International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 12, Number 12 (2019), pp. 2544-2549

Design of a Free-running Wheel Base for a Differential Robotic Platform Using Topology Optimization

Fernando Martínez Santa[1], Fredy H. Martínez S.[2] and Edwar Jacinto Gómez[3]

Facultad Tecnológica, Universidad Distrital Francisco José de Caldas, Bogotá D.C, Colombia.

[1]*ORCID:* 0000-0003-2895-3084, [2]*ORCID*: 0000-0002-7258-3909

[3]*ORCID:* 0000-0003-4038-8137

Abstract

In mechanical design used to prevail the aesthetic guidelines instead of performance requirements. The work shown in this paper uses topology optimization for designing the base for a free-running wheel of a differential robotic platform. This optimization algorithm is based on FEM (Finite Element Method), specifically the Von Mises stress analysis. This work starts from a previous robotic design, done to be made of acrylic sheets by a laser cutter, then the optimization is applied to the bi-dimensional structures that compose it, in order to reduce the amount of material used and eventually the final weight of the mechanical structure. Finally, after apply the topology optimization algorithm with different desired volume fractions, reductions from 43.3% to 56.7% in the used material were obtained over the final design of the freerunning wheel base of the robot.

Keywords: Structural design, Topology optimization, Mobile robot, Differential platform, Finite Element Method.

1. INTRODUCTION

Most of the designs of the mechanical devices are developed following some performance requirements, but mainly they follow aesthetic guidelines. Sometimes, the amount of materials for building those designs are limited, so it is necessary to prioritize the material optimization over the aesthetic features [1]–[5]. In those design processes the FEM (Finite Element Method) is usually applied to verified that it meets the standards of the mechanical design [6], [7], this method computes the equations that govern the system in order to obtain the behavior of the variables inside the structure in a specific key points (mesh). For instance, applying FEM it is possible to check the heat distribution in a heat sink design [8], [9], or the inner stiffness in a mechanical structure one [10], [11]. Frequently, it requires some trial and error steps, which widely delays the design process, due to that some automatic algorithms have been proposed to improve this time, like topology optimization. This last consists of applying over a specific mechanical structure an iterative algorithm in order to reduce its inner size and thus doing an area or volume (material) optimization. All of this reduction process has to guarantee that the resultant structure accomplishes the performance requirements of the design, being supported by the FEM simulations done on each iteration. Several evolutionary and bio-inspired algorithms have been used for implementing topology optimization such as genetic algorithms [12] –[14] and artificial neural networks [15], [16]. Topology optimization has been applied on several designs in architecture and engineering, such as design optimization of bridges [17], optimum design of chassis for cars [18] –[20] and motorcycles [21], [22] design of other vehicles and parts [23], [24], and design of wheels for different kind of vehicles [25] –[27]. Also in robotics those algorithms have been applied for the optimum design of part and joints [28] for industrial and mobile robots [29], and for humanoids [30]. The work shown in this paper is focused on the optimization of the mechanical structure of a wheeled mobile robot, specifically the free-running wheel used in a differential platform. The main idea is to optimize the amount of material used for building the base of the wheel over a previous design [31] done for a laser cutter. This design is optimized using several 2D topology optimizations and finally represented as a 3D object.

2. MATERIALS AND METHODS

This work is based on the robotic platform named SERB [31], which was designed to be made by means of a laser cutter. That platform is composed by a simple embedded system and two servomotors. By and large, the structure of SERB robot is not that heavy if it is compared with the rest of its components, due to it is made of acrylic sheets. However, if the design is wanted to be made of another stronger material like any metal, its weight could increase considerably. That is why the purpose of this work is to optimize the amount of material of the structure and so its weight, starting from the base of the free-running wheel of the robot, which works as the third support point (shown in Fig. 1). The wheel base is composed by three different parts, the top one and two identical sides, which fit themselves through square holes cut on the acrylic sheets, as shown in Fig. 2. Also, those shapes shown the t-shaped holes done for a couple of screws and nuts used to fix all the parts together.

