

International Journal of Engineering Systems Modelling and Simulation

ISSN online: 1755-9766 - ISSN print: 1755-9758

<https://www.inderscience.com/ijesms>

Modelling and structural analysis of coconut tree climbing mechanism

Pradip Solanki, Ravi Kumar Mandava

DOI: [10.1504/IJESMS.2023.10061187](https://doi.org/10.1504/IJESMS.2023.10061187)

Article History:

Received:	14 February 2022
Last revised:	15 February 2022
Accepted:	14 August 2023
Published online:	02 December 2024

Modelling and structural analysis of coconut tree climbing mechanism

Pradip Solanki and Ravi Kumar Mandava*

Department of Mechanical Engineering,
MANIT Bhopal,
462003, India
Email: psolanki586@gmail.com
Email: ravikumar1013@gmail.com
*Corresponding author

Abstract: Due to the shortage of professional climbers for cutting the coconuts, many farmers are looking for coconut tree climbing mechanisms. This article aims to develop the mechanism model and examine the various types of stresses, that is, Von-mises stress and maximum shear stress, and deformation of the various mechanical components of the climbing mechanism. The mechanical parts are required to run the mechanisms are modelled individually in CREO. Further, the analysis has been conducted using ANSYS 2021. The result shows that the stresses acting on each mechanism components are less than the permissible limit of the estimated loading conditions. Therefore, it can be observed that the designed components of the mechanism can effectively withstand the load during operating conditions.

Keywords: climbing mechanism; mechanical components; coconut tree; CREO; ANSYS.

Reference to this paper should be made as follows: Solanki, P. and Mandava, R.K. (2025) 'Modelling and structural analysis of coconut tree climbing mechanism', *Int. J. Engineering Systems Modelling and Simulation*, Vol. 16, No. 1, pp.1–12.

Biographical notes: Pradip Solanki received his BTech in Mechanical Engineering from RGPV University, Bhopal, India in 2012. He received his MTech in Hydro Power Engineering from MANIT Bhopal, Bhopal, India in 2016. From February 2021 to May 2022, he worked as a Junior Research Fellow under sponsored Project by DST in SYST Scheme SP/YO/2019/1052(G) "Design and Development of Coconut Harvesting Robot" at MANIT Bhopal, India.

Ravi Kumar Mandava received his BTech in Mechanical Engineering from Acharya Nagarjuna University, Guntur, India in 2008. He received his ME in CAD/CAM from Andhra University, Visakhapatnam, India in 2010. He received his PhD in the field of Robotics from School of Mechanical Sciences, IIT Bhubaneswar, Odisha, India in 2019. His research areas include legged robotics, motion planning and soft computing. He has published about 45 technical papers in various national, international conferences and journals. He is an active reviewer of some of the international conferences and journals. From June 2020 to May 2023, he worked as an Assistant Professor at MANIT Bhopal, Bhopal, India. Currently, he is working as an Assistant Professor at IIITDM Kurnool, India.

1 Introduction

Over the last few decades, researchers around the world have been widely using several robotic mechanisms in coconut cropping applications, such as cutting the coconuts, spraying pesticides, identifying diseases, etc. But, climbing the coconut tree up to the canopy is a challenging task due to the height of the coconut tree. Moreover, the people who have been climbing the coconut trees for years are facing skin diseases. To overcome this difficulty worldwide, many researchers have developed various climbing mechanisms, but still, they are controlled manually. The overall performance of the developed mechanism depends on mechanical design, and it can be able to climb the irregular surfaces on the coconut tree. It is important to note that the mechanism can withstand the static and dynamic loads

while climbing the tree. Jahan et al. (2018) developed various coconut tree climbing mechanisms for the farmers of Bangladesh. The developed climbing mechanism can take 7.23 m/min, with the height of the tree is 8.90 m. The authors suggested that the developed mechanism can reduce the weight by using aluminium alloys and composite materials. Pengfei et al. (2018) implemented a tree-climbing robot by considering two sets of anti-falling strategies: passive and active mechanisms. Further, the developed mechanism was tested on a rod. Haruhisa et al. (2008) discussed a novel climbing method for pruning robots similar to the traditional timber jacks in Japan. The main advantage of this pruning robot is that the centre of mass is located at the outside of the tree and climbs vertically. Mohmoud et al. (2015) addressed the integration of

