Estimation of Dynamic Characteristics of a Novel Non-Parallel Detachable-Jaw Robotic Gripper Using Finite Element Method

To cite this article: Aniket H Bhelsaikar et al 2021 J. Phys.: Conf. Ser. 1831 012010

View the article online for updates and enhancements.
Estimation of Dynamic Characteristics of a Novel Non-Parallel Detachable-Jaw Robotic Gripper Using Finite Element Method

1Aniket H Bhelsaikar, 2Debanik Roy, 3Viinod Atpadkar
1,2,3SVR InfoTechPure, India
2Division of Remote Handling & Robotics, Bhabha Atomic Research Centre and Homi Bhabha National Institute,
Dept. of Atomic Energy, Mumbai, India

Abstract. Robotic grippers are increasingly aiming towards versatility to suit industrial applications of modern times. Detachable-type jaws of the robotic grippers do play a significant role in diverse end-uses in real-time. A novel contigutive robotic gripper was used in the present study for direct adhesion contact to meet the technical challenges of force-closure of the grasp. The crux of the design was realized through an iterative optimization so as to ensure mismatch of natural frequency of the gripper and the forcing frequency in order to avoid resonance condition. Natural frequencies and mode shapes of the prototype gripper system were studied in order to understand its dynamic behaviour. Two commercial Finite Element Analysis Software were used for the effective characterization of the real-time dynamics of the gripper system. The dynamic analysis and simulation were instrumental in manufacturing of the test-pieces and final prototype of our indigenous robotic gripper.

1. Introduction
Modern-age manufacturing as well as material handling demands for adaptive robotic systems, equipped with jaw-type grippers that are more adaptable in use. In fact, robotic grippers are increasingly aiming towards versatility and customization to dovetail a huge gamut of industrial applications in recent past. Detachable jaws of the robotic grippers add design niche to this endeavour in an effective manner and such grippers, often customized, become instrumental in real-time industrial applications. On the other hand, attaining non-parallel motion of the gripper-jaws add design challenges, which gets appended whenever we intend for non-parallel jaws too. Curvilinear-shaped jaws are perfect example of such non-parallel jaws having non-parallel motion. These grippers are, by and large, contigutive and maintain a perfect ‘form closure’ during robotic grasp in real-time. With this research backdrop, we have designed and fabricated a novel contigutive-type detachable-jaw robotic gripper for direct adhesion contact with the object in order to meet the technical challenges of ‘force closure’ of the said grasp. An iterative topology optimization process was used in the ensemble design process to avoid condition of resonance to occur in the self-vibration phase of the gripper. In other words, the final design was invoked in such a way so that the natural frequency of vibration of the gripper should not have matched with the external forcing frequency (generated due to payload and/or induced impulse in the gripper-body). At resonance condition, we observe a sudden amplification in the amplitude of vibration that may cause damage to the structure, i.e. gripper-body. Hence, the overall objective of the present study is to check the natural frequency of vibration of the gripper system and to determine whether to perform dynamic analysis or static analysis to finalize the design for manufacturing.
The free vibration analysis was used in the present work for dynamic characterization of prototype gripper, which was expressed by its Eigen value and Eigenvector (function in case of continuum). The Eigen value analysis helped in uncoupling of the equations of motion for the gripper-jaws, aiding faster response analysis. Although use of proportional damping concept for decoupling damping matrix is mathematically elegant, it hardly predicts the observed values to acceptable accuracy. In order to avoid such uncertainty, we have ignored damping for the present free vibration analysis to overcome mathematical complexity.

