A method to determine the topology of custom torsional elastic element for the lightweight rotary series elastic actuator

Min Zhou1,a, Sun’an Wang1,b*

1School of Mechanical Engineering, Xi’an Jiaotong University, Xi’an, Shaanxi, China
*aemail: zhimazhou@163.com
*bCorresponding author; bemail: sawang@xjtu.edu.cn

Abstract. Compact and lightweight rotary series elastic actuator (SEA) is a new scheme to realize safe and friendly human-robot interaction of exoskeletons and other human-robot interaction applications. The custom monolithic torsional elastic element is a good choice for the compact design of rotary SEA, but there is a lack of guidance on selecting the topology of elastic elements in a lightweight SEA design. In this paper, with the help of the concept of specific energy, a method based on finite element simulation environment and parametric 3D CAD environment is proposed to determine the topology of the elastic element suitable for lightweight SEA by maximizing the specific energy. By adopting this method, we come to this conclusion: The topology, Archimedes-spiral-based double spiral structure without the outer ring, can be determined as a suitable topology of elastic element for a lightweight rotary SEA.

1. Introduction

The robots driven by traditional stiff actuators, represented by industrial robots, have made considerable progress in the past decades because of their precise and rapid positioning and large force output. Structured environments such as standard industrial applications require robots to achieve accurate position control. Given that the traditional stiff actuators show good performance in position control, the rule of thumb of "stiffer is better" has become the consensus of robot designers in the past.

However, when robots such as exoskeleton and manipulators are used in the human environment or need to work collaboratively with humans, we found that the rule of thumb of "stiffer is better" began to become inapplicable. Traditional stiff actuators, such as hydraulic cylinders or motors with large reduction ratio transmission, have large mechanical impedance, in other words, poor performance in back-drivability. Therefore, a friendly and safe human-robot interaction is not guaranteed, especially in unexpected situations, like an unexpected impact, which is likely to cause injuries. The way to solve the problem that the human-robot relationship is not friendly enough caused by the stiff actuators can be improved by establishing advanced control algorithms, such as Impedance control, Hybrid force/position control, Adaptive control. The method mentioned above still follows the principle of "rigidity by design, safety by sensors and control", but the paradigm "safety by design, performance by control" should be established in human-robot interaction devices to assure that satisfactory safety levels are reached[1].

The Series Elastic Actuator (SEA) is such an intrinsically compliant and safe actuator that was first completely proposed by Williamson and J. Pratt[2]. The SEA has become popular among compliant actuators. The application fields of SEA include walking robots[3, 4], assistance or rehabilitation robots[5, 6], manipulators[7], exoskeletons[1], powered orthotics [8], and other devices that interact
with humans. SEAs can be divided into linear series elastic actuators[9] and rotary series elastic actuators[10] according to the different forms of motion output, leading to different mechanical structures in SEA design. Rotary SEA is more advantageous in applications such as manipulators, exoskeletons, and powered orthotics. This is because most of the joints of these robots are rotary joints, while linear SEA usually requires additionally designed mechanisms (slider-crank mechanism, for example) to provide rotational motion. However, the current design of rotary SEA still faces some challenges, mainly the requirements of compactness and lightweight. Especially when introducing the SEA to the exoskeleton, a compact and lightweight design must be guaranteed. One-part elastic elements or so-called monolithic torsional elastic elements are more suitable to achieve a lightweight and compact solution[11], which are usually not commercially available and need to be customized by the designer. Unfortunately, the current design of monolithic torsional elastic elements for rotary SEA is very individual. There has not been enough research to guide the topology determination of the torsional elastic element in the lightweight rotary SEA design.

The purpose of this article is to adopt the proposed method based on finite element method(FEM) simulation environment and 3D parametric CAD environment to determine a suitable topology of elastic element for a rotary SEA to meet the design requirements of lightweight. The contribution of this paper includes two aspects: 1. This work provides suggestions on the topology determination of monolithic torsional elastic elements for lightweight rotary SEA. 2. The method for determining the topology of elastic elements proposed in this article can guide the SEA design in specific projects.

