function was defined imposing that the TO process was allowed to remove the 70 % from the initial total design space volume (for such a purpose, a volume fraction of 30% was set up in the code Altair Engineering Inc., 2002). In final, the density of elements was selected as design variables on which the code was allowed to work during the iterations.

The TO procedure leveraged on a gradient-based method to evaluate the impact of the elements density variation throughout the process. Moreover, a power law representation of the elements' stiffness was selected to evaluate the elements' stiffness variability with respect to the elements' normalized density. The iterative process stops when the code adds 30% of design space volume to the initial non-design space, thus reaching the requested optimized solution

4. Topology Optimization results

The resulting design shape of the bracket is shown in Fig. 4: normalized density of the elements, ranging from 0 to 1, is reported in the contour plot (elements with unmodified density in red, parts with lowered density in blue). As a first outcome, it was evident that a high removal of material was obtained from the TO, especially in the parts that were not contributing significantly to the stiffness of the component. Moreover, it could be observed a relevant densification of material in the connections between holes and bracket, mainly due to the stress concentration in those locations. Such resulting design shape turned out to have a first natural frequency of nearly 1142 Hz.

Subsequently, a post-optimization design of the bracket was developed based on such resulting shape (Fig. 4). This was required in order to achieve a design shape simple to manufacture. The so obtained post-optimization design shape is shown in Fig. 5. A further FRF analysis was conducted by considering this feasible design and the first natural frequency resulted equal to 1122 Hz whereas its mass was equal to 989 g.

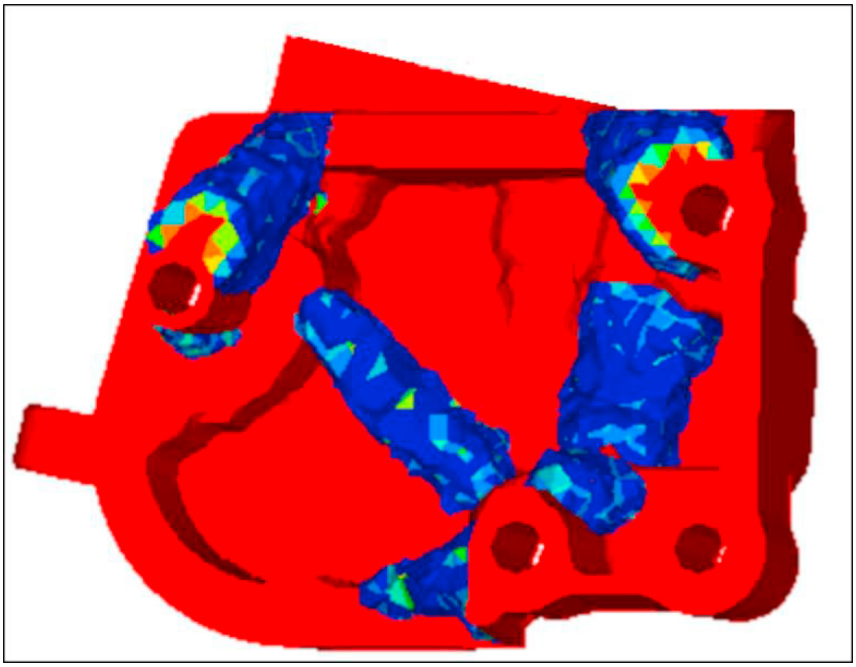


Fig. 4. Resulting design shape of the bracket from the TO process.