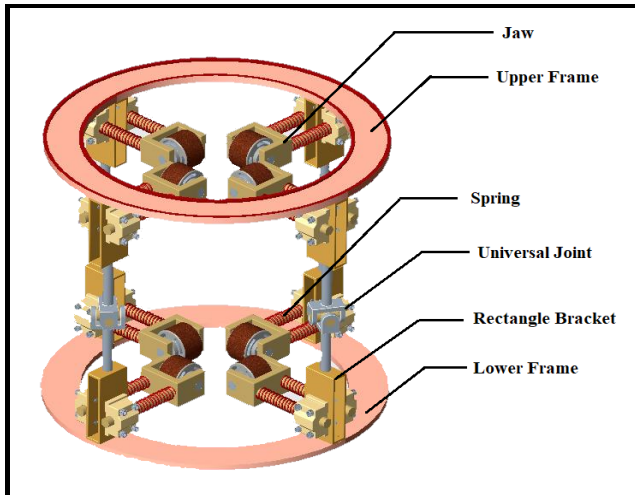
low-cost and lightweight exteroceptive sensors: an accelerometer and an optical flow sensor for Omni climbers. The robot's complexity, weight, and cost significantly increase while compensating for the error due to the wheel slippage. In addition, the authors integrated an algorithm to correct the kinematics of the robot on the curved structures based on the robot's heading angle. Jaikumaran et al. (2017) designed a manual-operated sitting-type climbing device for climbing the coconut tree. The total weight of the device is 9.35 kg, the time required to climb the tree is 3.16 minutes with a height of 12 m, and the angle of inclination between the upper and lower metal wire rope is 40°. Any unskilled person can easily operate the developed device for climbing trees. Afaq et al. (2018) designed a five-axis robotic arm using various materials to withstand the force applied to the end effector. Further, Gzal et al. (2017) discussed the stress distribution in helical springs with an elliptical cross-section and a small helix angle of less than 10°. The authors conducted both analytical and finite element analysis, and it was observed that the stresses were distributed uniformly over the structure, and the maximum stress in the yoke was reduced by 22.8%. To achieve the uniform stress distribution all over the entire steering yoke assembly, Anand et al. (2021) established a new methodology. The authors obtained the generated maximum stress on the yoke is 152.5 MPa, which is less than the yield strength of the material. In addition, Skybaa et al. (2019) focused on developing a 3D model of the slewing bearing subjected to axial load. The authors achieved the maximum stress and deformation at the bearing contact area. Chethan et al. (2019) studied the hip joint of the human body made of different materials, such as cobalt-chromium alloy, titanium alloy, and stainless steel. The authors considered four cross-sectional areas: circular, oval, ellipse, and trapezoidal at the base of the hip joint. The result shows that the minimum stress and deformation at the trapezoidal section is only obtained irrespective of the material. Chung et al. (2010) analysed the overall structure of the heavy-duty industrial robotic manipulator in kinematic and dynamic analysis. The authors conducted a simulation of the rigid body dynamic analysis at 600 kgf and the maximum reaction force obtained at each joint, used for static rigidity analysis. Dong et al. (2016) discussed finite element analysis and modal analysis on the substation water washing robot. The stress distribution and deformation of the robot are determined by static analysis, and the low order natural frequencies and mode shapes are determined by modal analysis. In Bhole and Fegade (2015), the authors discussed the material optimisation and finite element analysis of the connecting rod using two materials: structural steel & Titanium alloys. The developed component is to be optimised, and 1.5% of percentage weight reduction from the standard material and the efficiency of the engine increases. Pan et al. (2017) established a complete static stiffness model for the prismatic joint of a robot using a finite-element and analytical approach. Therefore, static structural analysis is used to determine the stiffness of the mechanical component. Li et al. (2013) designed a new

