International Journal of Engineering Sciences 2022 15(3) 120 - 128

INTERNATIONAL JOURNAL
OF
ENGINEERING SCIENCES

Finite Element Analysis of a Novel Robotic Gripper:*Grabo*

Aniket H Bhelsaikar[1*], Pratik Chothe[1], Viinod Atpadkar[1], Shantipal Ohol[2], Debanik Roy[3$]
[1]SVR InfoTech, Pune, India
[2]Associate Professor, Mechanical Engineering Department, College of Engineering, Pune
[3]Division of Remote Handling & Robotics, Bhabha Atomic Research Centre & Homi Bhabha National Institute, Department of Atomic Energy, Mumbai, India

(Received 28 September 2021; Accepted 2 February 2022)

DOI: https://doi.org/10.36224/ijes.150305

Abstract
Robotic grippers, with several design variations in its jaws, are gaining popularity in commercial market worldwide in recent past. In fact, these customized robotic grippers are widely used for diverse end-applications in various arenas. We have aimed to optimize the mechanical design of such customized robotic grippers having two fingers (jaws), as those grippers are the most prevalent variants in industrial robots. In this study, a niche contigutive robotic gripper for direct adhesion contactis developed, to meet the technical trials of force-closure of grasp. The prime focus of our research is reduction of the tare-weight of the prototype gripper keeping its strength unaltered under the threshold of grasping maximum payload. Through Finite Element Analysis and optimization thereof, material retention percentage is defined for the prototype gripper so as to improve its overall envelope.

Keywords: *Topology, Optimization, Finite Element Analysis, Design, Robotic Gripper, Articulated Jaw*

1. Introduction

Design modulation and customization of robotic grippers are gaining popularity in global perspective due to its effectiveness in a variety of end-applications. However, in many of the new designs prototyping poses considerable challenge because of the external envelope of the gripper and its self-weight. The objective of the present research is to reduce the tare-weight of a novel curvilinear-jaw robotic gripper keeping the strength unaffected by imbibing topology optimization niche. In fact, any new engineering component / system will be rather bulky until the optimization is performed on that entity. More weight means additional cost of manufacturing for no motive.

Although strong and light-weight design of a robotic sub-system in static condition can be satisfactorily obtained by optimization practices, designing an optimal robotic component under real-time dynamic condition requires a different approach. Topology optimization method permits users to attain ensemble specifications of robotic systems where supports and loads are positioned a-priori. Under such a situation, designer can pin-point the best shape / contour of the very robotic component. This absolute design freedom makes topology optimization a powerful design tool in many areas, including robotic systems.

Enhancing features of parts through topology optimization, deprived of applying manufacturing constraints, results in organic design. This is because the material will self-define the through-path from

---

[*] Corresponding author
Email address: aniket@svrinfotech.net (Aniket H Bhelsaikar)
ISSN 0976 – 6693.

the load to the restraint, ensuring the most effective shape mathematically. The present study describes the syntax of the customized topology optimization routine as well as use of optimization module of commercially available Finite Element Analysis (FEA) software, with specific reference to the prototype design of the robotic jaw gripper. The solver optimizes for maximum stiffness, deformation, Eigen frequencies and many other parameters. This has been accomplished by defining a design space from which the solver can eradicate material until the most optimal shape is realized.

In order to attain the prime objective of weight attenuation, static analysis of the prototype gripper is performed using Finite Element Method (FEM) and based on the simulation result and interpretation thereof, optimization region is summarized. As explained earlier, FEA aids in designing durable, lightweight components for the prototype robotic gripper and optimization has been used to improve the design of the same that is poised to be deployed in various real-time applications. The analysis also allows users to specify where supports and loads are acting in 3D space of the envelope of the gripper and permits the software to find the finest profile. This helps in reducing the cost of manufacturing by designing a real-life lightweight robotic gripper for specified application.