The paper is structured as follows: The basis and method for determining the topology of the lightweight elastic element are given in Section 2. The one critical part of this method, namely topologies set establishment and parametric modeling, is shown in Section 3, while another critical part of this method, namely settings of FEM-based simulation environment, is shown in Section 4. The results of topology determination are given in Section 5. Section 6 is dedicated to discussion and conclusions.

2. Method of Determining a suitable topology
To measure the lightweight performance of elastic elements, the specific energy that can be stored under safe stress conditions is used as a quantitative indicator. The specific energy of the elastic elements is defined as[1]:

$$r_{em} = \frac{k(\Delta \theta_{max})^2}{2m}$$  (1)

where $r_{em}$ is the specific energy, $k$ is the stiffness, $\Delta \theta_{max}$ is the maximum allowable deformation, and $m$ is the mass of the elastic element. The larger the specific energy value, it means that we have obtained a lighter design under the same maximum load and maximum deformation. Therefore, the lightweight design requirement is transformed into the design requirement of maximizing the specific energy, which leads us to determine the topology of the elastic element. The material may also be related to the specific energy, but it is not the object of our study.

A method based on FEM-simulation environment (ANSYS Workbench) and 3D parametric CAD environment (SolidWorks) is adopted to determine the topology of the monolithic torsional elastic element, and the specific working process is shown in Fig. 1. The topology with the largest specific energy will be determined as the most suitable topology. The details of this method are listed as follows:

(1) Create a set of topologies of the elastic element, and $N$ is the number of topologies in the the set $T={t_1,t_2,\cdots,t_N}$. All of these $N$ topologies can be characterized by $n$ independent geometric parameters $x_i(i=1,2,\cdots,n)$, and their shapes correspond to the set of the geometric parameters one-to-one. Each parameter is defined in a range, namely $x_i \in [x_i^{\text{MIN}},x_i^{\text{MAX}}]$.

(2) Select a topology $t_k$ to be analyzed, where $t_k \in T(k \in \{1,\cdots,N\})$. And set the range for each parameter, given the number of the design points $M$, the set of the design points $P$ was created, where $P = \{p_1,p_2,\cdots,p_M\}$. Each design point $p_j$ corresponds to a shape of the elastic element with the topology $t_k$, where $p_j = (x_1,x_2,\cdots,x_n)(j = 1,2,\cdots,M)$.
(3) Set value for each geometric parameter based on a particular design point \( p_j \), and perform parametric modeling in the 3D CAD environment according to the parameters.

(4) Import the 3D model into the FEM-based simulation environment to perform the statics simulation and record the simulation results.

(5) Continue searching for other design points of the topology, modeling, simulation, and recording the results until all the design points in the set have been traversed and the results of the statics simulation are obtained.

(6) Perform the optimization procedure based on the simulation results of the design points and the objective of maximizing the specific energy. Then, output the candidate of the topology \( t_k \) according to the optimization procedure that was performed.

(7) Perform the above optimization processes for other topologies and output the candidates that meet the maximum specific energy, respectively.

(8) Finally, evaluate the candidates corresponding to all of these topologies to determine the one for elastic element design.

![Topology determination flow of the elastic element](image)

**Figure 1** Topology determination flow of the elastic element

3. Parametric characterization and modeling

Generally, the basic structure of a monolithic torsional elastic element is composed of an outer ring, an inner ring, and elastic beams connecting the outer ring and the inner ring. By reviewing other's studies, we can find that the researchers pay more attention to the following four topologies of monolithic elastic elements.

(1) Topology 1: Multiple spiral structures with an outer ring. A double spiral spring with an outer ring\[12\] is a particular case.

(2) Topology 2: Multiple spiral structures without the outer ring. A double spiral spring without the outer ring\[5\] is a particular case.

(3) Topology 3: Symmetrical three-elastic-beams structures\[13-16\].

(4) Topology 4: Symmetrical two-elastic-beams structures\[11, 17\].
Topology 1, topology 3, and topology 4 are composed of the outer ring (width 8mm), inner ring, and elastic beams between the inner ring and outer ring. The main difference between them lies in the different styles of the elastic beams.