wearable lower extremity exoskeleton based on human walking postures. The authors conducted FE analysis to determine the stresses at the point of the exoskeleton. To reduce the stress on the leaf spring and improve its performance by experimenting with different materials. Kumar and Matawale (2020) calculated the maximum stress in a mono parabolic leaf spring whose thickness varies from the centre to the outer end in an illustrative manner. Sonowal et al. (2021) discussed static analysis of the loading gear shock absorber of aircraft boeing 787-8. Therefore, the piston is a critical component of the shock absorber and the authors experimented by assuming the two different materials, such as titanium and high-strength stainless steel. The result show that the maximum stress is obtained at the piston material is titanium. Fernandes et al. (2016) conducted the 3D static fracture analysis on the gas turbine compressor. The authors used an open crack model in which crack-tip driving parameters are calculated using 3-D single crack-tip elements in ANSYS. Wang et al. (2019) studied the mechanical characteristics of a 6-DOF spray-painting robot using FE analysis. The model is created using SolidWorks software, and the structural analysis is conducted on ANSYS Workbench. Further, the analysis is helpful to improve the structural strength and the motion efficiency of the robot. Shanmugasundar et al. (2019) discussed the 5-DOF robotic arm in terms of strength, stiffness, and weight, which help determine the robot's structural optimisation design. They calculated using FE analysis to reduce the total mass of the manipulator and provide optimum strength to the robot arm. Bedacht et al. (2020) studied the numerical investigation and material configuration on deep-groove ball bearings. It was concluded that the hybrid ball bearings of various materials have significant advantages, and the overall deformation is less. Tang and Sun (2011) discussed the static structural analysis on a deep groove ball bearing using ANSYS simulations. The result shows that the contact analysis changes in stress, strain, and penetration among ball bearing and its components. Dubey et al. (2016) designed a low-cost coconut tree climbing robot for harvesting the coconuts. The developed robot is made up of two segments connected by a pair of threaded rods powered by motors. It consists of two motorised arms for holding and cutting. Moreover, the cutting arm is operated manually from the ground by using a remote.

From the literature study, it has been observed that very few researchers have developed various climbing mechanisms (Jahan et al., 2018; Gui et al., 2018; Kawasaki et al., 2008; Tavakoli et al., 2015; Jaikumaran et al., 2017) that are not possible to operate without human beings but have not conducted the structural analysis on numerous components of the mechanisms. It is essential to calculate the stresses and deformation of each component of the mechanism while in static conditions. Moreover, many researchers (Gzal et al., 2017; Anand et al., 2021; Skybaa et al., 2019; Chethan et al., 2019; Chung et al., 2010; Dong et al., 2016; Bhole and Fegade, 2015; Pan et al., 2017; Li et al., 2013; Kumar and Matawale, 2020; Sonowal et al.,

2021; Fernandes et al., 2016; Wang et al., 2019; Shanmugasundar et al., 2019) had done structural analysis on various mechanical components such as automobile parts and some robots. But, very few researchers have done the structural analysis on the coconut tree climbing mechanisms. The present research work will develop a novel climbing mechanism that can be operated without a human being. The main objective of this study is to examine the structural stability of the climbing mechanism in static conditions. Therefore, the developed mechanism can withstand the various loads in static conditions and be optimised by better design. The main advantage of this mechanism can climb the irregular and curved surfaces of the coconut trees. Initially, various parts of the mechanism are modelled using CREO and conduct the structural analysis using ANSYS 2021 for determining the stresses and deformation of each component of the mechanism. The developed mechanism consists of several parts, which are shown in Figure 1.

Figure 1 Schematic diagram showing the 3D view of the climbing mechanism (see online version for colours)



2 Mathematical modelling

The mechanism consists of two frames, which are upper and lower frames, connected with the double universal joints with the help of jaw, brackets, and bearings. The double universal joints will support the mechanism to move smoothly on curvature surfaces on the trunk. Moreover, jaws are connected with the wheels, and the mobility of the jaw can depend upon the trunk diameter. To provide sufficient friction force for the wheels, the springs are provided on the jaw rods. The developed friction force can help to support the mechanism without slipping while climbing the upward direction. Further, it is essential to examine the mechanical characteristics of the material before going to manufacture of the mechanism. Because a wide range of loads may be applying either directly or indirectly, the properties of a material will change due to various external factors, such as temperature, wind pressure, etc. The material used for the multiple components of the

mechanism should be cost-effective as well as structurally sound. In the present research work, the authors considered suitable material for each component of the mechanism while conducting the structural analyses, which are mentioned in Table 1. Therefore, it is ensured that the selected material can be able to withstand the load without any failure while climbing the mechanism on a trunk of the coconut tree.