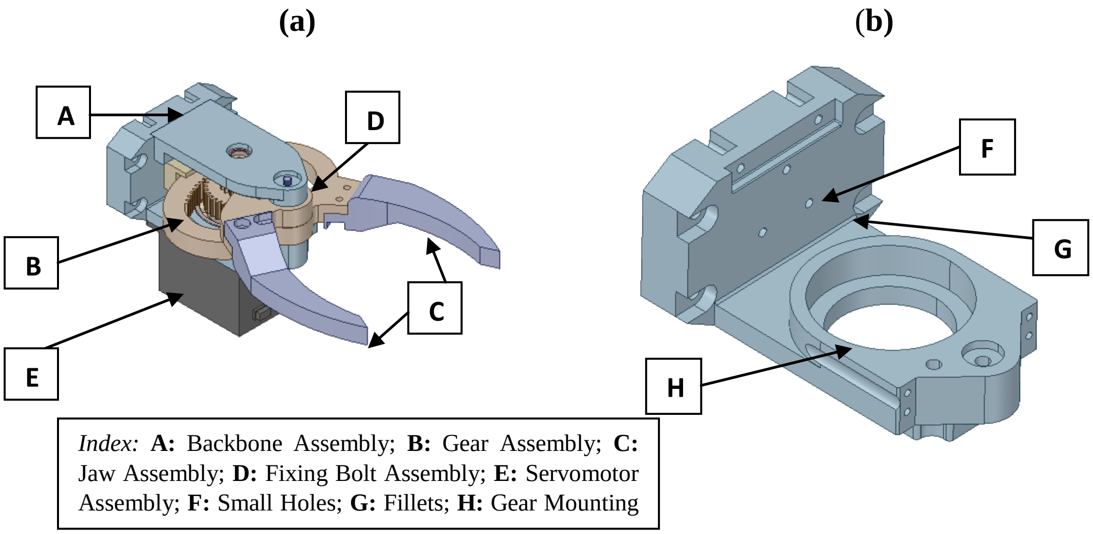
The overall external dimensions of our prototype gripper are 145 mm x 54 mm x 42 mm, which signals the small-scale of the envelope that we are dealing with. The designed gripper has a payload capacity of 1.4 kg (approx.) and being a contigutive type of gripper, it can pick up objects that are cylindrical and spherical in nature more effectively. In fact, the two important aspects of robotic grasp, namely, ‘form closure’ and ‘force closure’ have been implemented in the customized design of the jaws of our gripper. Since we have added complementary design metrics of contiguity, the ‘form closure’ of the final grasp has attained superior realization.

Manufacturing of the prototype gripper was very delicate due to its inherent subtleness of the design, post its optimization. Several iterations underwent to search out the best method of manufacturing using precise & high accuracy machine-tools, ensemble effort of which could make it possible to achieve the final ‘quality’ product. For example, laser techniques were invoked to cut the thin parts and Computer Numerical Control (CNC)-based machine tools were used for machining the rest of the mechanical assembly of the prototype gripper.

The paper is composed of six sections. Section 2 deals with the details of the finite element (FE) modelling of the architype gripper and de-featuring of CAD assembly before proceeding for the final analysis. section 3 deals with the design for manufacturing of *Grabo*. Static analysis is performed in the section 4. FEA based grasp synthesis of grabo is discussed in the fifth section and finally, section 6 concludes the paper.