For topology 1, the elastic beams are some symmetrically distributed elastic branches based on Archimedes spiral. The reason for adopting the symmetrical structure is that there is no radial force on the shaft when the elastic element rotates. There are 6 parameters (shown in Fig. 2(a)) used to characterize topology 1, they are the outer diameter of elastic element outer ring (D_max), the radius of the inner ring (R_1), spiral pitch (DS_D), the width of the elastic beams (DS_W), the thickness of the elastic element (DS_T), and the number of symmetric spiral beams (DS_N). Fig. 2(b) shows the 3D model when DS_N is 3. The parameter range of topology 1 is shown in Table 1, where DS_n can only take integer values (2,3,...,6).

For topology 2, the elastic beams are also some symmetric spiral branches based on Archimedes spiral. The difference from topology 1 is that topology 2 does not have the outer ring. There are 6 parameters (shown in Fig. 3(a)) used to characterize topology 2, and the meaning of parameters is the same as that of topology 1. Fig. 3(b) shows the 3D model when DS_N is 2. The parameter range of topology 2 is shown in Table 1, where DS_n is fixed to 2 (the reason will be discussed in Section 6).

| Parameters | Range of parameters for topology 1 | Range of parameters for topology 2 |
|------------|----------------------------------|----------------------------------|
| D_max      | Fixed 96mm                       | Fixed 96mm                       |
For Topology 3, there are 3 elastic beams and are symmetrically distributed at 120° intervals, and each elastic beam is composed of two basic geometric shapes: arc and line. There are 11 parameters (shown in Fig. 4(a)) used to characterize topology 3, they are the outer diameter of elastic element outer ring (D_max), the radius of the inner ring (R1), radius of fillet on inner ring (DS_R2), radius of fillet on outer ring (DS_R3), the width of the elastic beams (DS_W), the thickness of the elastic element (DS_T), central angle of large arc of elastic beam (DS_A1), central angle of small arc of elastic beam (DS_A2), the span angle occupied by each elastic beam (DS_A3), the length of straight line connecting the outer ring (DS_L1), and the length of the straight line connecting the arcs (DS_L2). Fig. 4(b) shows the 3D model of elastic element based on topology 3. The parameters range of topology 3 is shown in Table 2.

![Figure 4 Parameters and 3D model of elastic element based on topology 3](image)

For Topology 4, there are 2 elastic beams and are symmetrically distributed at 180° intervals. Its geometric shape is similar to that of topology 3. There are 11 parameters (shown in Fig. 5(a)) used to characterize topology 4, and the meaning of parameters is the same as that of topology 3. Fig. 5(b) shows the 3D model of elastic element based on topology 4. The parameters range of topology 4 is shown in Table 2.

![Figure 5 Parameters and 3D model of elastic element based on topology 4](image)
| Parameters | Range of parameters for topology 3 | Range of parameters for topology 4 |
|------------|-----------------------------------|-----------------------------------|
| D_max      | Minimum Fixed 96mm                | Maximum Fixed 96mm                |
|            | R1 Fixed 11mm                     | Fixed 11mm                        |
| DS_R2      | Minimum 2mm                       | Maximum 6mm                       |
| DS_R3      | Minimum 2mm                       | Maximum 6mm                       |
| DS_W       | Minimum 1.5mm                     | Maximum 3.5mm                     |
| DS_T       | Minimum Fixed 4mm                 | Maximum Fixed 4mm                 |
| DS_A1      | Minimum 170°                      | Maximum 190°                      |
| DS_A2      | Minimum 170°                      | Maximum 190°                      |
| DS_A3      | Minimum 90°                       | Maximum 160°                      |
| DS_L1      | Minimum 20mm                      | Maximum 25mm                      |
| DS_L2      | Minimum 12mm                      | Maximum 15mm                      |

