DEVELOPMENT OF FUEL FILTER CAP OF COMET DIESEL ENGINE USING STRUCTURAL OPTIMIZATION TECHNIQUE AND ADDITIVE MANUFACTURING

Hanmant Shete* - Vinay Chavan

Mechanical Engineering Department, Ashokrao Mane Group of Institutions, 416112, Kolhapur, Maharashtra, India

ARTICLE INFO

***Article history*:**
Received: 09.03.2023.
Received in revised form: 13.05.2023.
Accepted: 15.05.2023.

Keywords:
Comet diesel engine
Weight
Finite element method
Topology optimization
Size optimization

DOI: https://doi.org/10.30765/er.2115

Abstract:

As market competition and computational technologies grow, engineering design and development process heavily relies on the computer modelling, simulation and prototype manufacturing to accelerate the development cycles and to save the cost. This article intends to use finite element method as a structural optimization tool and 3D additive manufacturing for manufacturing of prototype to optimize the weight and strength of fuel filter cap of 5HP, 1500 rpm water cooled comet diesel engine within the lesser time cycle. The developed fuel filter cap is manufactured through the stages of pattern making using 3D printing machine, casting of cap by sand casting process, finishing operations and subsequently tested on the comet diesel engine testing set up. The development work has resulted in 48.67% stress reduction and 17.87% reduction in the weight of the fuel filter cap.

1 Introduction

A compact and lighter engine has become a need of every manufacturer. Therefore, designers have been continuously finding the means to reduce the overall engine weight in order to improve the engine performance, power to weight ratio and eventually fuel efficiency. In this scenario, optimization has become an integral part of the product development cycle. Weight reduction of an IC engine is the challenging task for engine manufacturing industry to meet the strategic requirements of material saving, energy saving, and eco-friendliness. Weight reduction can be achieved by using better mechanical properties, better manufacturing processes and design optimization. Structural optimization plays a vital role in the design, which has various types such as topology, size and shape optimization [1]. Numerical methods, especially finite element method is preferred to solve the structural analysis problems. Different mathematical software’s are widely used to obtain an accurate solution of complex optimization problem within the short duration of time.

In [1] optimization of front door structure of an automotive was performed using topology, size and shape optimizations and major weight reduction was obtained by size optimization. Topology optimization of front platen of injection moulding machine was undertaken using finite element analysis to reduce volume of platen and improve the stiffness as well as strength [2]. Topology optimization analysis was performed for fix jaw of rear vice of horizontal band saw machine using Hyper Work software to reduce the weight [3]. The static analysis of a cast iron crankshaft of a single cylinder four stroke engines using finite element method was executed to obtain variation of stresses at the critical locations and results obtained were employed for optimization of crankshaft [4]. Genetic Algorithm Optimization (GAO) approach was applied to multi-objective engineering design problem and design variables were determined [5].

The genetic approach was studied for the modelling and creation of a complex technical system of different physical nature in relation to kinematics of cutting and shaping [6]. The developments and trends in aluminium alloy sheets were studied for the automotive body panels of passenger cars to reduce the weight [7]. In [8], some methods were explained that can allow the end user to generate the structural design proposals suitable for 3D printing manufacturing with minimum changes. The computer-aided engineering tools and finite

* Corresponding author
E-mail address:sheteaditya@yahoo.co.in

element method were employed for designing and evaluating the performance of the robot arm and the study proposed a surrogate-based evolutionary optimization method, which integrates the response surface method and multi-objective evolutionary algorithm by decomposition for the shape design optimization of the robot arm [9]. A review of the general topology optimization and modern topology optimization was presented in [10]. Generally, a structural optimization problem aims to minimize the objective function under the given constraints. The objective is commonly the weight of structure, and constraints are imposed on the structural responses such as stresses, displacements. The topology optimization problem used in modern topology optimization [11] can be represented as,

