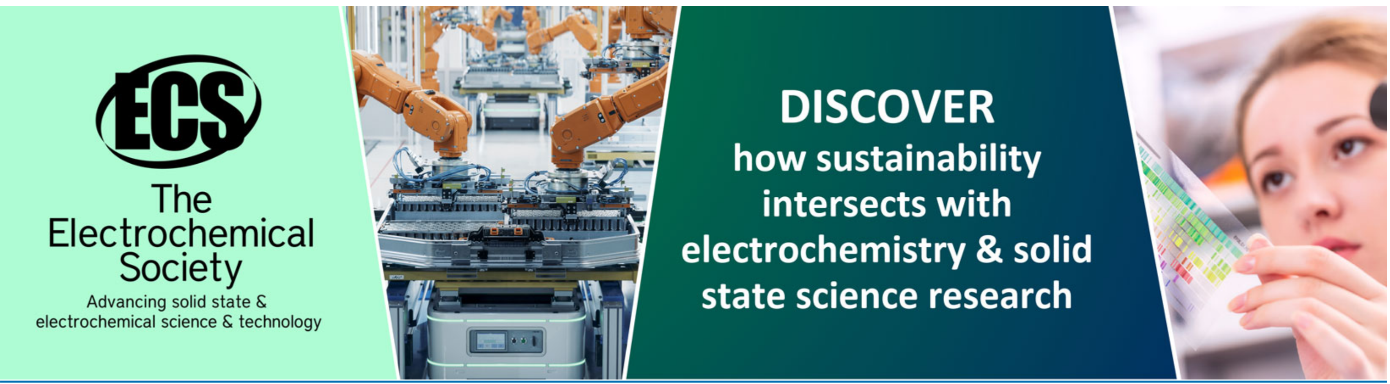
IOP Conference Series: Materials
Science and Engineering

Optimization Design of Support Arm of an Upper Limb Rehabilitation Robot



IOP Conf. Series: Materials Science and Engineering **688** (2019) 033048 doi:10.1088/1757-899X/688/3/033048

Optimization Design of Support Arm of an Upper Limb Rehabilitation Robot

Hairui LIU , Yu Wang*

School of Electromechanic Engineering, Qingdao University, Qingdao, Shandong Province, 266071, China

*Corresponding author's e-mail: ywang@qdu.edu.cn

Abstract. According to the structural design requirements of upper limb rehabilitation robot, the static analysis and harmonic response analysis are made in this paper. The structure of the upper limb rehabilitation robot is simplified in different ways according to the importance of each part of the structure. Selecting different element types and establishing finite element model Based on its analysis results, some unnecessary parts are removed and simplified, and the main forced parts are optimized by defining parameters to optimize and re-design the structure. The static analysis and harmonious response analysis of the modified model, comparing the structure before and after optimization, show that the modified structural mechanical properties are obviously improved and the design requirements are fully met.

1. Introduction

At present, the upper limb isokinetic rehabilitation robots used in the market are all single-degree-of-freedom motion. The upper limb movement trajectory can only be limited to the plane, there is no way to form arbitrary space trajectory, so that the rehabilitation effect is limited[1-3], and when the upper limb is extended at a large Angle, the shoulder joint center will rise, resulting in the misalignment of the man-machine joint axis[2-5], thus bringing additional binding force to the trainer. To solve these problems, this study realized a two-degree-of-freedom iso-velocity motion [6], and realized the dynamic alignment of human-machine joint axis in real time as figure 1.

Due to the increase of degree of freedom and the requirement of configuration [7-8], the high rigidity and low inertia of the motion arm are required [9-11]. In this paper, the structure is designed by topology optimization, and then based on the dynamic force pattern at work make harmonic analysis. Compared with the prototype before analysis, the mass and inertia of the moving part are reduced effectively, and the rigidity and stability of the structure are improved.

2. The frame and analysis

In this paper, the dynamic components and movement modes of upper limb rehabilitation robots are introduced, and the possible deformation caused by them in the working process is analyzed, as well as the conditions to be satisfied for the normal working of robots.