Finite Element Method (FEM) has been used quite extensively for dynamic analysis of semi-compliant ‘structures. As an extension of such analyses, off-line performance of jointed systems such as robotic gripper was also evaluated using commercially available Finite Element Analysis (FEA) software. In the present study, we have used two such commercial FEA packages to reveal the optimum throughput of the real-time dynamics of the prototype gripper system. The results of these dynamic / modal analyses, including off-line animation have been found effective in manufacturing of the test-pieces of our indigenous gripper units as well as the final prototype. Several researchers have used FEA for rheological studies of robotic end-effectors. Qassab & Sultan [1] analyzed the FEA results on stress and deflection for a two-fingered robotic hand and compared those with the results obtained from theoretical model-based calculations as well as experimentation. An interesting FEM-based modal analysis outcome of robotic arm reported that circular-shaped cross-section of the arm can sustain higher vibration than square-shaped robot arm structure, irrespective of the type of cross-section (solid or hollow) [2]. The principle of FEA was extended to fine-tune the design of a three-link slender flexible robotic manipulator, especially for the links and revolute joints [3]. Details of vibration signature & associated deflection of the links of the indigenously-designed flexible manipulator have been studied by the authors. FEM-based simulation thereof helped in pin-pointing the support locations & related kinetics between the base-assembly and the first link of the said flexible manipulator. Various attributes of parametric design and dynamic simulation of the indigenously-developed controller of a Multi-Gripper Assistive Robot was reported along with results of test-runs [4]. The ensemble programming logic for the robot was developed towards controlling in-built vibration of the robot in real-time. The hardware of the robotic manipulator was accomplished in a way so as to minimize the inherent shaking of the manipulator arms. Roy [5] discussed on the differences in vibration characteristics of multi-link flexible robot from that of a single-link system which is primarily due to the coupling effects of joints & flexible shafts. The proposed methodology builds up an optimal foundation for analyzing inherent vibration of flexible robots using strain gauge-based measurement as well as stochastic model-based fusion of sensory data. A methodology for reducing tare-weight and minimizing structural deformation in 3D space of a 6-axis articulated robot (ARISTO®) was addressed, based on the calculation of the loading forces applied in a static study [6]. An interesting treatise on the study of the natural frequency of a robotic arm in free vibration case was reported, wherein maiden analysis was made using the classical analytical method of well-known Euler-Bernoulli Theory [7]. The free vibration results therein showed the effectiveness of the proposed artificial neural network scheme in terms of computational time. The kinematic and dynamic performance of a reconfigurable robotic gripper system was analyzed theoretically, followed by validation of the ensemble design using Integrated Design Engineering Analysis Simulation software (I-DEAS®) [8]. The maximum load capacity of the picking mechanism and compliance matrix of the releasing and alignment mechanism of a flexure-based novel robotic gripper was determined analytically by considering the buckling effect of picking mechanism [9]. Equations for compliance of alignment and releasing mechanisms were formulated as a function of flexure dimensions and material properties, which were validated using a flexible multi-body model.

The paper is composed in six sections. An overview on the developed finite element model of the prototype gripper is presented in the next section. Section III describes the details of dynamic /modal analysis of the prototype gripper. Results of the FEA-based simulation are reported in section IV. Section V highlights on the manufacturing paradigms of the ‘test’ gripper(s) and subsequently, the prototype gripper. Finally, section VI concludes the paper.
2. Finite Element Model of The Prototype Gripper

We have created the 3D geometric model of the prototype detachable-jaw robotic gripper prior to finite element modeling in order to get an insight over its functional characteristics. This step is very vital and equally non-trivial so far as the background of FEA is concerned. The crux of our finite element model, geometrical as well as computational, is ‘de-featuring’ of the original Computer Aided Design (CAD) model of the prototype gripper, which was created using solid modeling software. The de-featured CAD model was imported to the FEA platform for basic analysis. However, it may be noted here that since minor features in the CAD like pin-holes, fillets, chamfers, screws and bearings can affect the mesh quality for the FEA, we need to take a call on the enhancement of the computational time.