$$\begin{gathered} \textit{Minimise: } H(u(\boldsymbol{x}), \boldsymbol{x}) \\ \textit{Subject to: } G_j\big((\boldsymbol{x}), \boldsymbol{x}\big) \geq 0 \quad \textit{for } j = 1, 2, \ldots\ldots, p \\ 0 \leq x_e \leq 1 \quad \textit{for } e = 1,2, \ldots\ldots, N \end{gathered} \tag{1}$$

where x is the design variable vector, u is the state field, H is the objective function, G_j is the j^{th} constraint, p is the total number of constraints, x_e is the e^{th} structural member that constitutes the design variable vector x, N is the total number of design variables (structural members). The equation (1) can be solved by using the homogenization method that determines the optimum material distribution by the homogenization of a microstructure [12]. The density method uses an energy approach for the approximation of the elements Young's moduli [13]. These methods have been adopted in the software systems such as ANSYS [14], OPTISHAPE [15], OptiStruct [16], GENESIS [17] and CONSTRUCT [18].

The size optimization problem is represented as,

$$\begin{gathered} \textit{Minimise: } W(X) \\ \textit{Subject to: } K(X)u = p \\ K(X)y = \zeta M(X)y \\ g_j(X) \leq 0, j = 1, \ldots\ldots, m \\ X_L \leq X \leq X_U \end{gathered} \tag{2}$$

X is the design variable vector, X_L and X_U are lower bound and upper bound vectors respectively, W(X) is the objective function, K(X) is the stiffness matrix, M(X) is the mass matrix, u is the displacement vector, p is the external load vector, ξ is the eigenvalue, y is the eigenvector and g_j is the j^{th} constraint out of m constraints. In size optimization, the design variables are the dimensions such as height, width, thickness, area, and moment of inertia. Additive manufacturing is recently employed for design and manufacturing of products in different fields. Song et al. presented fabrication of 3D metal–plastic composite structures demonstrating production of metal patterns inside plastic parts [19]. Fongsamootr et al. studied the effects of print parameters on mechanical properties of parts fabricated using additive manufacturing [20]. Jandyal et al. reviwed 3D printing processes, different materials of 3D printing process, and applications of each type of process [21]. The present work is sponsored by the company, which is a globally leading company for manufacturing of the comet diesel engines of wide variety and has different patents in the field of manufacturing of diesel engines. Weight and strength optimization of fuel filter cap of 5HP, 1500 rpm water cooled diesel engine (Figure 1) was one of the tasks in the overall optimization of this engine and therefore, this work presents the achievements of the same using the better material as well as topology and size optimization performed with ANSYS tool, which employs finite element method (FEM).

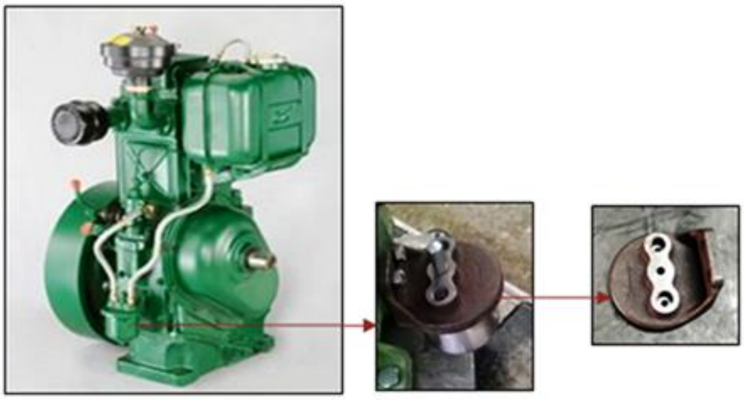


Figure 1. Comet diesel engine and existing fuel filter cap.