The main idea is to do some simulations using FEM to the top and side shapes of the wheel base, in order to define the location of the main stresses presented on them, when external forces are applied, emulating the loads presented during the regular working of the robot. After that, those results are used to optimize the shape, reducing the amount of its material, using topology optimization. The FEM simulations and the topology optimization are performed by using BESO2D free software, as well as freeCAD software was used with the purpose of displaying the initial and obtained designs in 3D.

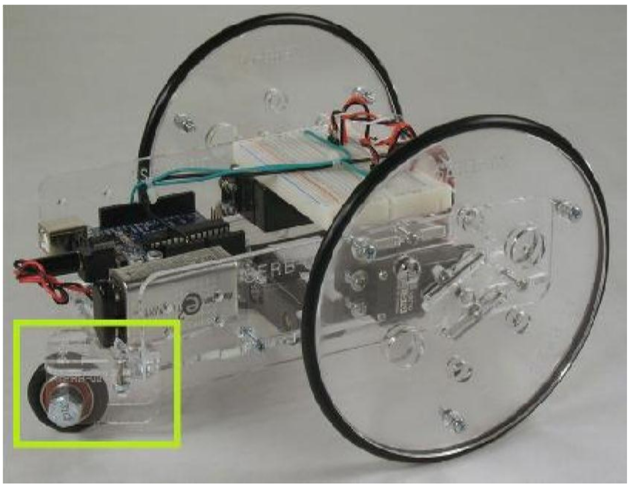

Fig. 1. Robot structure to be optimized (the free-running wheel in the square) [31].

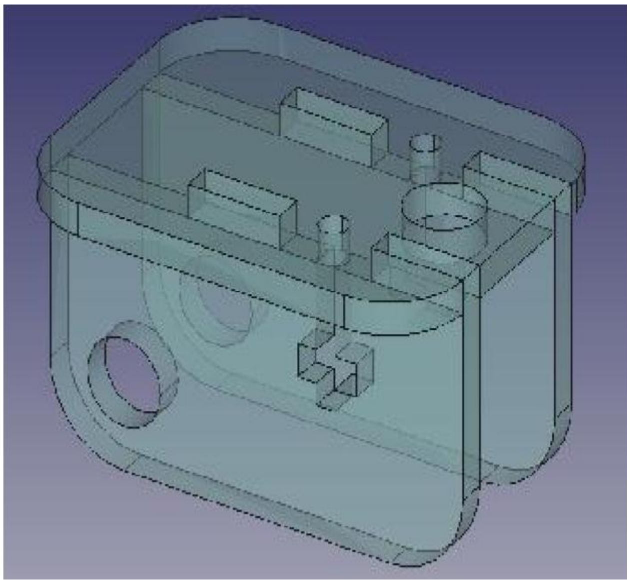

Fig. 2. 3D representation of the free-running wheel base.

3. STRUCTURAL ANALYSIS

Starting from the original design of the free-running wheel base, some FEM simulations were performed in order to detect the critical points after applying external forces. To do that, it was necessary to import the 2D shapes of the top and the side of the wheel base to the FEM software, as well as to define the mesh size and so the number of nodes to be applied to the problem. BESO2D software uses a typical rectangular mesh to solve all the problems, then for these simulations a rectangular mesh of 1x1mm was selected, in order to fit easily to the starting shape and avoid a high amount of nodes to solve. The resultant meshes are shown in Fig. 3 for the left/right side and in Fig.4 for the top side (only half shape due to it is simetrical).

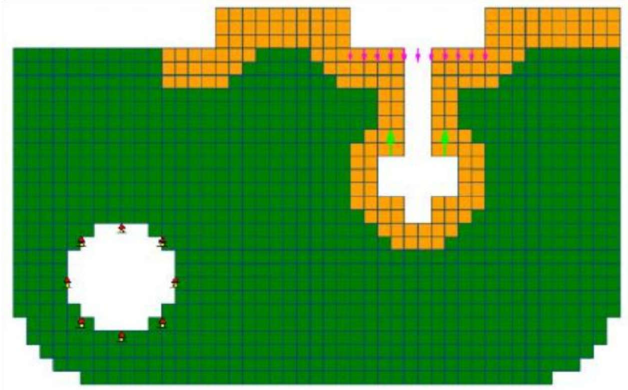

Fig. 3. Proposed rectangular mesh for the left/right side of the wheel base (1x1mm).