2.1. Description of the structure of upper limb rehabilitation robot

The upper limb rehabilitation robots shown in figure 1 is composed of six parts: 1 -- Vertical axis mechanism, 2 -- Horizontal arm plate, 3 -- Seat, 4 –Arm bar, 5 -- Man-machine joint alignment mechanism, 6 -- Horizontal axis mechanism.

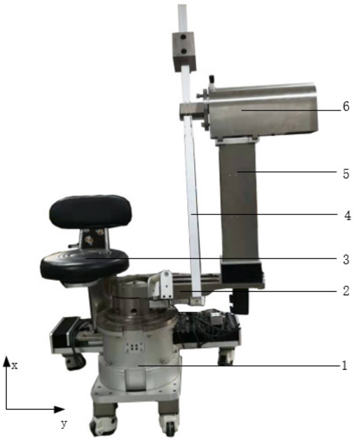


Figure 1. Schematic diagram of rehabilitation robot structure

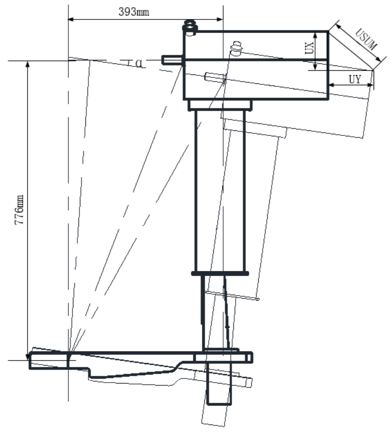


Figure 2. Schematic diagram of upper limb rehabilitation robot deformation

2.2. Mechanical-structural design requirements

As the mass of the arm is much smaller than that of other moving parts, it is ignored in order to simplify the calculation process. The horizontal arm plate is in the shape of a cantilever beam. Firstly, the deflection under static condition should be limited, as figure 2. The design requires an alpha of no more than 1°, the solution formula is as follows.

$$\alpha = \arctan\frac{UX}{393 - UY} \tag{1}$$

3. Build finite element model of upper limb rehabilitation robot

In this section, a finite element model is established for the numerical analysis of the rehabilitation robot, the actual structure is geometrically cleaned and simplified, the corresponding element type [12-14] is selected, and different finite element assembly technologies are selected according to the connection state of each part

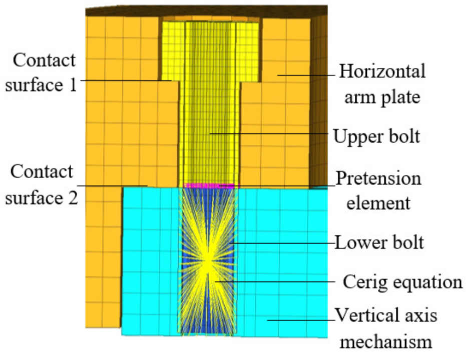


Figure 3. The connection schematic diagram

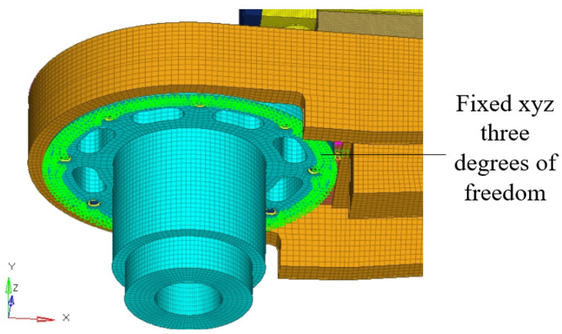


Figure 4. Schematic diagram of constraint application

3.1. Model simplification and the type selection of elements

In order to reduce the generation of deformed elements, some unstressed holes and narrow edges are cleaned out. Secondly, some coils, shields and other parts on the rehabilitation robot have little influence on the stress of the robot can be ignored. For some outsourcing standard parts such as reducer, sensor and other parts, can be regarded as a whole, reduce unnecessary assembly relations, reduce the amount of calculation. For bolt connections that do not need to be considered, the bolt is simplified and the BEAM188 element is used to replace the screw and the CERIG equations used to replace the rigid area between the nut and the connecting plate as figure 3. Contact surface 1 is a bonded contact, contact