Table 1 The material and mechanical properties used for the mechanism

S. no.	Properties and material	Spring	Universal joint (UJ)	All other part except spring and UJ
1	Material use	Cr-v alloy	Structural steel	Aluminum alloy
2	Young modulus	207 G Pa	210 G Pa	71GPa
3	Modulus of rigidity	79.3 G Pa	70 G Pa	27GPa
4	Poison ratio	0.3	0.3	0.33
5	Density	7,860 kg/m ³	7,850 kg/m ³	2,700 kg/m ³
6	Yield strength	1,550 MPa	340 MPa	280 MPa

2.1 Jaw

It is one of the main parts of the mechanism connecting the wheel and the bracket. In general, two forces are acting on the jaw, which is a vertical and horizontal forces.

- a Vertical force (F_t) acting at static and motion of mechanism which is calculated is as follows:

$$\text{Vertical force}(F_t) = \frac{\text{weight of the mechanism}}{\text{Number of jaws}} - \text{friction forces} \quad (1)$$

The vertical force was calculated when the mechanism is moving in an upward direction

$$\text{Vertical force}(F_t) = \frac{\text{weight of the mechanism}}{\text{Number of jaws}} + \text{friction forces} \quad (2)$$

- b Horizontal force (F_r) is the spring force that depends on the stiffness and compression of spring due to external load.

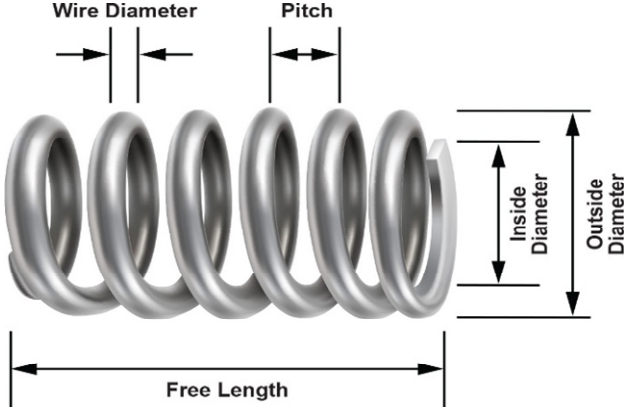
$$\text{Horizontal force}(F_r) = \text{stiffness of spring} * \text{compression of spring} \quad (3)$$

2.2 Spring

It is a flexible part of the mechanism which provides sufficient friction between the trunk of the tree and the wheel. Therefore, the required parameters pitch of spring

(p) is 20mm, mean diameter of spring (D) is 50mm, the diameter of the wire (d) is 4.5mm, and length of spring (L) is 270 mm are considered to determine the friction force.

Figure 2 Schematic diagrams showing the nomenclature of the spring



Length of spring no load condition

$$(L_f) = L_s + X_{\max} + 0.15 * X_{\max} \quad (4)$$

Solid length that is, no gap between the coil

$$(L_s) = n * d \quad (5)$$

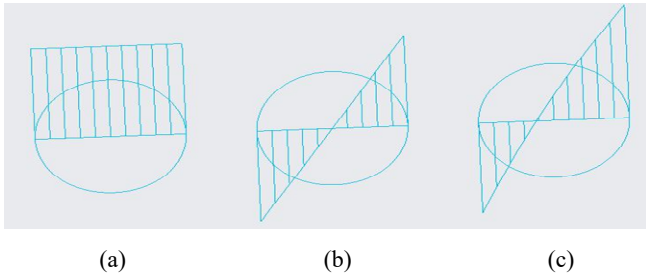
where 'X' represented as deflection of spring, 'n' indicated as number of turn, respectively.

The axial load is applied on a spring, which deforms and generates two types of shear stresses: direct shear stresses and torsional shear stresses.