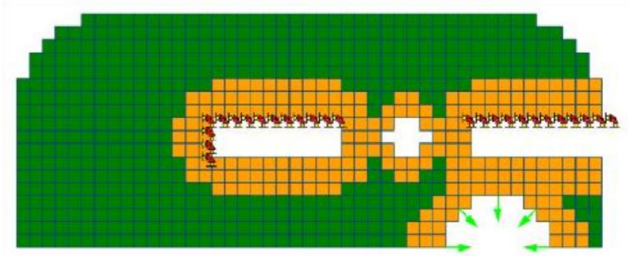

Fig. 4. Proposed rectangular mesh for the top side of the wheel base (1x1mm).

After defining the mesh size, the boundary conditions had to be specified, Fig. 3 and 4 show the constraints as red triangles with a yellow base, the point loads as green arrows and the line loads as magenta arrows. Likewise, the mechanical properties of the material (acrylic), were defined as 6 GPa for the Young's modulus and 0,33 for the Poisson's ratio. For the left/right side shape (Fig. 3) the constrains were located around the circular hole that works as the wheel axis, these eight constraints were defined with only the X degree of freedom. Also, a line load of -10 N (down) was defined in the square groove above the t-shaped hole, which is the edge that supports the robot weight through the top side shape. Additionally, two point loads of 1N (up) each were located in the sides of the t-shaped hole, to represent the force generated when the screws are tight. On the other hand, For the top side shape (Fig. 4) was represented only as the half of the final shape due to it is horizontally symmetrical, that implies it is only necessary to simulate the half and so duplicate it. The constraints were located in the borders of the rectangular grooves where the lateral sides fit, in this case they has X and Y degrees of freedom. This top side shape supports all the weight of the robot, near the circular hole (half circle in this case) that works as the turning point for all the wheel base structure, then 5 point loads of 1N were located on its border pointing to the center, in order to emulate the force applied by the robot weight.

Under these defined features, the Von Mises stress distribution was found through a FEM analysis, obtaining the graphical results shown in Fig. 5 and 6. The resultant values come from 1.2 KPa (blue) to 65 MPa (magenta). Von Mises stress is calculated following the equation (1), where σ_1, σ_2 and σ_3 are the main stresses.

$$\sigma l = \sqrt{\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}} \quad (1)$$

As shown in Fig. 5 and 6, the stress presented under the defined conditions is high near the circular holes, the wheel axis in the case of the left/right side and the turning axis in the case of the top side. Most of the regions of both shapes show a low stress (blue), then they are candidate to be reduced during the optimization process.

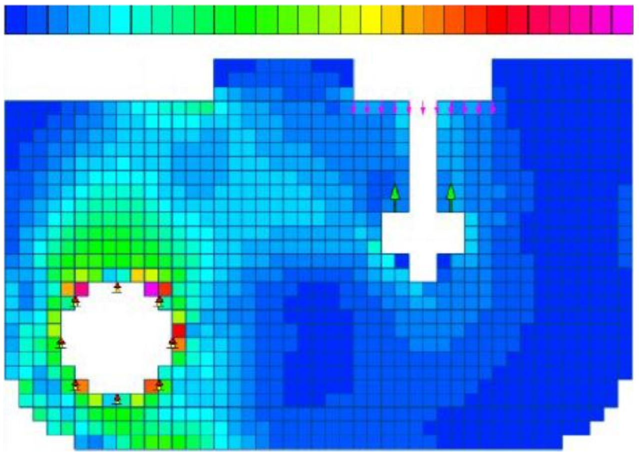

Fig. 5. Von Mises stress distribution for the left/right side of the wheel base.