2 Experimental investigation

2.1 Development of fuel filter cap using structural optimization

The dimensions and parts of existing (original) fuel filter cap are given in Figure 2. Material used for existing fuel filter cap is FG210 grey cast iron. In order to optimize the weight and strength of cap, finite element analysis (FEA) is performed using ANSYS software to determine the possible scope for application of topology, size and shape optimization. FEA is performed in the steps of CAD modelling, meshing, applying boundary conditions, applying load, determining equivalent stresses in existing cap. Then, reference model of cap is obtained based on topological optimization. Based on equivalent stresses and reference model, optimization variables are selected and new CAD model of cap is developed. Further, meshing, applying boundary conditions, applying load, determining equivalent stresses and deformations is performed for this newly developed model of cap.

The CAD model of existing filter cap is meshed using trapezoidal elements, and details of meshing are given in Table 1. The load on cap is only self-weight of cap in assembled position, wherein inlet and outlet fuel pipes are connected to the cap. As self-weight of cap in assembled position is 25 Kg, static point load of 245 N is applied as shown in Figure 3 (a). Fixed supports are applied at two holes in the vertical rib and stresses are obtained as depicted in Figure 3 (b). The maximum stress of 97.62 MPa occurs at the joint of vertical side ribs and base. The obtained maximum stress and weight of existing fuel filter cap by FEM is given in Table 2. In present study based on Figure 3 (b), topology optimization of fuel filter cap of an engine is carried using finite element method, which employs homogenization method to determine the optimum material distribution. The resulting optimized reference model of cap is shown in Figure 4. For the size optimization of fuel filter cap, the design variables can be the cross section, thicknesses, and width, height of base plate and ribs or height and diameter of the cylindrical parts. From Figure 3 (b) and Figure 4, thickness of the rib (7.5 mm), height of rib (47 mm) and cylindrical part (25 mm) on base plate is selected as the design variable for size optimization of cap. It is decided to remove the material and provide a slot in the region of vertical rib between two holes (topology optimization) and to reduce the thickness of the rib, height of rib and cylindrical part on base plate (size optimization) by 1mm. Also material can be added (topology optimization) at the two upper most extreme corners of ribs to avoid the rise in stresses. Considering these modified values of variables and topology, fuel filter cap is developed as shown in Figure 5. The design variables for existing cap and developed cap are specified in Figure 2 and Figure 5.

The material FG210 of existing cap is replaced by FG260 grey cast iron material, which is having more bending strength. It has higher compressive strength and deformation resistance than FG210 material. Therefore, a new model of fuel filter cap of modified geometry and material is developed and analysed in ANSYS to obtain the stresses and deformations as shown in Figure 6 and 7, respectively. The maximum stress of 48.947 MPa occurs at the edges of two holes in vertical rib and the maximum deformation of 0.326 mm is observed at the hanging end. The maximum stress and weight in the developed fuel filter cap by FEM is given in Table 2. The error in FEA is 0.05% which is acceptable and hence simulation results are correct.