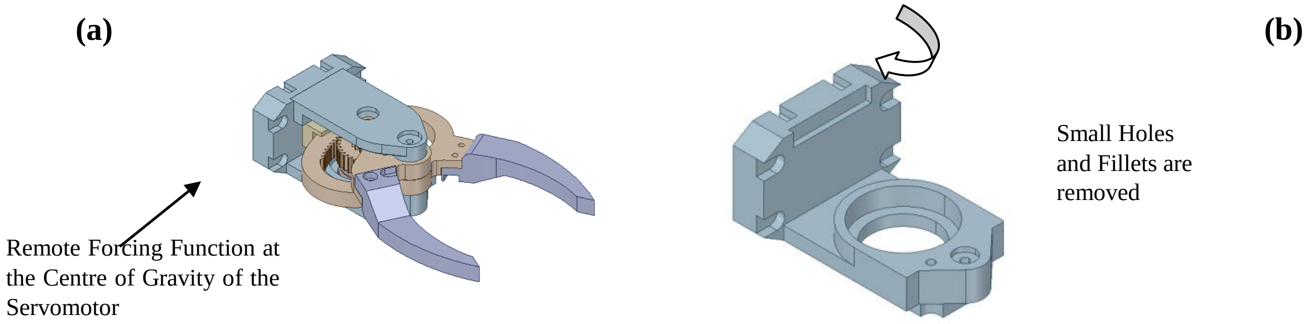
2. FE Modeling of the Prototype Gripper

The finite element (FE) modeling of the prototype robotic gripper and subsequently, analysis is carried out using commercial finite element (FE) software. At the outset, the modelling is done in commercial CAD software. The CAD is bring in (to the FE platform) in *.step* format, which is a universal format and any commercial FE software has an inbuilt option to process the same. We have used FE-Pre-processor for cleaning the unwanted features in the CAD assemblage and executed ‘de-featuring’ function seamlessly. Figures 1(a) & (b) illustrate the original CAD model of the *Grabo*, while the first-level ‘de-featured’ version of the same is presented in figs. 2 (a)& (b). Likewise, small holes & fillets are abandoned from the backbone topology of the *Grabo* at the end of first-level de-featuring (refer fig. 1a). This sort of selective trimming of non-fundamental geometric features is an interesting paradigm of topology optimization.



Index: **A:** Backbone Assembly; **B:** Gear Assembly; **C:** Jaw Assembly; **D:** Fixing Bolt Assembly; **E:** Servomotor Assembly; **F:** Small Holes; **G:** Fillets; **H:** Gear Mounting

Fig. 1: Original CAD Model of the Gripper: (a) Isometric View ; (b) Backbone Topology View



Remote Forcing Function at the Centre of Gravity of the Servomotor

Small Holes and Fillets are removed

Fig. 2: First Level De-Featured Version of the Prototype Gripper: (a) Isometric View); (b) Backbone Topology View

It may be noted here that since insignificant topographies in the CAD like pin-holes, fillets, chamfers, screws and bearings can disturb the mesh quality for the FEA, we need to go for augmentation of the computational time. Thus, the degree of 'de-featuring' will be decisive in taking the final verdict for FEA as the results will be highly exaggerated by the accuracy of the 'de-featuring'.

Since all the parts in the 'de-featured' CAD assemblage of the architype gripper are solids with considerable dimensions, 3D elements are cast-off for the FEA. Hexahedral elements are the best choice to be used in 3D domain for of its accurateness but multifaceted shapes cannot always be meshed using hex-elements. A combination of higher-order tetra and hexa elements is cast-off in the FE-model, as complex geometries cannot be meshed with perfect cubes. Figure 3 (a) shows the ensemble FE model of the architype gripper. Figure 3 (b) shows the portion of the FE- modelled gripper taking tetra elements only, while the part-assembly of the FE- modelled mesh by means of shared tetra and hexahedral elements is illustrated in fig. 3 (c).However, it may be well-known that hex dominant technique has been used in the meshing of the portion of the gripper assemblage of fig. 3 (c).

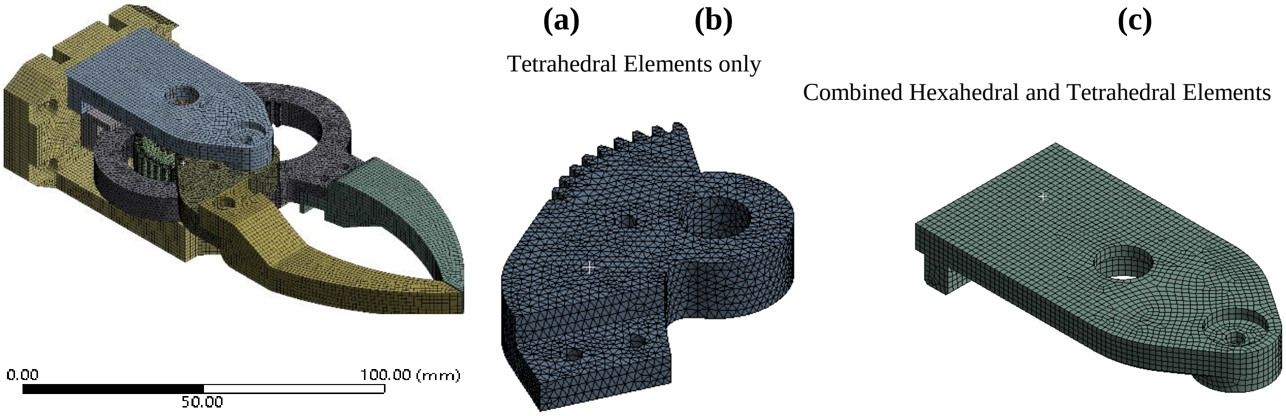


Fig. 3: FE Model of the Prototype Gripper: (a) Ensemble Model; (b) Regions with Hexa Elements; (c) Regions with Collective Hexahedral and Tetrahedral Elements