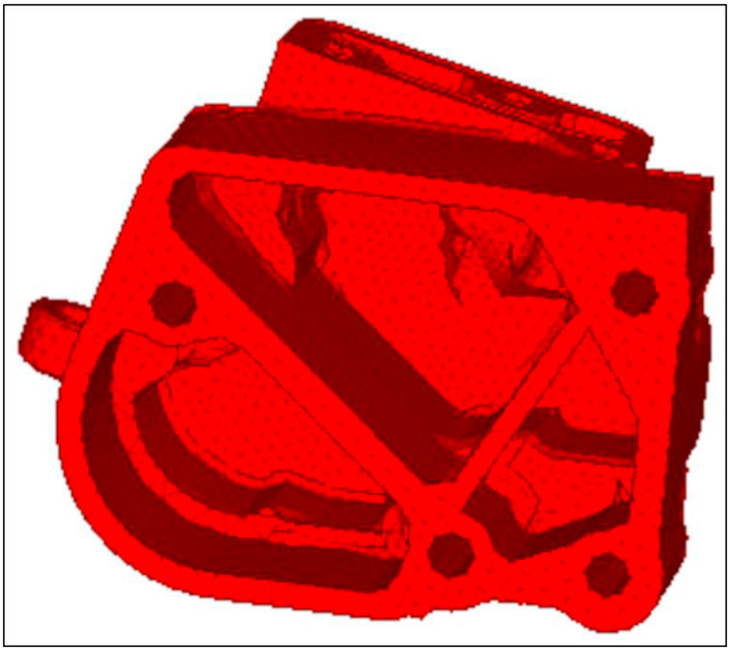
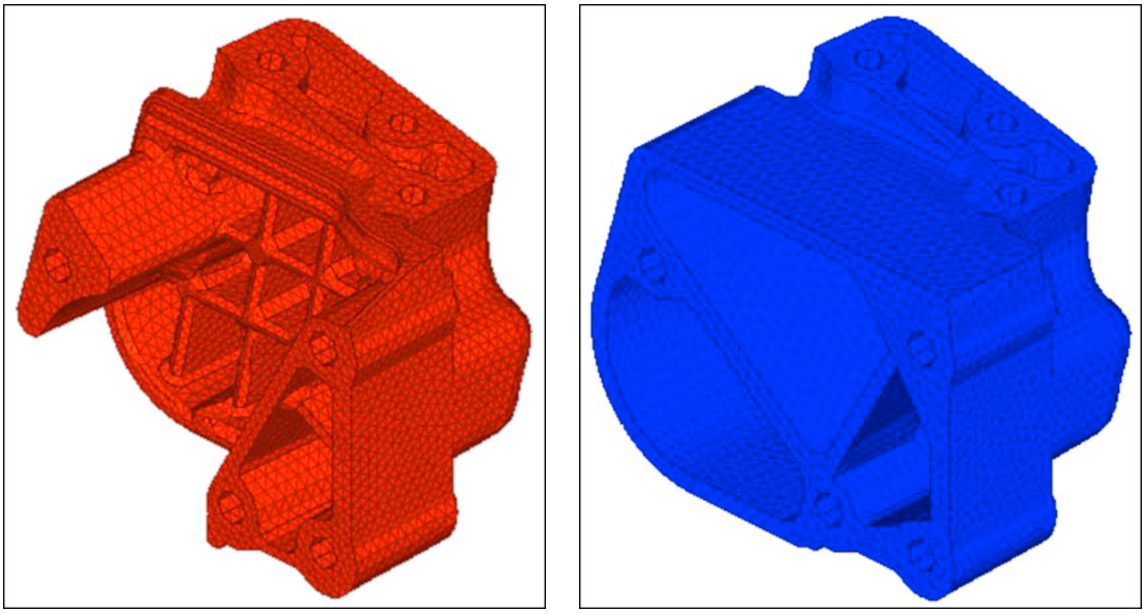


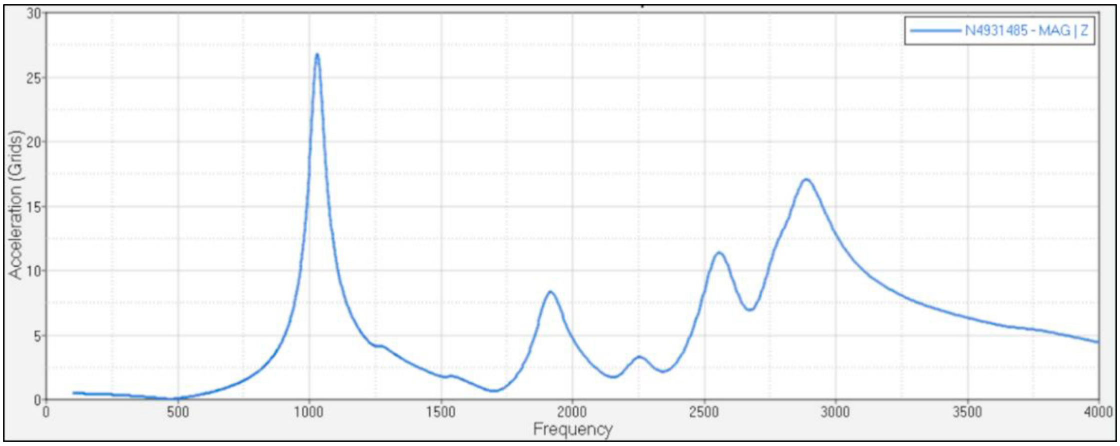
Fig. 5. Post optimization design shape of the bracket built up from the TO result.