IOP Conf. Series: Materials Science and Engineering **688** (2019) 033048 doi:10.1088/1757-899X/688/3/033048

surface 2 is ordinary contact, pretension element is applied in the middle of the bolt, and the lower end of the bolt is connected with the vertical shaft mechanism through the CERIG equations.

3.2. The establishment of constraints and loads

As the whole rehabilitation robot is connected to the vertical axis mechanism of the robot through the horizontal arm plate, it is necessary to focus on the analysis of the horizontal arm plate. Therefore, only the structure connected with the horizontal arm plate can be restrained. The degrees of freedom of the three translational directions are restricted, as shown in figure 4.

When calculating static analysis, only the deformation and stress of the robot under the action of gravity are calculated. F2 lags behind F1 by 90 degrees. At the initial stage, F1 reaches its maximum value. As the arm gradually reaches the horizontal position, its attenuation decreases to zero, and F2 gradually reaches its maximum value from zero. The loading diagram is shown in figure 5.

$$F1 = 40\cos 8\pi t \tag{2}$$

$$F2 = 40\cos 8\pi (t + 0.25) \tag{3}$$

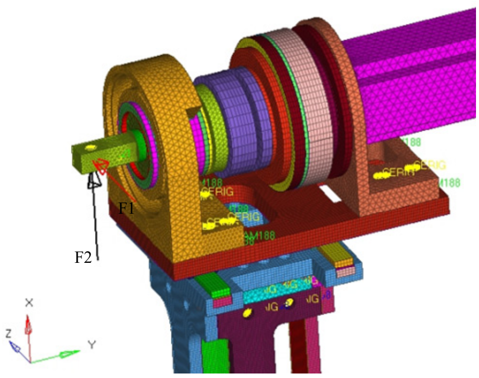


Figure 5. The loading diagram

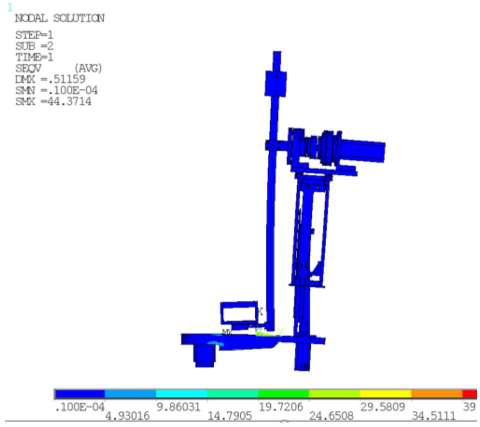
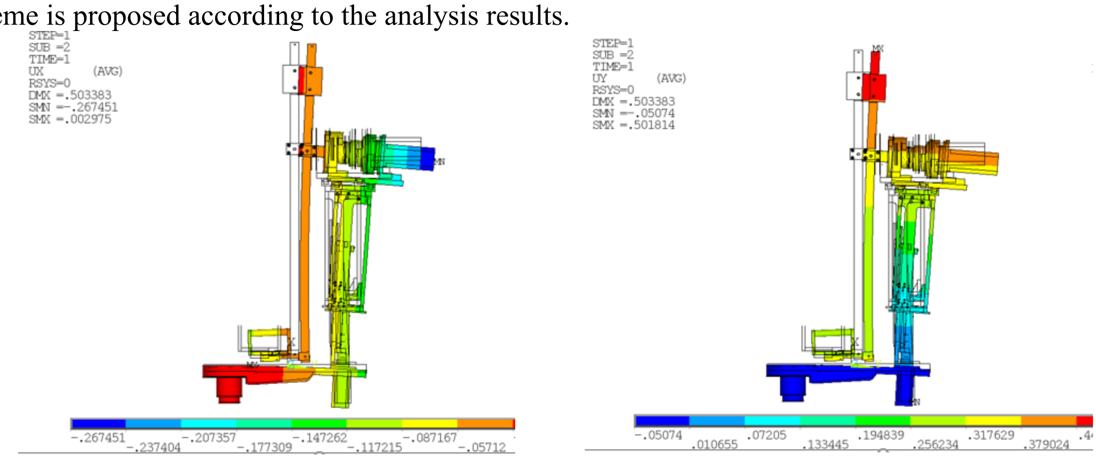