![Post-de-featured CAD Model of the Prototype Gripper](image1.png)

**Figure. 1:** Post-de-featured CAD Model of the Prototype Gripper

Thus, the extent of ‘de-featuring’ will be crucial in taking the final judgement for FEA as the results will be highly affected by the accuracy of the ‘de-featuring’. Figure 1 shows the post-de-featured CAD model of the prototype gripper, with major functional constituent parts indexed. Since all the parts in the ‘de-featured’ CAD assemblage of the prototype gripper are solids with considerable dimensions as shown in Fig. 1, 3D elements are used for the FEA. As per the 3D CAD of the gripper, hexahedral elements are the best choice for FEA because of its accuracy, though complex shapes cannot always be meshed using hex-elements. The hexahedral elements are preferred in FE-model in those specific regions where the model is not complex and vice-versa for the tetrahedral elements. Thus, a combination of tetrahedral and hexahedral elements was used in the Finite Element (FE)-model of the gripper. Figure 2 shows the ensemble finite element model of the prototype gripper. However, it may be noted that hex-dominant method has been used in the meshing of the relevant portions of the gripper assembly.

![Finite Element Model of the Prototype Gripper](image2.png)

**Figure. 2:** Finite Element Model of the Prototype Gripper
The meshing phase of the FEA was carried out with a focus towards minimization of the degrees-of-freedom(s) at critical locations of the gripper-body. Accordingly, finer mesh was created at crucial areas of the gripper. In fact, we have used multi-point constraint criteria in the ensemble FEM-based modeling process, spanned in several iterations. The improvement of the mesh data over these FEA iterations is presented in Table 1.

### Table 1: Mesh Data of the FE-Models of the Prototype Gripper

| Parameter       | Original Model | Final Model |
|-----------------|----------------|-------------|
| Number of Nodes | 1584931        | 756749      |
| Number of Elements | 1035011 | 124500 [Hex.: 44900; Tetra: 79600] |

The part of the FE-modeled gripper having tetrahedral elements only is shown in Fig. 3(a), while Fig. 3(b) presents the part-assembly of the FE-modeled mesh using combined tetra and hexahedral elements. It may be noted that hex dominant method has been used in the meshing of the portion of the gripper assembly of Fig. 3(b).

![Figure 3](image)

**Figure 3:** FE-Mesh of the Prototype Gripper: (a) Zones with Hexahedral Elements; (b) Zones with Combined Hexahedral and Tetrahedral Elements

We have used celebrated Euler-Bernoulli beam theory for the FE-based dynamic analysis of the prototype gripper. We have assumed that the end-displacement of the gripper-jaws can be separated into two parts, respectively dependent on position and time as shown below:

\[ w(x,t) = \Lambda(x)\Psi(t) \]

(1)

where, \( w(x,t) \): overall displacement of the gripper-jaws; \( \Lambda(x) \): component of the displacement as a function of position and \( \Psi(t) \): component of the displacement as a function of time. The governing equation that was used for estimating these two components of displacement under FEA to ensure simple harmonic motion is presented below:

\[
EI \frac{\partial^4 \Lambda(x)}{\partial x^4} = \frac{1}{\Psi(t)} \frac{\partial^2 \Psi(t)}{\partial t^2} = -\omega^2
\]

(2)

where, \( E \): Young's modulus of the material; \( I \): Moment of inertia of the 'beam' (gripper-jaw / body); \( A \): Cross-sectional area of the 'beam'; \( k \): Linear mass density of the 'beam' (= \( k = \rho A \)); \( \omega \): Frequency of vibration of the 'beam'. Now, segregation of the position variable of eqn. 2 will lead to the following FE-equation:

\[
\Lambda(x) = C_1 \sinh(\bar{\alpha}x) + C_2 \cosh(\bar{\alpha}x) + C_3 \sin(\bar{\alpha}x) + C_4 \cos(\bar{\alpha}x)
\]

(3a)
\[
\delta = \left( \frac{\rho A \omega^2}{EI} \right)^{1/3} \quad (3b)
\]
Likewise, we can get the following extraction of eqn. 2 for time variable, as used in FEA:

\[
\frac{\partial^2 \psi(t)}{\partial t^2} + \omega^2 \psi(t) = 0 \quad (4)
\]

The FEA solution for eqn. 4 will be:

\[
\psi(t) = C_5 \sin(\omega t) + C_6 \cos(\omega t) \quad (5)
\]

where, \(C_5\) and \(C_6\) are constants.