A comparison between the initial bracket and the new bracket, obtained from the post-optimization process, was then performed. The FRFs obtained with these two design shape are provided in Fig. 6 whereas the data in terms of weight and first natural frequency are reported in Table 2 for both bracket supports. Also the two FRFs are reported in Fig. 7 for both initial and post-optimization brackets.

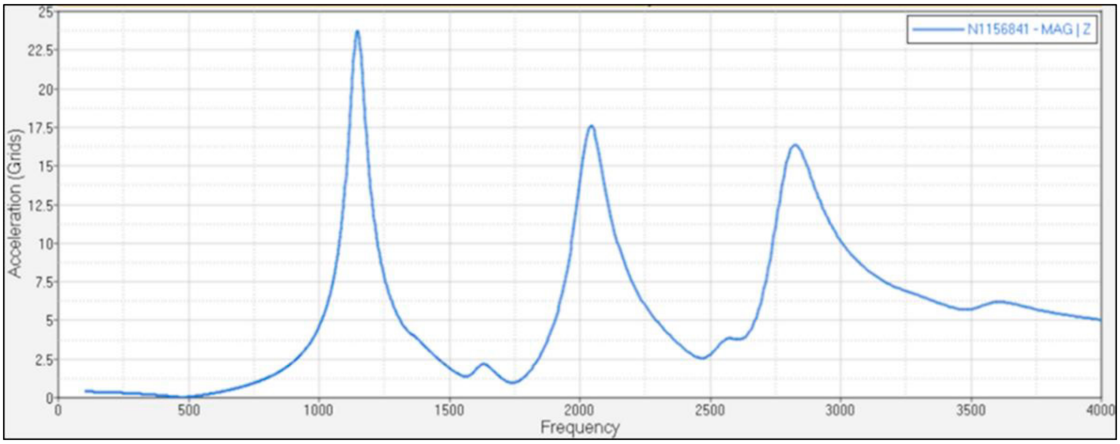


(a) (b)

Fig. 6. (a) Pre and (b) final optimization design shapes of the bracket.



(a)



(b)

Fig. 7. FRFs for (a) initial and (b) post-optimization bracket.

Table 2. TO results: comparison of results between pre and post optimization design shapes of the bracket.

Design	Weight	First natural frequency
Pre-optimization bracket	1243 g	1024 Hz
Post-optimization bracket	989 g **(-20%)**	1122 Hz **(+10%)**

5. Conclusions

The main goal of this work was to perform a Topology Optimization (TO) of an engine bracket support based on its vibrational behavior, in order to reduce the vibrations transferred from the engine to the chassis, with a consequent improvement for the passengers' comfort. In particular, the increase of the first natural frequency of the bracket support was the target function of the TO, whereas the bracket mass reduction was considered as a constraint function.

The vibrational characterization of the bracket support was based on FRF analyses conducted via FEM that allowed to identify the resonant frequencies of the different FEM models built up during the optimization process. Such FEM models included the cylinder head to constraint the bracket support under optimization and a second bracket on which the load was applied.

The TO process turned out to be effective in lowering the engine bracket weight of nearly 20% and increasing also the first natural frequency of nearly 10%, in such a way to guarantee a higher comfort for passengers.