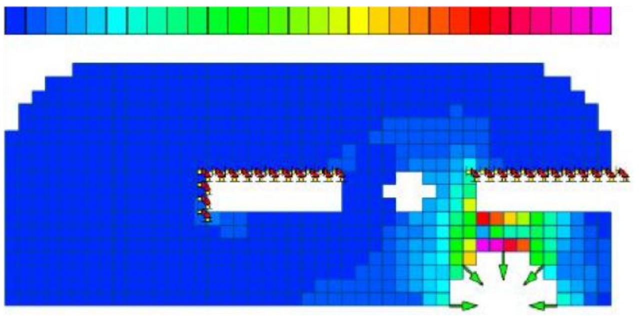

Fig. 6. Von Mises stress distribution for the top side (half) of the wheel base.

4. RESULTS

After doing the structural analysis, the topology optimization algorithm was run, in order to reduce the amount of material of each side of the wheel base. This was done using BESO2D software, which uses an iterative and evolutionary algorithm that reduces step by step the nodes with less stress into the mesh and does a new FEM analysis of the reduced model in each iteration, until obtaining the desired reduction percentage (volume fraction), maintaining the nodes which support the most stress in the structure. As shown in Fig. 3 and 4, the working area is a square green mesh, but some of the nodes are shown in orange, this means that it is a protected area, in other words, the topology optimization algorithm has any reduction effect over these nodes. The protected area was defined taking into account the fitting regions between the different side shapes (rectangular grooves).

For all of the FEM simulations they were established the next features: *evolutionary volume ratio* 2% (maximum volume ratio changed by iteration), *filter radius* 0.3 (action radius), *convergence tolerance* 0.1%, and m*ax iteration number* 200. On the other hand, the *objective volume ratio* that refers to the percentage of material to be maintained in the final design, was changed in order to find the best result with the minimum amount of used material. The Fig. 7 shows the optimization result obtained after applying the algorithm to the left/right side shape, using 45%, 50%, 55% and 60% of volume fraction.

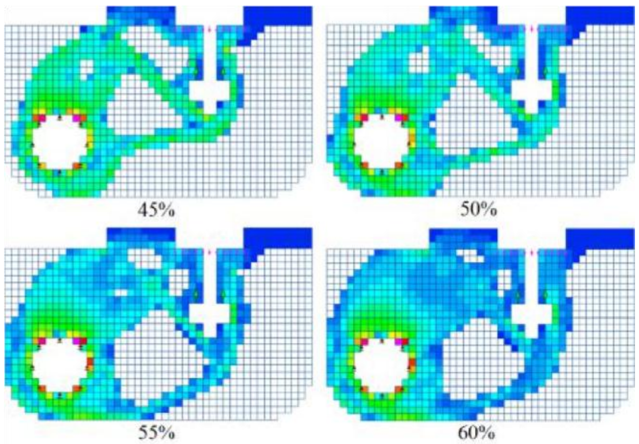

Fig. 7. Results of applying the topology optimization to the right side shape, with 4 different volume fraction.

The same process was applied to the top side shape, but now using as volume fraction of 40%, 60%, 70% and 80%, as the Fig. 8 shows. It is possible to see graphically (according with the color pattern) that the top side shape is able to be reduced more than the left/right side, and in both cases the regions with low stress match with the protected regions.

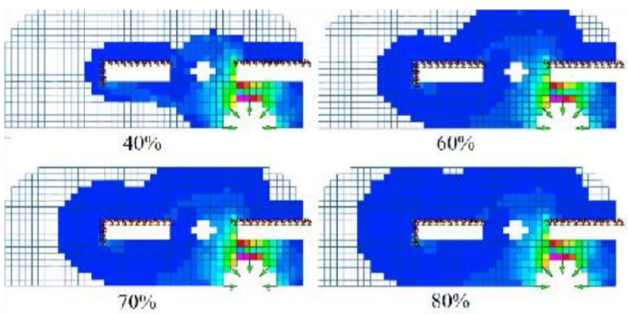

Fig. 8. Results of applying the topology optimization to the top side shape, with 4 different volume fraction.