The direct shear stress is generating due to applied direct axial force acting as a shear load on the spring.

$$\text{The direct shear stress } (\tau_d) = \frac{F_r}{\pi r^2} \quad (6)$$

Figure 3 Schematic diagram showing the various types of stresses in spring, (a) direct shear stress (b) pure torsional stress (c) combination stress (see online version for colours)



The torsional shear stress is generating due to axial load acting as the eccentric shear load on the spring.

$$\text{The torsional shear stress } (\tau_t) = \frac{8F_r D}{\pi d^3} \quad (7)$$

Combined stress is nothing but the algebraic sum of both direct shear stress and torsional stress.

$$\text{Combined stresses } (\tau_r) = \tau_d + \tau_t \quad (8)$$

$$\tau_r = K_s \frac{8F_r D}{\pi d^3} \quad (9)$$

where K_s represented as shear stress concentration factor

$$K_s = 1 + \frac{1}{2c} \quad (10)$$

where 'c' denoted as spring index $\left(c = \frac{D}{d}\right)$.

If considering curvature factor (K_c), then the combined stress is

$$\tau_r = K_s * K_c * \frac{8F_r D}{\pi d^3} \quad (11)$$

where K_w indicated as wall correction factor ($K_w = K_s * K_c$)

$$K_w = \frac{4C-1}{4C-4} + \frac{0.615}{C} \quad (12)$$

$$\tau_r = K_w * \frac{8F_r D}{\pi d^3} \quad (13)$$

The deflection in spring

$$(X) = \frac{8F_r D^3 n}{G d^4} \quad (14)$$

where 'G' represented as modulus of rigidity in Gpa.

Therefore, weight of mechanism (W) = frictional force between the wheel and tree

$$\text{Weight of mechanism } (W) = \mu * R \quad (15)$$

where μ and R represents as coefficient of friction and reaction force between the wheel and tree respectively.

Where ' F_s ' represented as spring force

$$(F_s = K * X) \quad (16)$$

where K is stiffness and X is deflection in spring.

2.3 Rectangle bracket

The developed rectangle brackets are connected between the universal joint and frame of the mechanism and also it will support the jaw-wheel. Moreover, it is also one of the critical parts of the mechanism which withstands various types of loads such as:

- Direct spring force is in a horizontal direction.
- Eccentric loading developed due to the weight of the mechanism, frictional force, and motion.
- The frictional force of bearing is very small, so it is neglected.
- The shock was generated due to running and coconut tree rough surfaces.

Therefore, forces are calculated in two ways.

The horizontal force is generated due to the compression of the spring in between the jaw and the bearing. Therefore, forces act radially in outward directions concerning the frame.

$$\text{Horizontal force } (F_H) = \text{Spring force } (K \cdot x) \quad (17)$$

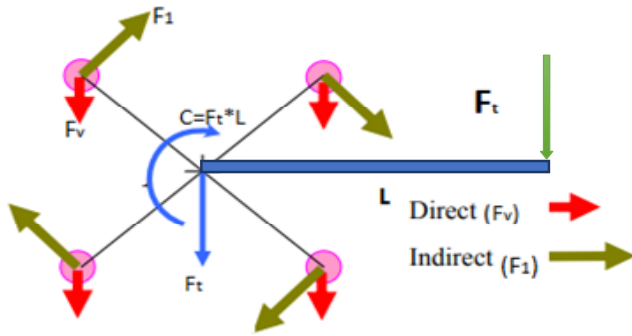
The forces are acting on eccentricity but they are acting on both sides of the rectangle bracket; therefore, the eccentricity effect on both sides is the same, thus neutralising the eccentric impact.

The vertical forces are generated due to the weight and motion of the robot. It is equal to:

$$\text{Primary Vertical load on each jaw } (F_t) = \frac{\text{total load in vertical direction}}{\text{number of jaw}} \quad (18)$$

These vertical forces are eccentric forces with respect to the bracket; thus the effect of eccentricity is primary and secondary forces are generated in the bracket.

Figure 4 Schematic diagram showing the eccentricity effect of force (F_t) on surface of support wall (see online version for colours)



$$\text{Total vertical force } (F_v) = \sqrt{F_v^2 + F_i^2 + 2 \cdot F_v \cdot F_i \cdot \cos \Phi} \quad (19)$$

Here, F_v and F_i are secondary forces generate due to eccentricity, and Φ is the angle between these forces.