Higher order hexahedral element is defined by 20 nodes having 3 degrees- of-freedom (DOF) in translatory motion per node, viz.{Tx, Ty, Tz} and higher order tetrahedral element is defined by 10 nodes having 3 DOF in translatory motion per node, viz.{Tx, Ty, Tz}. The hexa elements are favoured in FE-model in those specific regions where the model is not complex. The tetrahedral on the other hand is preferred in sections where the model is complex. As evident from the CAD model of the architype gripper shown in fig. 1, the upper plate and jaw base are not as complex as internal and external gears. Hence, we have judiciously used hexahedral, tetrahedral and combination of these two elements as per the requirement, which has culminated in not only improvised FEA of the entire gripper system but also helped in reducing the number of element count in the model. The element size is chosen based on the minimum length of the geometric feature in the model, i.e. the gripper assembly. Mesh density is one of the very important parameters in obtaining acceptable results, since the algorithm cannot alter the nodal locations. Since the optimization results will be based on the efficiency of removal of elements so finer mesh is expected in the iterations of the analysis. The node and element count of the pre-optimized FE-model are shown in below table 1.

Table 1: Mesh Statistics of the Finite element Models of the Gripper System

	Original Model	Final De-featured Model
Node Numbers	1584931	756749
Total Number of Elements	1035011	124500
Total number of Hexahedral Elements		44900
Total number of Tetrahedral Elements		79600

3. Design for Manufacturing of *GRABO*

The final revised CAD of the prototype gripper (post-de-featuring) is shown in fig. 4, along with the location for fixation of the driving servomotor.

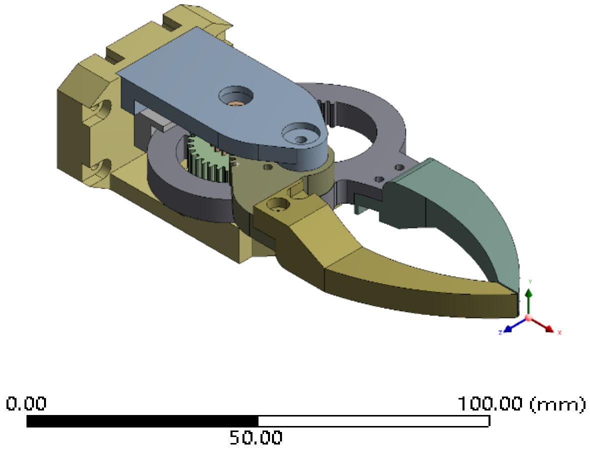


Fig. 4: Post -de-featured CAD of a Gripper

The next step of our FEA is to define the loads and boundary conditions. The load and boundary conditions are applied to the gripper model as shown in figs.5 (a) & (b) and tabulated in table 2.