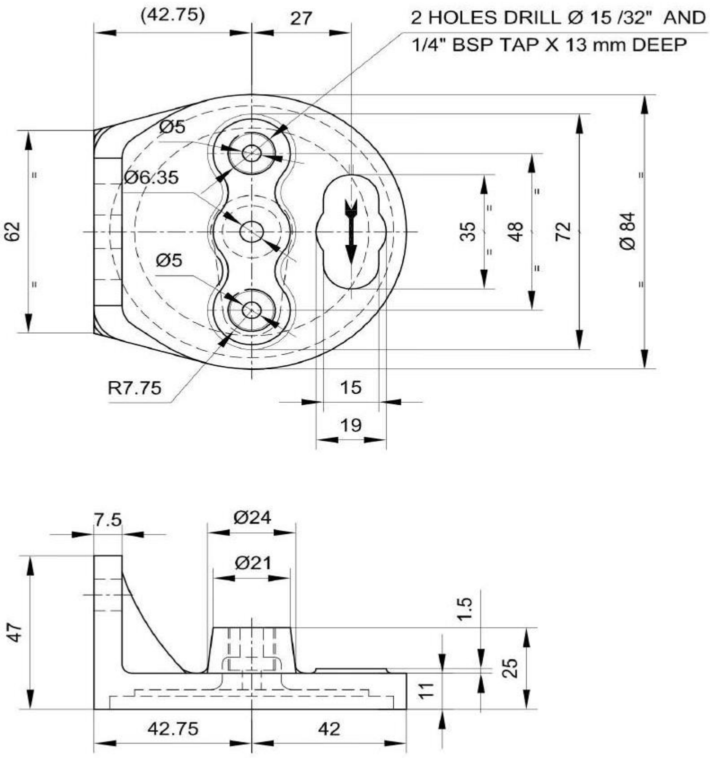
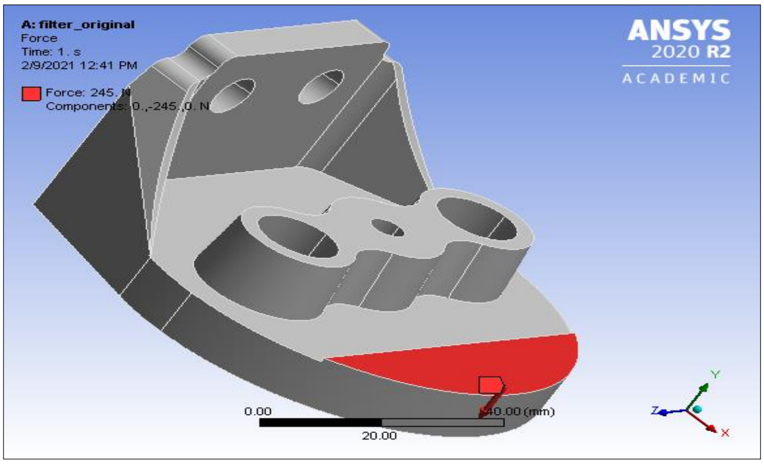


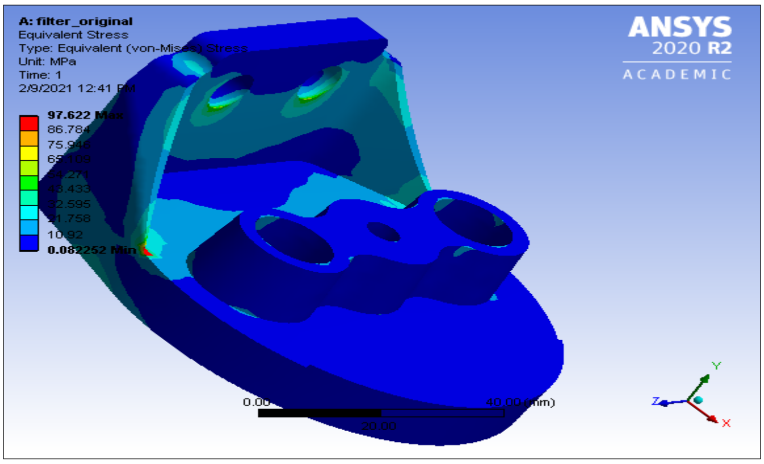
Figure 2. Dimensions of existing fuel filter cap.

Table 1. Mesh details for existing and developed fuel filter cap.

Details	Existing fuel filter cap	Developed fuel filter cap
Type	Trapezoidal elements	Trapezoidal elements
Mesh refinement method	P-Refinement	P-Refinement
Statistics of meshing		
Nodes	35813	19924
Elements	20073	10782
Bounding Box		
Length X (mm)	84.75	89.5
Length Y (mm)	47	42.259
Length Z (mm)	84	85.985
Properties		
Volume (mm³)	70612	58127
Mass (kg)	0.5543	0.41851



(a)



(b)

Figure. 3 Structural analysis of the existing fuel filter cap.