Finally, a combination of the result of optimizing the left/right side shape using 55% of volume fraction and the one of optimizing the top side shape using 60% of volume fraction, was selected to be represented in 3D by means of using Bsplines in freeCAD, to obtain a model with soft curved edges, ready for the laser cutter. These designed shapes are shown in Fig. 9.

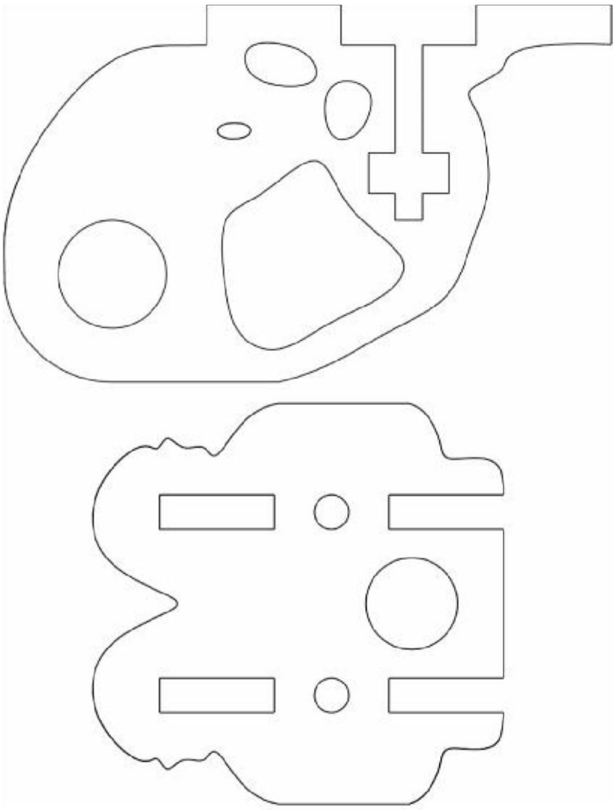

Fig. 9. Optimized shapes for the left/right side (up) and the top side (down).

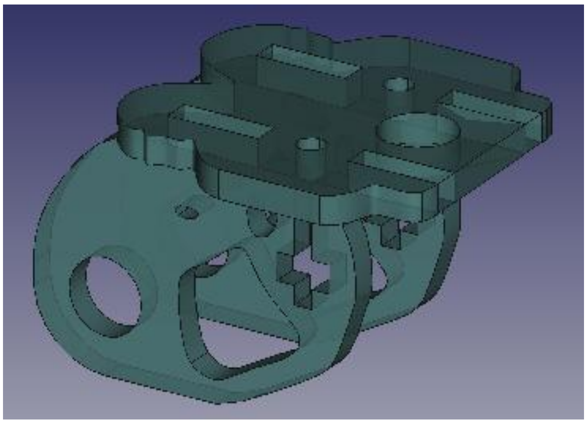

Fig. 10. Obtained optimized 3D structure for the free-running wheel base.

As it is possible to show in Fig. 9 (down) the optimized shape of the top side was mirrored horizontally to generate the complete shape. Considering the acrylic sheet thickness, the final 3D representation was done as shown in Fig. 10. Counting the volume fraction obtained in both optimized shapes (left/right 55%, top 60%), a total of 56.7% of volume fraction was obtained, according to the original volume of the complete base (left, right and top sides), that means a reduction of 43,3% of the used material to made the complete base and therefore a proportional reduction of the weight. As previously stated, the top side shape is able to be reduced more than the left/right one, due to its stress distribution, therefore, if the optimized shape chosen is the one with a volume fraction of 40%, the total volume would be reduced up to 50% or 43.3% if the left/right of 45% of volume is used.

5. CONCLUSION

Significant material reductions were obtained applying topology optimization over the base of the free-running, maintaining the critical structural points, all this supported by Finite Element Analysis specifically computing the Von Mises stress of the structure. This structural stress analysis ensure that the reduction performed after applying the topology optimization, does not affect the structural integrity of the designed mechanical element. Additionally, these material reductions imply a future significant weight reduction when the design is made of heavier and stronger materials like metals instead of acrylic. This optimization approach can turn into a design approach, where the structural requirements prevail over aesthetic parameters; anyways the topology optimization can be used as staring point for designers, in order to take into account the mechanical requirements since the start of the design.