4. Settings of FEM-based simulation environment

Response surface optimization is performed in ANSYS Software to obtain the candidate with maximizing specific energy for each topology. The response surface is obtained by DOE (design of experiments), namely by performing statics simulation on the design points and fitting the simulation results. This response surface is the relationship between the design parameters and the design results. In the statics simulation, a fixed constraint is applied to the inner ring of the torsional elastic element, and a constant moment (8 Nm) is applied to the outer ring as the load. Since the stress concentration tends to occur in the narrow areas and the arcs of the elastic beams, advanced size function based on proximity and curvature is performed when meshing to improve the accuracy of the results. Fig. 6 shows the meshing of the elastic element of topology 3 as an example. The material used in simulation is spring steel 65Mn.
Here is the expression of stiffness of the torsional elastic element:

\[
    k = \frac{\tau_{\text{max}}}{\Delta\theta_{\text{max}}}
\]

where \( k \) is the stiffness, \( \tau_{\text{max}} \) is the maximum torque applied (fixed 8 Nm), \( \Delta\theta_{\text{max}} \) is the maximum allowable deformation. So Equation (1) can be rewritten as:

\[
    r_{on} = \frac{\tau_{\text{max}}\Delta\theta_{\text{max}}}{2m}
\]

Therefore, the objective of maximizing specific energy in optimization can be expressed by the follows: minimize mass, maximize deflection, and seek the target of maximum equivalent stress equal to 653 Mpa (the safety factor is 1.2).

5. Results
The results obtained by the response surface optimization are shown in Table 3, which gives the candidate of the maximum specific energy of each topology. The optimization results of topology 1 indicate that the optimal number of spirals (DS_N) is 2, and the relationship between DS_N to mass and maximum deflection is shown in Fig. 7(a) and Fig. 7(b), respectively.

| Results | Topology 1 | Topology 2 | Topology 3 | Topology 4 |
|---------|------------|------------|------------|------------|
| DS_D    | 18.52mm    | 29.67mm    | N/A        | N/A        |
| DS_W    | 3.64mm     | 3.91mm     | 2.18mm     | 2.97mm     |
| DS_N    | 2          | 2          | N/A        | N/A        |
| DS_R2   | N/A        | N/A        | 3.48mm     | 3.72mm     |
| DS_R3   | N/A        | N/A        | 2.17mm     | 2.48mm     |
DS_A1  N/A  N/A  177.50°  184.17°
DS_A2  N/A  N/A  176.89°  180.89°
DS_A3  N/A  N/A  91.64°  145.37°
DS_L1  N/A  N/A  20.46mm  20.46mm
DS_L2  N/A  N/A  14.56mm  12.68mm

Maximum equivalent stress  652.0 Mpa  653.7 Mpa  656.5 Mpa  647.4 Mpa
Maximum deflection (0.249 rad) (0.180 rad) (0.051 rad) (0.053 rad)
Mass  0.134 kg  0.060 kg  0.114 kg  0.121 kg
Maximum specific energy  7.43 J/kg  12.00 J/kg  1.79 J/kg  1.75 J/kg

Figure 7  Simulation results corresponding to DS_N & DS_W of topology 1

6. Discussion and Conclusions
It can be found from Table 3 that the optimization results of topology 3 and topology 4 are very close, indicating that it is not significant whether the number of elastic beams is 2 or 3 in the case described in this article. The double spiral topology is a very potential topology of elastic element for the lightweight SEA. It can be seen from Fig. 7 that the increase of the number of spiral beams leads to the increase of mass and the decrease of deflection angle. Therefore, in the optimization of topology 2, DS_N is fixed to 2 to shorten the simulation time. Removing the outer ring will significantly decrease the mass and so increase the specific energy. The specific energy of the double spiral topology without the outer ring is the largest among all the topologies analyzed under the same conditions.

This article proposed a method to determine a suitable topology of the monolithic torsional elastic element, which references the custom design of the lightweight rotary SEA. The method proposed in this paper is a simulation method based on the finite element analysis environment and the parametric 3D CAD environment. The work of this paper shows that the topology, Archimedes-spiral-based double spiral structure without the outer ring, has higher specific energy than the other three topologies under the same conditions; in other words, it has tremendous potential for designing an elastic element for a lightweight rotary SEA.

References
[1] Carpino, G., Accoto, D., Sergi, F., et al. (2012) A novel compact torsional spring for series elastic actuators for assistive wearable robots. Journal of Mechanical Design, Transactions of the
ASME, 134: 1-10.