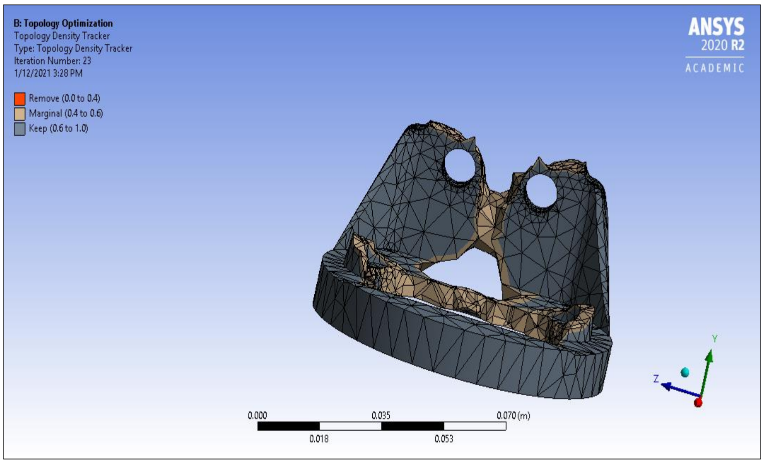


Figure 4. Reference model of fuel filter cap using ANSYS.

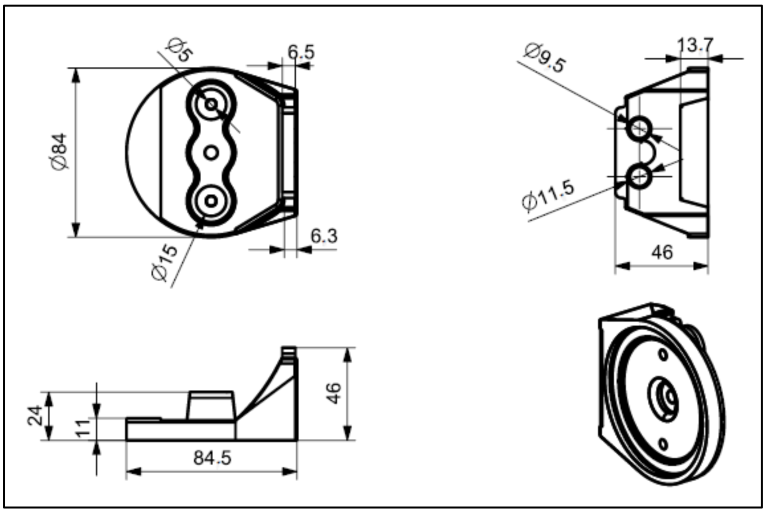


Figure 5. Dimensions of developed fuel filter cap.

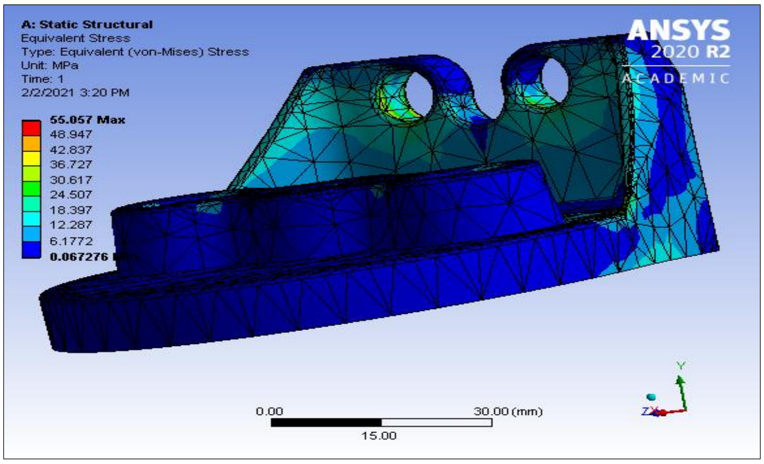


Figure 6. Structural analysis of the developed fuel filter cap.