Finally, the combined FE-equation for the displacement of the gripper-jaw /body becomes:

\[
\begin{align*}
\delta(x, t) &= (C_1 \sinh \delta x + C_2 \cosh \delta x + C_3 \sin \delta x + C_4 \cos \delta x) X (C_5 \sinh \omega t + C_6 \cosh \omega t) \\
& \quad \text{where, the constants \{C_1, C_2, C_3, C_4\} can be obtained from the boundary conditions and \{C_5, C_6\} are obtainable from the initial conditions under the FEA.}
\end{align*}
\]

Figure 4 illustrates the final FE-model of the prototype gripper (side view), with demarcation for its fixation with the wrist of a robotic arm. The ‘Cantilever Beam’ type modeling was accomplished here using FEAST® software.

**Figure. 4:** Cantilever Beam-based Final FE-Model of the Prototype Gripper using FEAST® software

3. **Paradigms of Modal Analysis of the Gripper**

In a typical FE-based modal analysis, single or multiple natural frequencies of vibration of a dynamic system can be evaluated based on the requirement of the end-application. The numerical values of such modal frequencies can be plotted suitably and interpreted for various physical parameters pertaining to the system. With this backdrop, the modal analysis of the prototype robotic gripper system under study was carried out with the underlying objective of minimization of the volume and mass of the gripper system. To begin with, we have selected the ‘rigid body mode’ of analysis that is demarcated as the free translation or rotation of a body deprived of enduring any substantial internal deformation. As part of this analysis, there will be six rigid body modes, namely, three translational (TX, TY, TZ) and three rotational (RX, RY, RZ). For free-free normal modes analysis, we have assumed that there are no loads or restraints over the gripper body. In other words, the gripper body will not undergo any internal deformation but will be able to move or rotate without restrictions. This free-free run also ensures that there are no appreciable modeling errors in the assembly. If any component in an ensemble (muster) is not relatively associated with the other or if any contact between the two mating members is not defined

\[1\text{ Developed indigenously by Vikram Sarabhai Space Centre (VSSC), ISRO, India. The code is primarily aimed at structural analysis.}\]
or some zonal meshes are not compatible, we will get some non-zero frequencies within first six frequency results in the analysis. After the free-free run, the boundary conditions are added to perform the fixed-free analysis on the structure, i.e. the gripper-body. Number of modes that are to be extracted depends upon the percentage ratio of effective modal mass and total mass of the gripper system. We have used two commercially available FE-platforms to carry out the modal analysis, namely, ANSYS® and FEAST® software. While FEA using ANSYS® does have the leverage of detailed CAD as the backbone, the FE-modeling in FEAST® calls for slight deviation, so far as the external geometry of our prototype gripper is concerned.

stated earlier, the gripper system was modeled as a cantilever beam, having two different cross-sections. We have used these two unequal cross-sections, respectively for the gripper-body and gripper-jaws, in order to facilitate 1D simulation and to run the one-dimensional problem. Figure 5 illustrates this dual cross-section analytical framework of the cantilever beam model of our gripper, made in FEAST®.

We may note that it is not possible to determine the exact cross-section of the cantilever beam, as the modeling of fig. 5 is merely an approximation that is felt sufficient for the FEA under FEAST®. With reference to parts ‘A’ & ‘{B1, B2} of fig. 1, we have arrived at the thickness of the backbone assembly and jaw assembly of the gripper as 38 mm. and 21 mm respectively. As illustrated in fig. 5, the ensemble 1D model is composed of two contiguous planar beams, the larger one symbolizes ‘Gripper-Body’ (65 mm x 38 mm.) while the smaller beam represents ‘Gripper-Jaws’ (80 mm x 21 mm.). We have used these two different cross-sections judiciously, in order to run the 1D problem effectively in FEAST®. It may be noted that as per the above layout as well as the overall external dimension of the gripper, the gripper-jaws are modeled using a rectangular cross-section of 54 mm x 21 mm and the gripper-body is modeled by another rectangular cross-section of 42 mm x 38 mm for the FEA using FEAST®.