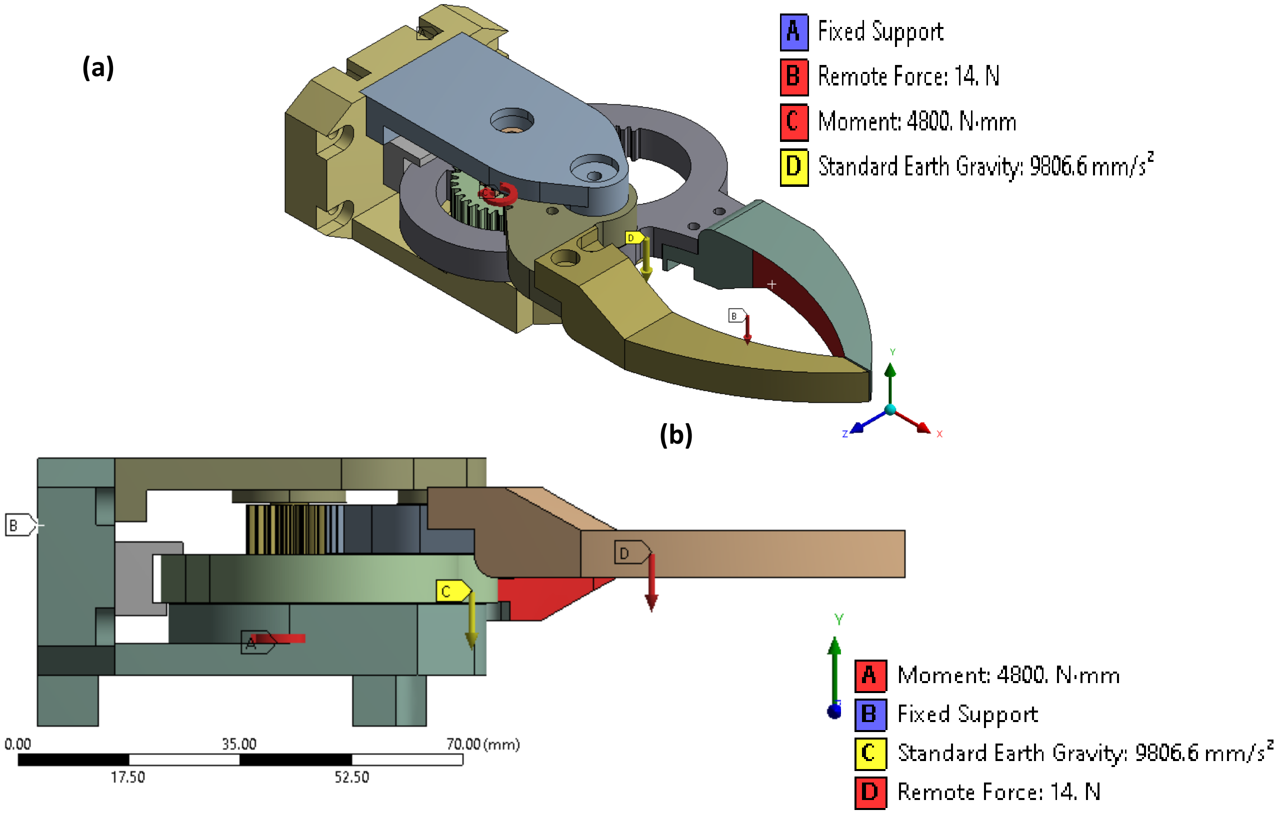


Fig.5: Applied Loads and Boundary Conditions of a Gripper: (a) in Isometric View, (b) in SideView

Table 2: Loading and Boundary Conditions of the FEA of the Gripper System

Loading Conditions	Magnitude
A. Motor Torque	4.8 Nm
B. Fixed Support	As shown in Fig. 7
C. Standard Earth Gravity (Acceleration)	9.8066 m/s^2
D. Maximum Graspable Weight	1.4 kg

Since, the geometric modelling and material properties for the FEA are assigned in 'N' and 'mm', accordingly, we need to convert torque and forces in 'N-mm' and 'N' respectively. We have considered the total maximum payload that needs to be grasped by the prototype gripper is 1.4 kg. Thus, a *remote point* is created in the FE-software at the centre of the jaws. The remote force of 14 N (1.4x9.8066=>14N) is then applied. The servomotor, responsible for the overall actuation of the gripper system, needs to supply continuous torque of 4.8 Nm to the gears. Hence, total torque of 4800 Nmm is applied at the gear location at the fitment of the servomotor.

We have carried out the pre-processing of the FEA (node selection, meshing, loading & boundary condition) of the prototype robotic gripper system in subtle detail, as explained above. Once the pre-processing is done the FE-model is submitted to the FE-solver. Direct solver is used in our application, wherein equivalent stress and the total deformation are evaluated.

In order to keep the tare weight of the gripper minimum, majority of the components are proposed to be manufactured using aluminium. One of the sliding parts in the model is proposed to be made up of brass in order to keep the friction minimum. The gears are proposed to be made up of hardened steel with hardness up to 55 HRC. Various engineering properties of the Materials for Manufacturing of the Gripper are shown in table 3.

Table 3: Properties of the Materials for Manufacturing of the Gripper

Material Properties	Young's Modulus, GPa	Poisson's Ratio	Density, kg/m^3
Aluminum	70	0.33	2800
Steel	200	0.3	7850
Brass	105	0.35	8470

4. Static Analysis of a Gripper

Extensive FE-simulation was taken for static analysis of the gripper using ANSYS®. Figure 6(a) shows the Von-Mises stress contour of the gripper assembly obtained in ANSYS®. From the analysis we have found that the maximum stress of 42.624 MPa occurs near the gear location. The stress values are lower at the tip locations and gradually go on increasing towards the gear locations. Since the gears are hardened and the yield strength of steel is 250MPa, the result obtained thus is lower and hence safe with a factor of safety of about 5. Also, the stress result obtained for aluminium components reaches a peak value of 30 MPa. Naturally, this is also on lower end of the overall stress hue.