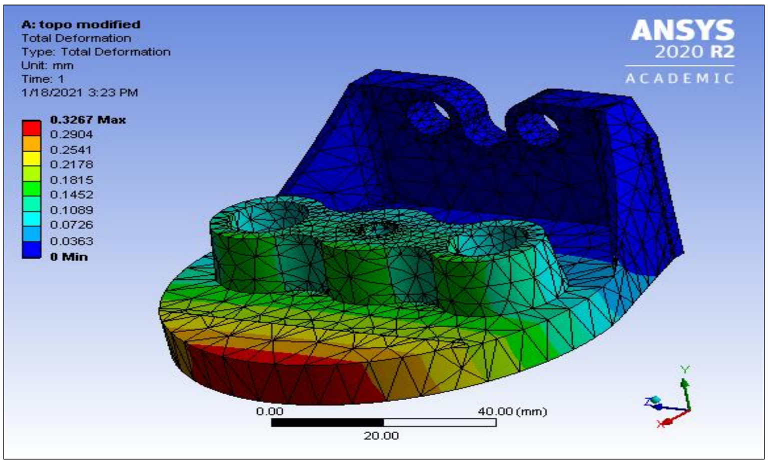
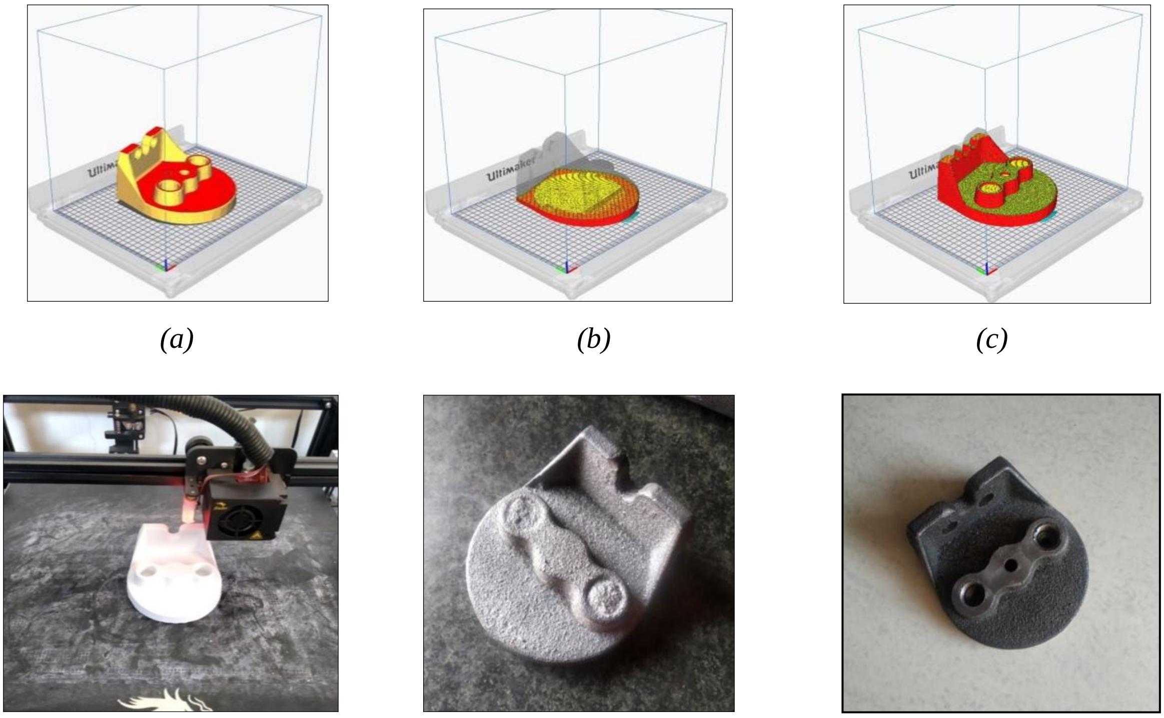


Figure 7. Structural analysis of the developed fuel filter cap.