6. FUTURE WORK

Next step is to apply the same optimization algorithm to the rest of the mechanical pieces of the robot base, in order to obtain complete optimized mechanical structure. Additionally, this optimization could be applied to the complete 3 D structure (3D topology optimization) instead of separated sides, this could be really useful if the final design is made by means of using a 3D printer, due to the significant reduction of the used material and the printing time.

ACKNOWLEDGMENTS

This work was supported by the Universidad Distrital Francisco José de Caldas, specifically by the Technological Faculty. The views expressed in this paper are not necessarily endorsed by District University. The authors thank all the students and researchers of the research group ARMOS for their support in the development of this work.

REFERENCES

[1] V. R. R. and E. Acero, “Diseño óptimo de un semirremolque para vehículos de carga,” Tecnura, vol. 16, no. 0, pp. 129–139, Oct. 2012.

[2] H. Patel, K. C. Panchal, and C. S. Jadav, “Structural analysis of truck chassis frame and design optimization

for weight reduction," Int. J. Eng. Adv. Technol., vol. 2, no. 4, pp. 665–668, 2013.

[3] F. Ballo, G. Mastinu, and M. Gobbi, "Lightweight Design of a Racing Motorcycle Wheel," 2016.

[4] K. A. Kumar, J. K. Putra, and V. Monishram, "Analysis and Optimization of Material For KTM Motorcycle (Duke 390) Front Alloy Wheel," Int. Jou rnal Innov. Eng. Technol., vol. 8, no. 2, pp. 113–130, 2017.

[5] G. V. Bhunte and D. T. R. Deshmukh, "A Review on Design and Analysis of Two Wheeler Chassis," Int. J. Res. Emerg. Sci. Technol., vol. 2, no. 1, pp. 42–45, 2015.

[6] W. F. J. Alfonso, F. E. González, and L. O. Torres, "Diseño y análisis del chasis de un vehículo monoplaza Formula SENA," Rev. Integr. Investig. Apl. Desarro. Tecnológico e Innovación, vol. 5, no. 1, pp. 58–86, May 2017.

[7] E. Becchio, A. Chiara, R. Fileccia, and M. Mastrocola, "Use of FEM in the drawing die structure design," SAE Technical Paper, 1998.

[8] S. Lee, "Optimum design and selection of heat sinks," in Proceedings of 1995 IEEE/CPMT 11th Semiconductor Thermal Measurement and Management Symposium (SEMI-THERM), 1995, pp. 48–54.

[9] J. R. Culham and Y. S. Muzychka, "Optimization of plate fin heat sinks using entropy generation minimization," IEEE Trans. Components Packag. Technol., vol. 24, no. 2, pp. 159–165, Jun. 2001.

[10] [10] R. P. Selvam, S. Govindaswamy, and H. Bosch, "Aeroelastic analysis of bridges using FEM and moving grids," Wind Struct., vol. 5, no. 2_3_4, pp. 257–266, Apr. 2002.

[11] P. Wei, M. Y. Wang, and X. Xing, "A study on X-FEM in continuum structural optimization using a level set model," Comput. Des., vol. 42, no. 8, pp. 708–719 , 2010.

[12] W. Tang, L. Tong, and Y. Gu, "Improved genetic algorithm for design optimization of truss structures with sizing, shape and topology variables," Int. J. Numer. Methods Eng., vol. 62, no. 13, pp. 1737 –1762, Apr. 2005.

[13] K. Deb and S. Gulati, "Design of truss-structures for minimum weight using genetic algorithms," Finite Elem. Anal. Des., vol. 37, no. 5, pp. 447–465 , May 2001. [14] H. Kawamura, H. Ohmori, and N. Kito, "Truss topology optimization by a modified genetic algorithm," Struct. Multidiscip. Optim., vol. 23, no. 6, pp. 467–473 , 2002.