Figure 6. Stress nephogram

4. Finite element analysis of upper limb rehabilitation robot

In this section, the static and dynamic analysis of the robot is carried out to observe whether the mechanical properties of the rehabilitation robot can meet the requirements, and then the optimization scheme is proposed according to the analysis results.



(a) (b)

Figure 7. Deformation nephogram

4.1. Statics analysis of upper limb rehabilitation robot

The finite element model was imported into ANSYS, and the global gravity acceleration was applied to the whole rehabilitation robot for static analysis. The analysis results of the rehabilitation robot were observed in the post-processor to find the maximum deformation and stress. The maximum stress value of 44.37MPa is much less than the allowable deformation and stress， as figure 6. The displacement in the X direction is 0.27mm, and the displacement in the Y direction is 0.46mm, as figure 7. According to equation (1), the α is 0.04 degrees, which can meet the design requirements of the structure by no more than 1 degree.

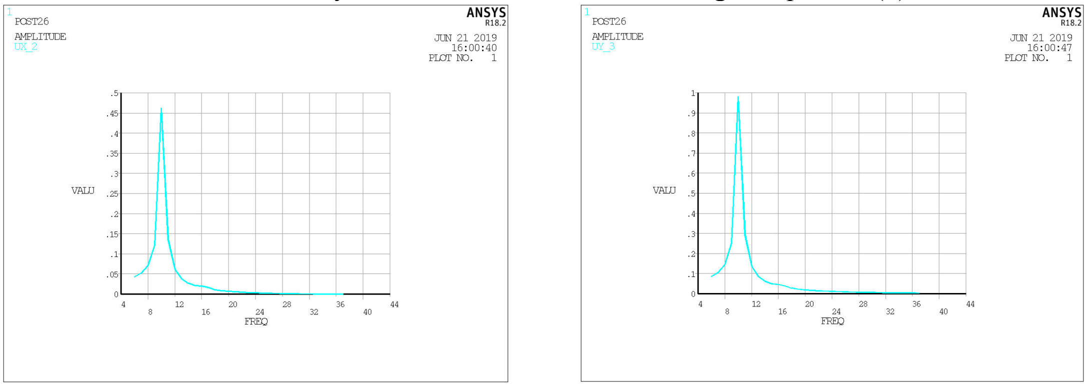
4.2. Harmonic response analysis of upper limb rehabilitation robot

The first six modes and modes of rehabilitation robot are calculated by modal analysis. The order and frequency are shown in table 1.

Table 1. The first six frequencies of the rehabilitation robot

Order	1	2	3	4	5	6
Frequency/(Hz)	8.0	8.5	10.4	13.8	15.1	39.6

Because the working frequency of servo motor is much higher than the first 6 frequencies of rehabilitation robot, it can't be considered. According to the actual working conditions, the working frequency of the patient is 4Hz, which is relatively small and generally does not reach the resonance frequency. The harmonic response of rehabilitation robot was analyzed, and view the post-processing results in ANSYS as shown in figure 8. The maximum deformation in the x direction is 0.46mm, and the maximum deformation in the y direction is 0.98mm. According to equation (1), the α is 0.06 °.