2.2 Manufacturing of the developed fuel filter cap

3D Printing technology, known as Additive Manufacturing (AM) is employed to manufacture the pattern of developed fuel filter cap accurately within short period of time as shown in Figure 8 (d). The model of developed fuel filter cap is generated using UNIGRAPHICS software and imported in 3D printing machine database as shown in Figure 8 (a).



(a) *(b)* *(c)*

(d) *(e)* *(f)*

Figure 8. Additive manufacturing of the developed fuel filter cap.

Acrylonitrile butadiene styrene (ABS) material is used for 3D printing of pattern. It is light, has good impact strength, abrasion resistance and affordable thermoplastic polymer material. Specifications of 3D printing are; model: Ender 5 pro; moulding technology: Fused Deposition Modelling (FDM); print dimensions: 280 x 280 x 280 mm; print speed: 180mm/s; print accuracy: ±0.1mm; layers thickness: 0.1-0.4mm; nozzle diameter: standard 0.4 mm; nozzle quantity: 1; infill density: 40%; infill pattern: triangular; build orientation: X-Y; bed temperature: 110^{0}C. Some of the successive slicing of model in the process of 3D printing is shown in Figure 8 (b, c). The casting of fuel filter cap is produced as given in Figure 8 (e) by using sand cast moulding process and then facing, boring of two holes, all drilling operations like counter boring, countersinking, reaming, and tapping are performed on vertical machining centre (VMC) to produce the developed fuel filter cap as shown in Figure 8 (f).

3 Results and discussion

Testing of the developed fuel filter cap is performed on 5HP, water cooled engine (model no. 20H 15737) test set up (Figure 9) in the engine development laboratory of the sponsoring company at the test conditions of 1500 rpm, 22.6 N-M torque, 332^{0}C exhaust temperature, 88^{0}C oil temperature and for the 5 cycles each of 8 hour time duration. The engine operation is observed to be satisfactory as per test norms with no failures in the developed fuel filter cap.

Figure 9. Testing of the developed fuel filter cap on engine testing set up.

The obtained results of finite element analysis for the existing and developed fuel filter cap are presented in Table 2. Figure 7 shows that the reduced stresses are observed in the developed fuel cap and maximum stress is 48.67% lower than the existing filter cap. The maximum deformation of 0.326 mm is observed at the hanging end of the developed fuel cap, which is acceptable. Weight of the developed cap is reduced to 17.87% than the existing fuel filter cap.

Table 2. Comparison of results of existing and developed fuel filter cap.

Parameter	Existing fuel filter cap	Developed fuel filter cap	% Change
Max. Stress by FEM (N/mm^2)	97.62	48.947	48.673
Weight by FEM (Kg)	0.548	0.465	17.84
Actual Weight (Kg)	0.550	0.469	17.87

4 Conclusion

The development work of the fuel filter cap of comet diesel engine is performed in stages of FEA analysis of existing cap, size and topology optimization based on reference model, and FEA of developed fuel filter cap. The pattern of cap is developed using modern 3D printing machine technology to increase the accuracy and to reduce the cost of development work. Subsequently, casting of cap is produced using sand casting process and finishing operations are carried on VMC. The development work has resulted 48.67% stress reduction and 17.87% reduction in the weight of fuel filter cap. Therefore, the work presents the role of structural optimization and advanced 3D printing in product development cycle and is concerned to the direct usefulness to the industry sector by improving the strength and reducing the weight of fuel filter cap of an engine.

Acknowledgement

The authors are grateful to the Rocket Engineering Pvt. Ltd., Shiroli, India for providing the testing facility existing in their engine testing laboratory.