4. Simulation & Results of the Modal Analysis

The preliminary FEA results and simulation thereof have revealed that the natural frequency of the prototype gripper system is very high, approximately above 1000Hz. This ensured enough rigidity of the gripper body so far as its actuation (grasping) in real-time was concerned. Thus, there was no need to go for further dynamic analysis as no resonance would occur. Hence, only static simulation and its optimization were useful for our application. Under this perspective, we carried out FE-simulation for the first two mode shapes only (at two different working planes) that are illustrated here. Figures 6(a) and 7(a) show the mode shape results using ANSYS® code and figs. 6(b) and 7(b) show the mode shapes plotted in FEAST® software. It may be noted here that although both FEA-software do perform the requisite modal analysis with sufficient accuracy, the modeling approach is different. We have incorporated the results of both ANSYS® as well as FEAST® in order to get better insight to the vibration characteristics of the prototype robotic gripper, besides comparison of the numerical values.
Mode #1: 1302.4 Hz [ANSYS®]

Mode #1: 1487.34 Hz [FEAST®]

Figure 6: FE-screenshots of the First Mode Shape of the Prototype Gripper: (a) ANSYS®; (b) FEAST® Software

Index: A: Initial Posture; B: Deflected Posture

Mode #2: 2084.6 Hz [ANSYS®]

Mode #2: 1855.83 Hz [FEAST®]

Figure 7: FE-screenshots of the Second Mode Shape of the Prototype Gripper: (a) ANSYS®; (b) FEAST® Software
As per the modeling in FEAST® software (refer fig. 5) and subsequent simulation, mode shapes can be simply pictured as deflections of a continuum cantilever beam, using 1-D results. This is, no doubt, a unique characterization and quite effective in our case. The cantilever-beam approximation is invoked because the gripper model is fixed at the end and free at the other, so a generalized free body diagram can be represented by a line-body and its mode shape can be idealized by the mode shapes of this line-body. This representation of a line-body along with its mode shapes can easily be compared with the 3D model to check whether the mode shapes are correct as per the fundamentals of vibration. The ‘beam’ is essentially demonstrated using 3D beam elements of arbitrary length, material properties and cross-section, since only the mode shapes are of interest and not the frequencies. The frequencies will never match with 1D result as the geometric shape cannot be realized in 1D.

Figures 6(a) & (b) and figs. 7(a) & (b) show the analogous mode behavior of 3D model of the prototype gripper and its 1D simplification, respectively using ANSYS® and FEAST®. The colour contour shows the variation of mode shapes results in form of deflection hue, while the horizontal span of the FEA remains same, i.e. 145 mm. (total length of the prototype gripper). The red colour shows the maximum magnitude of mode which is the modal displacement. Figure 6(a) shows the deflection of gripper in the X-Y plane (first mode shape), as per FEA result from ANSYS® (maximum deflection at jaw-tip: 154.99 mm). At resonant condition this deflection may increase in the amplitude and can damage the components in the surrounding area. Figure 7(a) shows the mode shape results in X-Z plane (second mode shape), as per analysis in ANSYS®. Similar screenshots for these two mode shapes under FEAST® are shown in figs. 6(b) & 7(b). Thus, these modes can affect the adjacent components in contact. Since the frequencies at these two mode shapes are high enough, there is no need to work out on the modal frequency further as those will never occur. As the paradigms of FE-modeling are not identical in these two FE-platforms, we have got different values of the modal frequencies for the prototype gripper. The crux of the FE-simulation for the modal analysis is linked with defining the contact geometries between various mating parts of the prototype robotic gripper. The success of the modal analysis depends on the very aspect of contact kinematics. As modal analysis of the prototype robotic gripper is primarily aimed at its real-time dynamics, it is crucial to model the gripper system with details of the contact kinematics. ANSYS® provides good options to define contact between the constituent and contiguous parts of the prototype gripper under FE-modeling. However, contact body and target body needs to be defined adequately for establishing contact kinematics in ANSYS®. This paradigm is at times, becomes tricky, especially for small-sized envelopes like our gripper system. We have overcome this difficulty in modeling by using some advanced features of ANSYS® towards defining the contact, which is geometrically equivalent to ‘bond contact’. Nonetheless, we could not avoid a small amount of ‘gap’ in the FE-model despite proper definition of the contact. This inherent shortcoming of FE-modeling (mesh generation phase) gets alleviated in the analysis phase. The FEA of the prototype robotic gripper using ANSYS® provided us with several useful options to detect this ‘gap’ (of FE-mesh in bond contact) such as pinball radius. The FE-program of ANSYS® considers the closure of the bond contact region within the user-defined pinball radius. In other words, the contact and target pairs need to be fully inside this pinball region. In contrary, the modal analysis of the prototype gripper under FEAST® relies largely on the structural syntax of the ensemble. Thus, FEAST is more assertive to rigid body contact kinematics and a bit poor in identification of the ‘gaps’ of the FE-mesh. At times, this restricts the FE-simulation of modal analysis using FEAST® for systems with semi-compliant interconnected members, such as our gripper system. Besides the details of the modal analysis of the robotic gripper system as reported above, the Eigen values and Eigen vectors were studied in depth. Those values were also plotted, jointly using Euler-Bernoulli beam theory and FEAST®. Figure 8 shows the representative plot of Eigen values (“Frequencies” in Hz.) & Eigen vectors (“Mode numbers”).
5. Manufacturing of the prototype Gripper