(a) (b)

Figure 8. The relationship between displacement and frequency

5. Structural optimization of rehabilitation robot

5.1. optimization of the curricular structure

To reduce the weight of the robot, it mainly modifies the human joint alignment mechanism and the horizontal arm plate. In order to reduce the assembly of each connecting plate, the original four connecting plates are changed to two connecting plates. And the original screw to choose a smaller model of the screw. As the vertical axis mechanism of the robot moves, the width of the horizontal arm plate is too large and may touch the legs of the patient, so the horizontal arm plate is narrowed and supported by a single beam and thickened. The modified model of the horizontal arm plate is shown in figure 9.

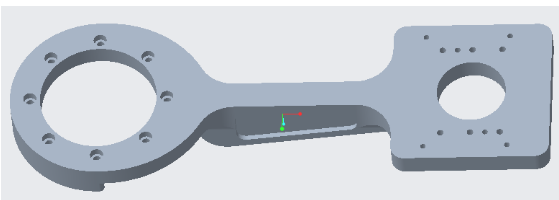


Figure 9. Modified horizontal arm plate

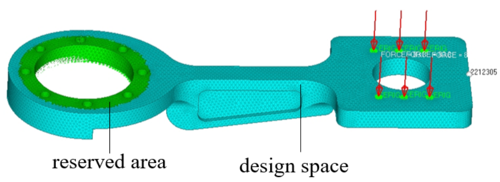


Figure 10. Define design variables

5.2. Topology optimization of horizontal arm plate

5.2.1. Topology optimization design

The modified horizontal arm plate was topologically optimized. In OptiStruct topology optimization technology, density method is adopted for the material pattern, that is, the optimal design variable is the density of each element in the design space, and the element density values continuously from 0 to 1 according to the importance. When the element density is close to 0, it means that the material is not important and can be removed. The reserved area is the place where constraints are applied, and the design space is the place to be optimized as figure 10.

5.2.2. Optimization analysis results

The contour map of density results is shown in figure 11. The optimization threshold is set as 0.25, and the contour map of the optimal layout is shown in figure 12. The structural design of the horizontal arm plate is conducted according to the optimized iso-surface diagram, as shown in figure 13. The edge of the left end of the horizontal arm plate is thinned, the stress concentration part is thickened, the middle floor is hollowed out, and the connection part of the right end is thinned.

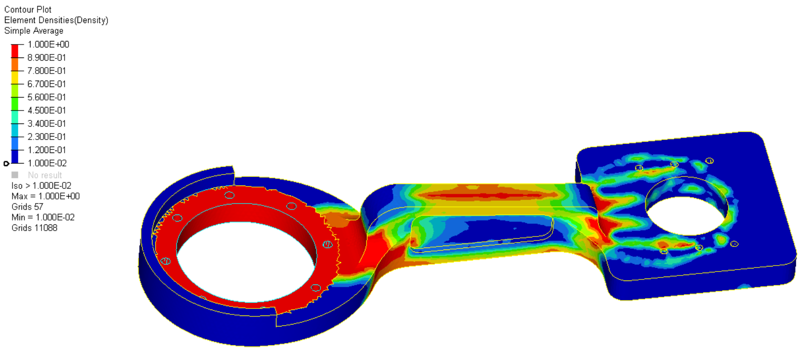


Figure 11. The contour map of density results

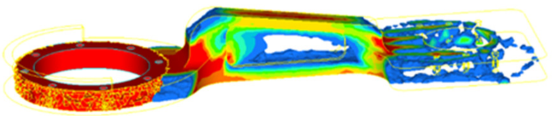


Figure 12. Optimize the layout of the contour map

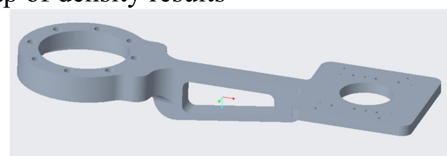


Figure 13. Optimized horizontal arm plate

5.3. Comparison of dynamic and static performance of rehabilitation robot before and after optimization.

The static analysis of the optimized rehabilitation robot was carried out and compared with the structure before optimization. The results are shown in table 2. After optimization, the mass of the model is reduced by 38.8%, deformation and stress are reduced, and the inertia of the rehabilitation robot in three directions around the center of the rotation axis is significantly reduced.