References

[1] K. H. Lee, J. K. Shin, S. Song, Y. M. Yoo, and G. J. Park, “Automotive door design using structural optimization and design of experiments”, *Proc. Instn Mech. Engrs, Part D: J. Automobile Engineering*, vol. 217, pp. 855-865, 2003.

[2] E. G. Zhang, “Finite element analysis and Topology optimization design for front platen of injection molding machine”, *Applied Mechanics and Materials*, vol. 365-366, pp. 42-46, 2013.

[3] N. V. Hargude, and P. H. Patil, “Cost reduction of fix jaw of rear vice of horizontal band saw machine using topology optimization”, *International Journal of Science and Research,* vol. 3(8), pp. 268-272, 2014.
[4] R. Garg, and S. Baghla, “Finite element analysis and optimization of crankshaft design” *International Journal of Engineering and Management Research,* vol. 2(6), pp. 26-31, 2012.
[5] A. Khazaee, and H. M. Naimi, “Two multi-objective genetic algorithms for finding optimum design of an I-beam”, *Engineering,* vol. 3(10), pp. 1054-1061, 2011.
[6] N. Musa, “The use of genetic approach to the kinematics of cutting”, *World Journal of Mechanics,* vol. 6, pp. 396-405, 2016.
[7] M. S. Reddy, and N. N. Krishna, “Trends in weight reduction of automobiles: Alu – maximized”, *International Journal of Engineering Research and General Science,* vol. 3(3), pp. 1-6, 2015.
[8] J. P. Leiva, H. Dong, and B. Watson, “Structural optimization methods and techniques for additive manufacturing”, *The World Congress of Structural and Multidisciplinary Optimization,* May 20-24, 2019, Beijing, China, pp. 1-6.
[9] J. C. Hsiao, K. Shivam, C. L. Chou, and T. Y. Kam, “Shape design optimization of a robot arm using a surrogate-based evolutionary approach”, *Appl. Sci.*, vol. 10, pp. 1-17, 2020.
[10] O. Yuksel, An overview on topology optimization methods employed in structural engineering, *Yüksel/Kırklareli University Journal of Engineering and Science,* vol. 5(2), pp. 159-175, 2019.
[11] O. Sigmund, and K. Maute, “Topology optimization approaches”, *Structural and Multidisciplinary Optimization”,* vol. 48(6), pp. 1031-1055, 2013.
[12] M. P. Bednsoe, and N. Kikuchi, “Generating optimal topologies in structural design using a homogenization method”, *Computer Meth. In Appl. Mechanics and Engng,* vol. 71, pp. 197–224, 1988.
[13] H. P. Mlejnek, and R. Schirrmacher, “An engineer’s approach to optimal material distribution and shape finding”, *Computer Meth. In Appl. Mechanics and Engng,* vol. 106, pp. 1–26, 1990.
[14] See http://www.ansys.com/
[15] See http://www.quint.co.jp/
[16] See http://www.altair.com/
[17] See http://www.vrand.com/
[18] See http://www.mscsoftware.com/
[19] K. Song, Y. Cui, T. Tao, X. Meng, M. Sone, M. Yoshino, S. Umezu, and H. Sato, “New metal–plastic hybrid additive manufacturing for precise fabrication of arbitrary metal patterns on external and even internal surfaces of 3D plastic structures”, *ACS Applied Materials & Interfaces,* vol. 14(41), pp. 46896–46911, 2022.
[20] T. Fongsamootr, I. Thawon, N. Tippayawong, K. Y. Tippayawong, and P. Suttakul, “Effect of print parameters on additive manufacturing of metallic parts: performance and sustainability aspects”, Sci Rep 12, Article no. 19292, 2022.
[21] A. Jandyal, I. Chaturvedi, I. Wazir, A. Raina, and M.I.U. Haq, “3D printing – A review of processes, materials and applications in industry 4.0”, *Sustainable Operations and Computers,* vol. 3, pp.33-42, 2022.