In order to realize the ensemble novelty of the present research, we phased out the manufacturing of this robotic gripper in three facets, viz, a] ‘test’ gripper; b] un-optimized gripper and finally c] topology-optimized gripper (as per the finalized design attributes). The ‘test’ gripper was fabricated largely using non-metallic materials (except gears) so as to confirm the minimization of the tare-weight and subsequent dynamics in FEA. In contrast, the un-optimized design of the gripper was placed for manufacturing with metallic components in order to have the realization of the size & shape of the final product. State-of-the-art manufacturing methodology was invoked in the fabrication of the un-optimized version of the prototype gripper. Especially, fabrication of the curvilinear-shaped jaws required very high precision of the machine tool. Likewise, both internal as well as external gear modules called for utmost accuracy control of the manufacturing technology. For example, laser-based cutting technique was incorporated to cut the thin parts and CNC/VMC (Vertical Machining Centre) were used for machining the rest of the structure. Nonetheless, we have completed the manufacturing of both the ‘test’ as well as un-optimized gripper as per the original design. Figure 9 shows the photographic view of the ‘test’ gripper that was used for our maiden trials for grasp synthesis. The confidence gathered in successful completion of the first ‘test’ prototype, as shown in fig. 9, was instrumental in selection of material for fabrication of the next set, i.e. the prototype gripper having un-optimized design and made up with metallic materials in totality. Table 2 highlights various engineering properties of these materials that were used for the fabrication of the said all-metallic prototype gripper.

Figure 8: Representative Plot of Eigen Values & Eigen Vectors as Obtained in FEAST®

Figure 9: Test gripper in use for Pick and Place Application
Table 2: Engineering Properties of the Materials Used in the Manufacturing of the Un-optimized Prototype Gripper

| Material | Young’s Modulus, GPa | Poisson’s Ratio | Density, kg/m³ |
|----------|----------------------|----------------|---------------|
| Aluminum | 70                   | 0.33           | 2800          |
| Steel    | 200                  | 0.3            | 7850          |
| Brass    | 105                  | 0.35           | 8470          |

The post-manufacturing photographic view of the curvilinear-jaw prototype robotic gripper (un-optimized design) is presented in fig. 10. The hardware realization of this gripper helped us doing a vast round of grasp analysis that are essential to ascertain the deployability of this novel gripper for material handling in industry.