Table 2. Comparison results before and after optimization

Contrast type	before optimization	after optimization
Mass(kg)	68.79	42.1
Static analysis of maximum deformation(mm)	0.50	0.49
Static analysis of maximum stress (MPa)	44.37	31.9
Analysis of maximum deformation in X direction by harmonic response(mm)	0.46	0.32
Analysis of maximum deformation in X direction by harmonic response (mm)	0.98	0.68

Analysis of maximum deformation in X direction by harmonic response (mm)	6.4	3.3
Moment of inertia about the y axis (kg·m^2)	11.3	6.18
Moment of inertia about the z axis (kg·m^2)	17.7	9.48

6. Summarize

In this paper, the static analysis of the harmony response of the upper limb rehabilitation robot is carried out to calculate its deformation and stress in its own gravity and work. According to the analysis results, the structure of rehabilitation robot is optimized. The topological optimization of the horizontal arm plate of the main stressed parts is carried out. The optimized rehabilitation robot has significantly reduced mass and greatly improved mechanical properties. This research method is also applicable to other structural design and analysis.

Acknowledgement
Foundation item: Shandong Science and Technology Development Plan Project Funding (40214010075)

References

[1] Yang, y. (2009) Research on the Arm Rehabi1itative Robot System. Harbin: Harbin Engineering University.
[2] Hu, yc. Ji, LH. (2004) Designing of the Rehabilitation Training Robot for Hemiplegic Upper-limb from the View of the Medical Sciences. Zhongguo Linchuang Kangfu. 8(34): 7754-7756.
[3] Liu, XQ. (2017) Structure Design and Kinematic Analysis of Limb Rehabilitation Robot. Machinery Design & Manufacture. 9: 246-249.
[4] Zhao, M. J. (2018) Mechanism Design and Control of Upper Limb Rehabilitation Mechanical Arm. Yan Tai: Yantai University.
[5] J, X. (2018) Design and Research of Exoskeleton Robot Mechanisms for Shoulder Rehabilitation. Nan Jing: Nanjing University of Posts and Telecommunications.
[6] Ning, Y., Chen, Z.L. (2012) Inverse Kinematics Analysis And Simulation of Seven Degrees Of Freedom Robotic Hand On Matlab. Science Thechnology & Engineering.
[7] Loh, B.G. (2013) Rosen, J. Kinematic Analysis Of 7 Degrees of Freedom Upper-limb Exokeleton Robot With Tilted Shoulder Abduction. International Journal of Precision Engineering & Manufacturing. 14(1): 69-76.
[8] Tobias, N. (2013) Robert, R. Shouler Actuation Mechanisms For Arm Rehabilitation Exoleletons. Proceedings Of The Second IEEE/RAS-EMBS International Conference On Bio-medical, October 19-22, Scottsdale, AZ, USA.
[9] Tobias, N. Robert, R. (2013) Rene, M. Comfort of Two Shoulder Actuation Mechanisms for Arm Therapy Exoskeletons: A Comparetive Study In Healthy Subjects. Med Biol Eng Comput. 51: 71-789.
[10] Bo, S., Zhang, Y.X. Wei, M. et al. (2016) Bilateral Ro-bots For Upper-limb Stroke Rehabilitation: State Of Art Future Prospects. Medical Engineering&Physics, 38(7): 587-606.
[11] Zienkiewicz, O.C. (2014) Taylor R.L, Fox D.D. The finite element method for solid& structual mechanics. Waltham: Butterworth-Heinemann.
[12] Zienkiewicz, O.C. Taylor R.L, Zhu J.Z. (2013) The finite element method its basis& fundamentals. Waltham: Butterworth-Heinemann.
[13] Wang, X. (2017) Finite Element Analysis of Impeller and Frame of Axial Fan. https://wenku.baidu.com/view/8b0ac4405bcfa1c7aa00b52acfc789eb172d9ed5.html.