Then, the resultant force on each bracket

$$F = \sqrt{F_s^2 + F_H^2 + 2 \cdot F_v \cdot F_H \cdot \cos \theta} \quad (20)$$

where F_s and F_H are the algebraic sum of vertical and horizontal force, respectively, and θ is the angle between vertical and horizontal forces. Therefore, the maximum resultant force where $\theta = 90^\circ$.

Therefore,

$$F_{\max} = F_s + F_H \quad (21)$$

2.4 Universal joint

The universal joint connects and transmits motion and forces between the upper and lower frame, and it also gives angular flexibility. They have mainly two parts: as jaw and the spider. Therefore, two forces are acting on the universal joint which are, 2

- weight of robot
- due to the motion of the robot.

2.5 Frame

The frame of the mechanism consists of two parts which are upper and lower frames. In the present research work, the authors have conducted two types of analysis: symmetric analysis on the X-Y plane and Y-Z plane. Two types of failures have been observed while performing the systematic analysis on frames: Tearing and crushing of the plate.

3 Result and discussion

This article attempts to determine the deformation behaviour, von-misses stress, maximum shear stress, and directional deformation by conducting the structural analysis on the coconut tree climbing mechanism. Moreover, the stresses related to various components of the mechanism are subjected to withstand several loads without any failure. In this analysis, the results are validated with the structural stability of the mechanism. It is to be noted that conducting the structural analysis is essential for each mechanical design component. Therefore, structural analysis in this study was performed in ANSYS 2021 using standard methodology. The step by step procedures for conducting the structural analysis are as follows:

- **Material selection:** A wide variety of materials are available in ANSYS 2021. But in this research work, the material selected in the mechanism is an aluminium alloy because it helps to minimise the weight of the mechanism without compromising the strength and increases the mechanism's performance.
- **Meshing:** Meshing is an essential step for achieving better results. In the present research work, the authors applied uniform meshing, which is a combination of tetrahedral and hexahedral elements. The meshing is refined in complex sections like holes, corners, etc. because in these parts, it fails due to more surfaces in the small area. For improvement of result, the meshing quality should be high. Thus, checking optimises the meshing structure and the number of grids.
- **Boundary condition:** Once the meshing is done on the component, the boundary conditions are fixed before conducting the structural analysis. The common boundary conditions applied on a mechanism like dead load, external load, velocity, temperature, pressure, etc., while conducting the structural analysis.
- **Solution:** Once the above steps are completed, the results related to deformation, von-misses' stresses, maximum shear stresses, and directional shear stresses are provided in further sub-sections.

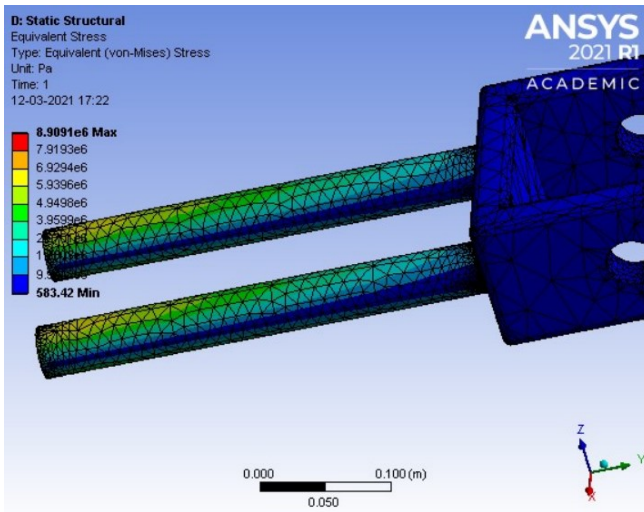
3.1 Jaw

Figure 5 shows the structural analysis and result related to the jaw rod's various stresses and deformation behaviour. The maximum and minimum stresses and deformation behaviour are given in Table 2. It can be observed that the stresses and deformation achieved from the ANSYS are less than the failure limit of the jaw. Moreover, the various stresses obtained from each jaw section are less than the failure stresses generated by aluminium alloy. The results indicate that the jaw is safe and it will meet the design requirements.