References

Altair Engineering, Inc., 2002. OptiStruct user's manual, version 5.1. Troy, MI.
Armentani, E., Caputo, F., Esposito, L., Giannella, V., Citarella, R., 2018. Multibody Simulation for the Vibration Analysis of a Turbocharged Diesel Engine. Appl. Sci. 8 (7), pp. 1192.
Armentani, E. Sbarbati, F., Perrella, M., Citarella, R., 2016. Dynamic analysis of a car engine valve train system. Int. J. of Vehicle Noise and Vibration, Vol.12, No.3, pp. 229-240.
Armentani, E., Sepe, R., Parente, A., Pirelli, M., 2017. Vibro-Acoustic Numerical Analysis for the Chain Cover of a Car Engine, Applied Science, 7 (610), 1-11.
Armentani E., Trapani R., Citarella R., Parente A. and Pirelli M., 2013. FEM-BEM Numerical Procedure for Insertion Loss Assessment of an Engine Beauty Cover, The Open Mechanical Engineering Journal 7, 27-34.
Bendsoe, MP. 1989. Optimal shape design as a material distribution problem. Struct Optim 1:193-202.
Bendsoe, MP. 1995. Optimization of structural topology, shape, and material. Springer, Berlin Heidelberg New York.
Bendsoe, M.P., Kikuchi, N., 1988. Generating optimal topologies in structural design using a homogenization method. Comput Methods Appl Mech Eng 71, pp. 197-224.
Citarella, R., Federico, L., 2018. Advances in Vibroacoustics and Aeroacustics of Aerospace and Automotive Systems, Appl. Sci. 8, 366.
Citarella, R., Landi, M., 2011. Acoustic analysis of an exhaust manifold by Indirect Boundary Element Method, The Open Mechanical Engineering Journal 5, 138-151.
Harzheim, L., Graf, G., 1995. Optimization of engineering components with the SKO method. Proceedings of the ninth international conference on vehicle structural mechanics and CAE. April 4–6, Troy, MI, pp. 235-243.
Harzheim, L., Graf, G., 1996. Shape and topology optimization in automotive industry. Proceedings of the 3rd international conference on high performance computing in automotive industry. October 7–10, Paris, pp. 167-182.
Harzheim, L., Graf, G., Klug, S., Liebers, J., 1999. Topology optimization in practice. ATZ Worldw 7–8, pp. 11-18.
Hassani, B., Hinton, E., 1998. A review on homogenization and topology optimization: I. Homogenization theory for media with periodic structure. II. Analytical and numerical solution of homogenization equations. III. Topology optimization using optimality criteria. Comput Struct 69, pp.707-756.
Ma, Z-D., Kikuchi, N., Cheng, H-C., Hagiwara, I., 1992. Topology and shape optimization technique for structural dynamic problems. Recent Adv Struct Mech-ASME PVP-Vol. 248/NE-Vol. 10, pp. 133-143.
Mlejnek, H.P., Schirrmacher, R., 1993. An engineer's approach to optimal material distribution and shape finding. Comput Methods Appl Mech Eng 106, pp. 1-26.
Pederson, P., 1998. Influence from non-linearities on optimal shape design. 7th AIAA/USAF/NASA/ISSMO symposium on multidisciplinary analysis and optimization, Part 2, pp. 1123-1130.
Pederson, P., 2000. On optimal shapes in material and structures. Struct Multidiscipl Optim 19, pp. 169-182.
Rozvany, G.I.N., Bendsoe, M., Kirsch, U., 1995. Layout optimization of structures. Appl Mech Rev 48, pp. 41-119.
Siano, D., Citarella, R., 2014. Elastic Multi Body Simulation of a Multi-Cylinder Engine. The Open Mechanical Engineering Journal 8, pp. 157-169.
Siano, D., Citarella, R., Armentani, E., 2018. Simulation of the vibrational behaviour of a multi-cylinder engine. International Journal of Vehicle Noise and Vibration 14(2), pp. 101-123.
Yang, R.J., 1997. Multidiscipline topology optimization. Comput Struct 63, pp. 1205-1212.
Zhou, M., Pagaldipti, N., Thomas, H.L., Shyy, Y.K. 2004. An integrated approach to topology, sizing, and shape optimization. Struct Multidisc Optim 26, pp. 308–317.
Zhou, M., Rozvany, G.I.N., 1991. The COC algorithm: Part II: topological, geometrical and generalized shape optimization. Comput Methods Appl Mech Eng 89, pp. 197-224.