Figure 6(b) shows the deformation contour of the prototype gripper, highlighting the maximum value of the total deformation of 0.0026425mm under load (almost negligible for all practical purpose). As shown in the FE-screenshot, the maximum deformation has been obtained at the tip location, where the payload is located. The deformation has also been found with a linear gradient that gradually decreases along the length and finally reaches to zero at the fixed location. The deformation has been found to occur in same direction of applied load.

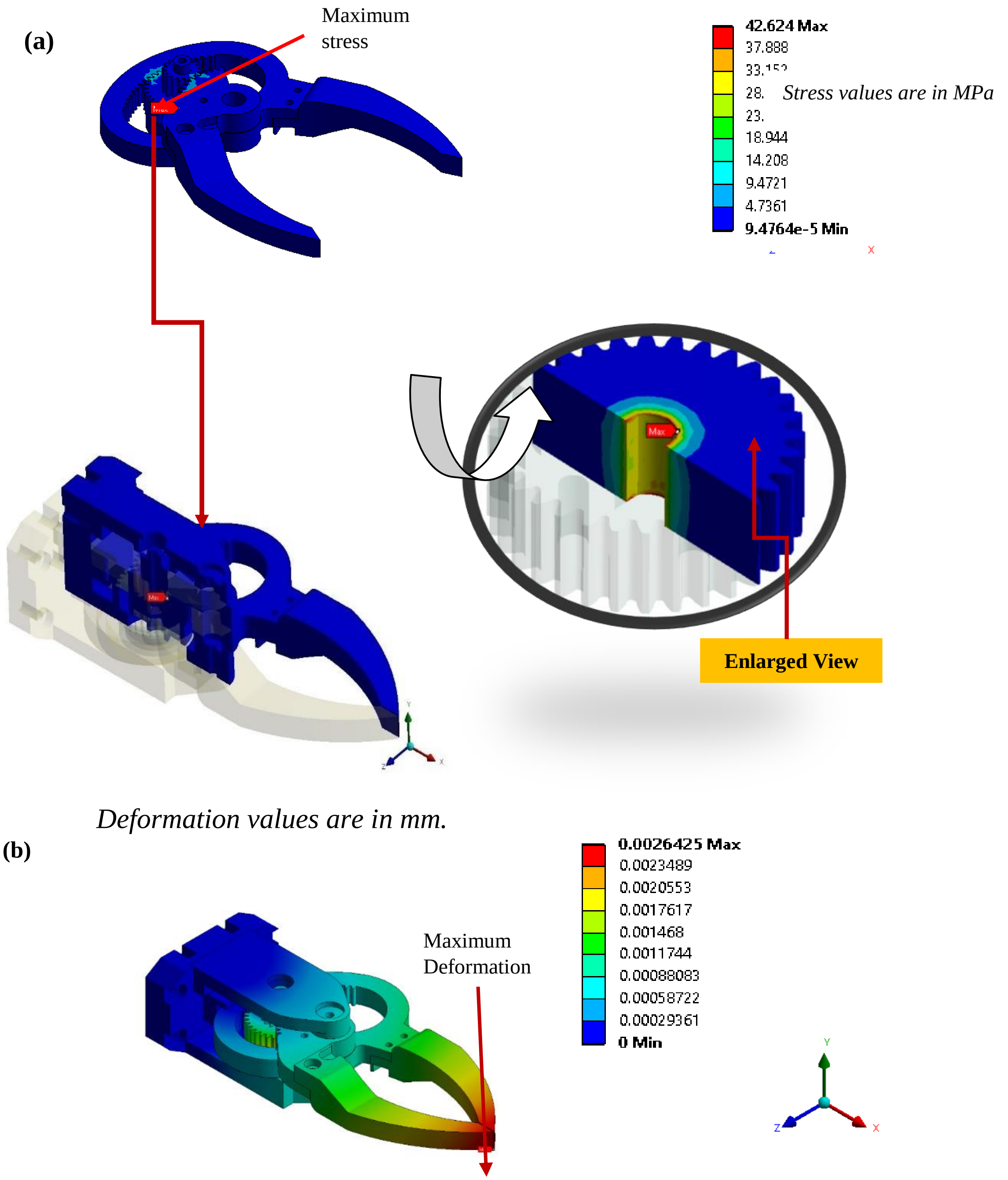
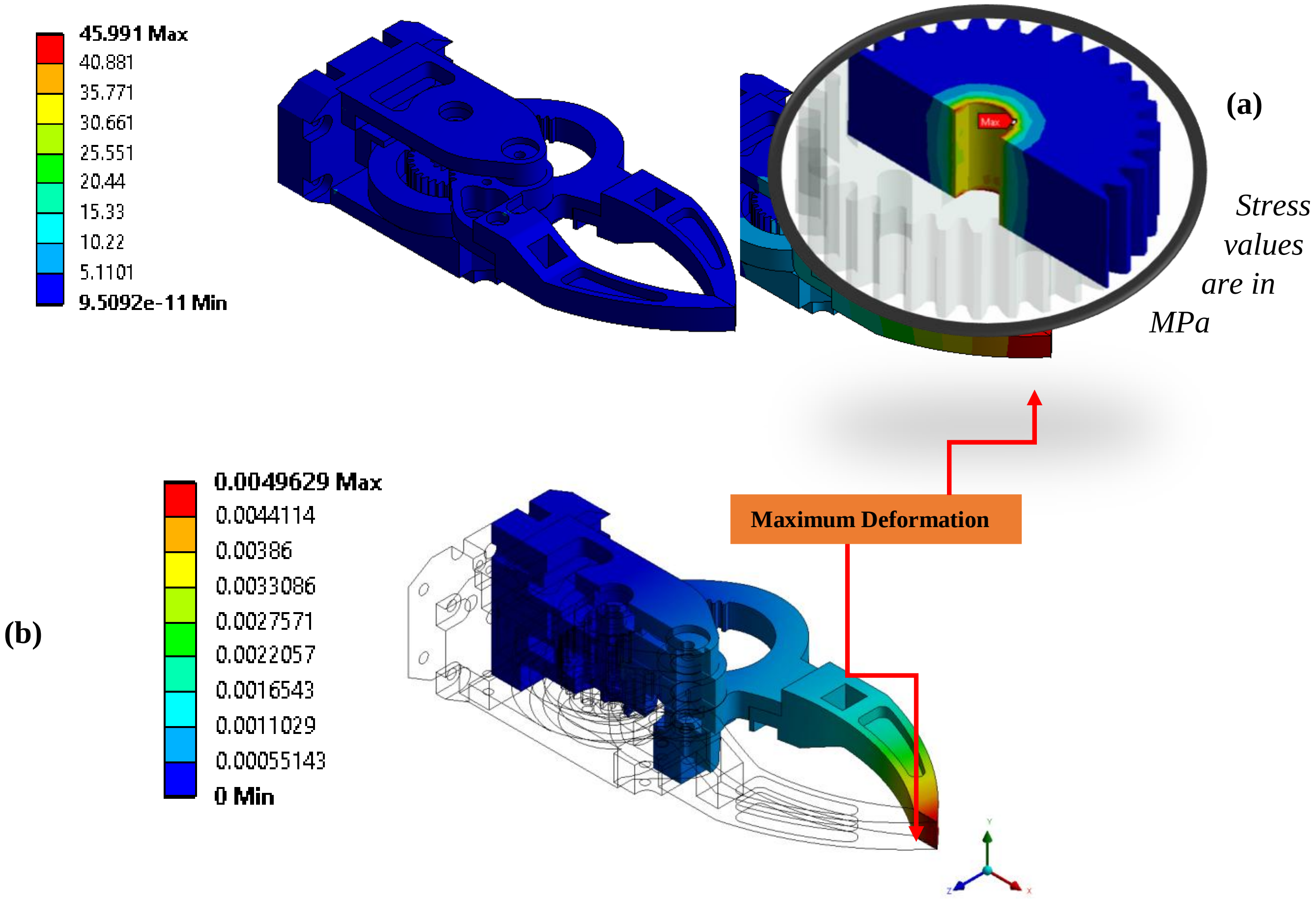


Fig.6: FE-Screenshots of Static Analysis of the Prototype Gripper: (a) Equivalent (Von-Mises) Stress Contour (b) Deformed Shape Contour

It is interesting to observe the FE-screenshots of both equivalent (Von-Mises) stress contour as well as deformation contour of the gripper under the simulated condition of strength & form closure of the grasp. By the virtue of the design, the curvilinear-jaw gripper maintains a perfect 'form closure' at the end of the grasp, maintaining contiguity.