[2] Pratt, G.A., Williamson, M.M. (1995) Series elastic actuators. In: Proceedings of the 1995 IEEE/RSJ International Conference on Intelligent Robots and Systems. Part 3 (of 3). Pittsburgh. pp. 399-406

[3] Lee, C., Oh, S. (2019) Development, Analysis, and Control of Series Elastic Actuator-Driven Robot Leg. Frontiers in Neuro Robotics, 13.

[4] Knox, B.T., Schmiedeler, J.P. (2009) A unidirectional series-elastic actuator design using a spiral torsion spring. Journal of Mechanical Design, Transactions of the ASME, 131: 125011-125015.

[5] Lagoda, C., Schouten, A.C., Stienen, A.H.A., et al. (2010) Design of an electric series elastic actuated joint for robotic gait rehabilitation training. In: 2010 3rd IEEE RAS and EMBS International Conference on Biomedical Robotics and Biomechatronics, BioRob 2010. pp. 21-26.

[6] Lee, C., Kim, J.Y., Kim, S.Y., et al. (2018) Human force observation and assistance for lower limb rehabilitation using wire-driven series elastic actuator. Mechatronics, 55: 13-26.

[7] Song, Z.P., Zhou, L.L., Li, Y.B., et al. (2019) Design and Control of a Series Elastic Actuator with High Compliance for Serial Manipulators. In: 9th IEEE International Conference on Cyber Technology in Automation, Control and Intelligent Systems, CYBER 2019. Suzhou. pp. 872-876.

[8] Accoto, D., Carpino, G., Sergi, F., et al. (2013) Design and Characterization of a Novel High-Power Series Elastic Actuator for a Lower Limb Robotic Orthosis. International Journal of Advanced Robotic Systems, 10.

[9] Chauhan, R.J., Ben-Tzvi, P. (2019) A series elastic actuator design and control in a linkage based hand exoskeleton. In: ASME 2019 Dynamic Systems and Control Conference, DSCC 2019. Park City.

[10] Zhang, T., Xia, J. (2019) Development of a Clutchable Series Elastic Actuator for Robotic Hip Exoskeleton. In: Proceedings of the IEEE 2019 9th International Conference on Cybernetics and Intelligent Systems and Robotics, Automation and Mechatronics, CIS and RAM 2019. Bangkok. pp. 269-274.

[11] Irmscher, C., Woschke, E., May, E., et al. (2018) Design, optimisation and testing of a compact, inexpensive elastic element for series elastic actuators. Medical Engineering and Physics, 52: 84-89.

[12] Stienen, A.H.A., Hekman, E.E.G., Ter Braak, H., et al. (2008) Design of a rotational hydro-elastic actuator for an active upper-extremity rehabilitation exoskeleton. In: 2nd Biennial IEEE/RAS-EMBS International Conference on Biomedical Robotics and Biomechatronics, BioRob 2008. Scottsdale. pp. 881-888.

[13] dos Santos, W.M., Caurin, G.A.P., Siqueira, A.A.G. (2017) Design and control of an active knee orthosis driven by a rotary Series Elastic Actuator. Control Engineering Practice, 58: 307-318.

[14] Leal-Junior, A.G., Theodosiou, A., Min, R., et al. (2019) Quasi-Distributed Torque and Displacement Sensing on a Series Elastic Actuator's Spring Using FBG Arrays Inscribed in CYTOP Fibers. IEEE Sensors Journal, 19: 4054-4061.

[15] Leal-Junior, A.G., Frizera, A., Marques, C., et al. (2018) Polymer Optical Fiber for Angle and Torque Measurements of a Series Elastic Actuator's Spring. Journal of Lightwave Technology, 36: 1698-1705.

[16] Sanchez, M.R.A., Leal-Junior, A.G., Segatto, M.V., et al. (2018) Fiber Bragg grating-based sensor for torque and angle measurement in a series elastic actuator’s spring. Applied Optics, 57: 7883-7890.

[17] Huo, X.J., Xia, Y.F., Liu, Y.W., et al. (2014) Humanoid arm with the integrated serial elastic actuator. In: 2014 IEEE International Conference on Robotics and Biomimetics, IEEE ROBIO 2014. Bali. pp.2321-2326.