Figure. 10: Assembled Prototype Gripper post-Manufacturing

6. Conclusion

The results of the FEA-based modal analysis for the prototype robotic gripper revealed interesting technological paradigms with respect to the evaluation of natural frequencies of vibration and mode shapes. Since the natural frequency of vibration of the prototype robotic gripper was observed to be above 1000 Hz in both FEAST® & ANSYS®, we were confident about the rigidity of the prototype robotic gripper. The FEA-results in totality also indicated that there would be no effect of vibration during actual operation of the gripper. Further, the resonant frequencies were found to be less than 50 Hz. Thus, we can infer that static analysis will be sufficient to determine the overall safety of design and operation of similar robotic grippers. Successful manufacturing of this novel curvilinear-jaw robotic gripper resulted in slip-free grasp of objects in real-time, either in stand-alone mode or upon getting interfaced 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_lm6lg; (iii) https://www.youtube.com/watch?v=E5G_veak8tU]. (iv) https://www.facebook.com/SVRINFOTECH.NET/videos/346400913019942

References

[1] Safeen Yassen Qassab and Hussein Ali Sultan, “Modal Analysis of Pneumatic Two Finger Robotic Hand by Finite Element Analysis and Experimental Testing”, International Journal of Engineering & Technology IJET-IJENS, vol. 13 no. 03, June 2013, pp. 1-9.
[2] Jeevan, Amar Nageswara Rao, “Modeling and Analysis of Robot Arm using ANSYS”, International Journal of Scientific Engineering and Technology Research, vol. 4, issue 33, ISSN 2319-8885, August 2015, pp 6692-6697.

[3] Prathamesh Warude, Manoj Patel, Pankaj Pandit, Vikram Patil, Harshal Pawar, Chinmay Nate, Shreyash Gajlekar, Viinod Atpadkar, Debanik Roy “On the Design and Vibration Analysis of a Three-Link Flexible Robot Interfaced with a Mini-Gripper” Recent Trends in Wave Mechanics and Vibrations, Springer, Online ISBN 978-981-15-0287-3, Nov.2019, pp 29-45.

[4] Harshal Pawar, Chinmay Nate, Manoj Patel, Pankaj Pandit, Vikram Patil, Prathamesh Warude, Neel Wankhede, Pratik Chothe, Viinod Atpadkar, Debanik Roy “Design, Dynamic Simulation and Test-run of the Indigenous Controller of a Multi-Gripper Revolute Robot by Minimizing System Trembling”, Recent Trends in Wave Mechanics and Vibrations, Springer, ISBN 978-981-15-0287-3, Nov. 2019, pp. 47-60.

[5] Debanik Roy, “Towards the Control of Inherent Vibration of Flexible Robotic Systems and Associated Dynamics: New Proposition and Model”, International Journal of Robotics Research, Applications and Automation, vol. 1 issue 1, DOI: 10.18689/IJRA-1000103 April 2019, pp 6-17.

[6] P. G. Chitte and S. S. Bansode, “Structural and Vibrational Analysis of Six axis ARISTO robot using ANSYS”, International Journal of Innovative Research in Technology, vol. 3 issue 1, ISSN: 2349-6002, June 2016, pp 291-296.

[7] Israa Rafie Shareef and Iman Ahmed Zayer, “Natural Frequency Estimation of a Free Vibrated Robotic Arm Using Artificial Intelligence”, International Journal of Applied Engineering Research, ISSN 0973-4562 Vol.13, no. 7,2018, pp. 5393-5399.

[8] Ramesh Kolluru and Kimon P. Valavanis, “Design Fundamentals of a Reconfigurable Robotic Gripper System”, IEEE Transactions on Systems, Man and Cybernetics-Part A: Systems and Humans, vol. 30, no. 2, March 2000, pp. 181-187.

[9] Varan Gupta and Rithul Perathara, “Design and Analysis of a Flexure-based Passive Gripper”, Precision Engineering, ISSN: 0141-6359, March 2019, vol. 56, pp. 537-548.

Acknowledgments
I would acknowledge the support of Mr. Pratik Chothe and Mr. Jaipal Solanki from SVCR InfoTech for kind hearted support in designing and modelling the gripper.