5. FEA Based Grasp Synthesis of Optimized Grabo

The mass of the *Grabo* is reduced by performing the topology optimization on the above gripper. The material removal is done in such a way that the manufacturing can be done easily. Figure 7 (a) illustrates the FE-screenshot of the equivalent (Von-Mises) stress of the gripper during grasp. As per FEA, we have got a maximum stress of 45.991 MPa during grasp that occurs near the gear location. The deformation contour of the gripper under grasp synthesis is shown in fig. 7 (b). Although negligible numerically, the maximum deformation will occur at the jaw tip during force closure, as illustrated in the FE-screenshot. The total deformation of 0.0049629 mm occurs at the tip location and reduces gradually to zero at the fixed location.



Deformation values are in mm.

Fig. 7: FE-Screenshots of Grasp Synthesis of the Prototype Gripper: (a) Von-Mises Stress Contour; (b) Maximum Deformation Contour

6. Conclusions

Besides saving of material and fabrication cost, product aesthetics was improved by this novel methodology of topology optimization. Our module of topology optimization has resulted successfully in attaining the maximum rigidity to the prototype gripper system, along with mass control criteria. The topology optimization has resulted in minimizing the compliance, along with reduction of volume & tare weight. The customized module of iterative topology optimization has resulted in design for manufacturing as well as physical realization of real-life ensemble hardware of a novel lightweight curvilinear-jaw robotic gripper. The successful prototyping of this curvilinear-jaw robotic gripper has been attributed to various real-life contigutive slip-free grasp of objects, both in stand-alone mode as well as integrated with robotic manipulator [for details, please refer to the following video-clips: (i) https://www.youtube.com/watch?v=048N9Xqsseg; (ii) https://youtu.be/29gz3_Im6lg; (iii) https://www.youtube.com/watch?v=29gz3_Im6lg;

References

[1] Kishan Anand and AnadiMisra, “Topology Optimization and Structural Analysis of Continuous Linear Elastic Structures Using Optimality Criterion Approach in ANSYS”, International Journal of Innovative Research in Science, Engineering and Technology, vol. 4, issue 6, June 2015, ISSN (Online): 2319-8753, pp. 2347-6710.

[2] Adrian Ghiorghe, “Optimization Design for the Structure of a RRR type Industrial Robot”, U.P.B (UniversitateaPolitehnicaBucuresti). Science Bulletin, Series D, vol. 72, issue. 4, 2010, ISSN 1454-2358, pp. 121-128.

[3] Anmol Pandey, Rituparna Datta and Bishakh Bhattacharya, “Topology Optimization of Compliant Structures and Mechanisms Using Constructive Solid Geometry for 2-D and 3-D Applications”, Springer-Verlag, Berlin Heidelberg, 2015.

[4] Sudhanshu Tamta and Rakesh Saxena, “Topological Optimization of Continuum Structures Using Optimality Criterion in ANSYS”, International Research Journal of Engineering and Technology (IRJET), e-ISSN: 2395 - 0056, vol. 3, issue 7, July-2016, pp 1483-1488.

[5] Gunwant, D. and Misra, A,“Topology Optimization of Sheet Metal Brackets Using ANSYS”, MIT International Journal of Mechanical Engineering, vol. 2, no. 2, Aug. 2012. pp 119-125.

[6] C. D. Chapman, “Structural Topology Optimization via the Genetic Algorithm”, Thesis, M. S. Massachusetts Institute of Technology, U.S.A., 1994.

[7] KishanAnand and Anadi Misra, “Topology Optimization of Beam Supported at Ends and Elastic Plate with Central Elliptical Hole Using Optimality Criteria Approach in ANSYS”, International Journal for Research In Emerging Science And Technology, vol.2, issue 5, May 2015, pp 241-247.

[8] Caner Demirdogen, Jim Ridge, Paul Pollock and Scott Anderson, “Weight Optimized Design of a Front Suspension Component for Commercial Heavy Trucks”, Proceedings of the SAE Commercial Vehicle Engineering Congress and Exhibition Chicago, Illinois, U.S.A., October 26-28, 2004, pp 400-410.