[15] M. Attik, L. Bougrain, and F. Alexandre, "Neural network topology optimization," in International Conference on Artificial Neural Networks, 2005, pp. 53–58.

[16] I. Sosnovik and I. Oseledets, "Neural networks for topology optimization," Russ. J. Numer. Anal. Math. Model., vol. 34, no. 4, pp. 215–223 , 2019. [17] J. RONG, J. JIANG, D. YAN, and B. XU, "Bridge Structure Topology Optimization with Multiple Constraints [J]," Eng. Mech., vol. 4, 2002.

[18] M. Cavazzuti, A. Baldini, E. Bertocchi, D. Costi, E. Torricelli, and P. Moruzzi, "High performance automotive chassis design: a topology optimization based approach," Struct. Multidiscip. Optim., vol. 44, no. 1, pp. 45–56 , Jul. 2011.

[19] M. Cavazzuti, D. Costi, A. Baldini, and P. Moruzzi, "Automotive chassis topology optimization: a comparison between Spider and Coupe designs," in Proceedings of the world Congress on Engineering, 2011, vol. 3 , pp. 6 –8.

[20] M. Cavazzuti, L. Splendi, L. D'Agostino, E. Torricelli, D. Costi, and A. Baldini, "Structural Optimization of Automotive Chassis: Theory, Setup, Design," Struct. Multidiscip. Optim., pp. 1–3 , 2011.

[21] L. Scappaticci, N. Bartolini, E. Guglielmino, and G. Risitano, "Structural optimization of a motorcycle chassis by pattern search algorithm," Eng. Optim., vol.49, no. 8, pp. 1373–1387 , Aug. 2017.

[22] Y. Chang-zheng and D. Zhao-xiang, "Optimization design of frame vibration response base on the integrated motorcycle [J]," Mod. Manuf. Eng., vol. 10, p.26 , 2008.

[23] S. Q. Y. C. M. Hengyong, "Topology Optimization Design of the Frame of Container Semi-dragging Trucks [J]," Trans. Chinese Soc. Agric. Mach., vol. 1, 2005.

[24] M. N. Sudin, M. M. Tahir, F. R. Ramli, and S. A. Shamsuddin, "Topology optimization in automotive brake pedal redesign," Int. J. Eng. Technol., vol. 6, no. 1, pp. 398–402, 2014.

[25] D. Xiao, H. Zhang, X. Liu, T. He, and Y. Shan, "Novel steel wheel design based on multi-objective topology optimization," J. Mech. Sci. Technol., vol. 28, no. 3, pp. 1007–1016, 2014.

[26] H. Joshi, A. Powar, S. Khuley, and D. P. Yesane, "Analysis and Topological Optimization of Motorcycle Front Wheel," Elastic, vol. 201, p. t43, 2016.

[27] C. P. V. R. Kumar and R. S. Meher, "Topology Optimization of Aluminium Alloy Wheel," Int. J. Mod. Eng. Res., vol. 3, pp. 1548–1553, 2013. [28]H. Zhang, M. Y. Wang, F. Chen, Y. Wang, A. S. Kumar, and J. Y. H. Fuh, "Design and development of a soft gripper with topology optimization," in 2017 IEEE/RSJ International Conference on Intelligent Robots and Systems (IROS), 2017, pp. 6239–6244. [29] B. J. Kim, D. K. Yun, S. H. Lee, and G.-W. Jang, "Topology optimization of industrial robots for systemlevel stiffness maximization by using part-level metamodels," Struct. Multidiscip. Optim., vol. 54, no. 4, pp. 1061–1071 , 2016.

[30] S. Junk, B. Klerch, L. Nasdala, and U. Hochberg, "Topology optimization for additive manufacturing

using a component of a humanoid robot," Procedia CIRP, vol. 70, pp. 102 –107, 2018.

[31] oomlout, "Arduino Controlled Servo Robot (SERB)," Instructables, 2007. [Online]. Available: https://www.instructables.com/id/How-to-Make-anArduino-Controlled-Servo-Robot-SER/.