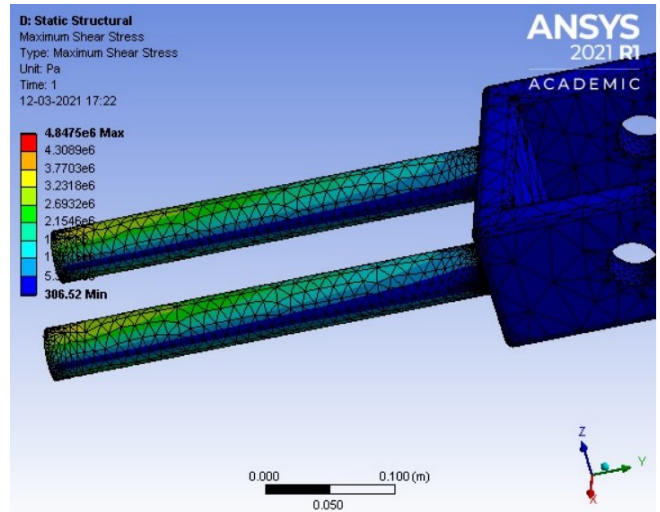
Table 2 The maximum and minimum value of stresses and deformation behaviour of the jaw rod

S. no.	Parameter	Critical section	Max. value	Min. value
1	Normal stress	Fix support	8.9891×10^6 Pa	583.42 Pa
2	Shear stress	Fix support	4.847×10^6 Pa	306.52 Pa
3	Total deformation	Free section of jaw	0.4141 mm	0.00 mm
4	Directional deformation (y-axis)	Free section of jaw	8.195×10^{-2} mm	6.47×10^{-2} mm

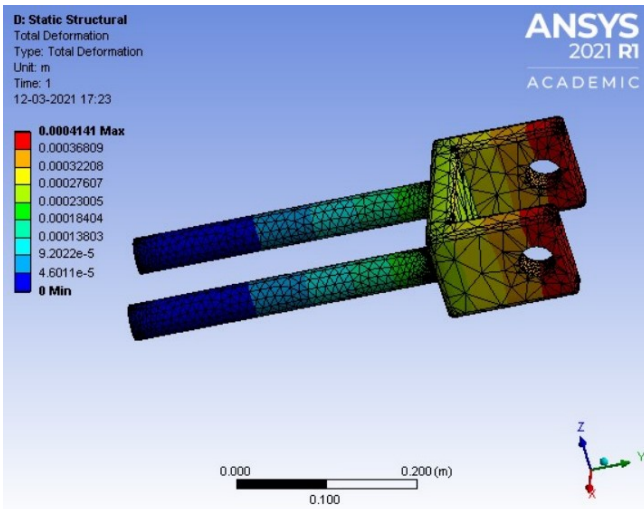
Figure 5 Results show the structural analysis of the jaw rod, (a) von-mises stress (b) maximum shear stress (c) total deformation (d) directional deformation (see online version for colours)



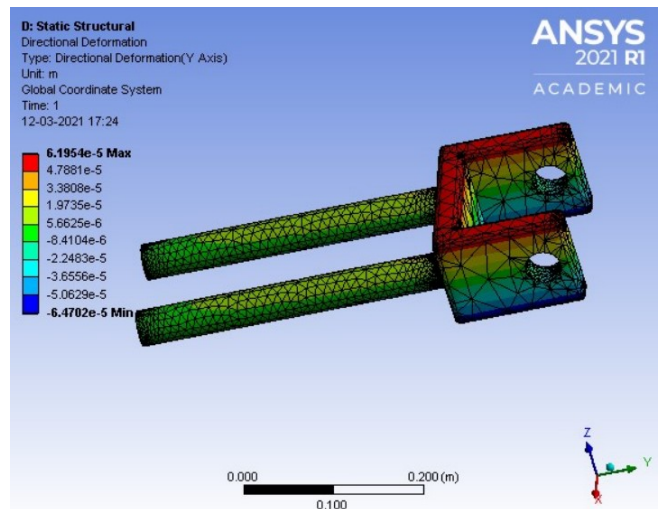
(a)



(b)



(c)



(d)

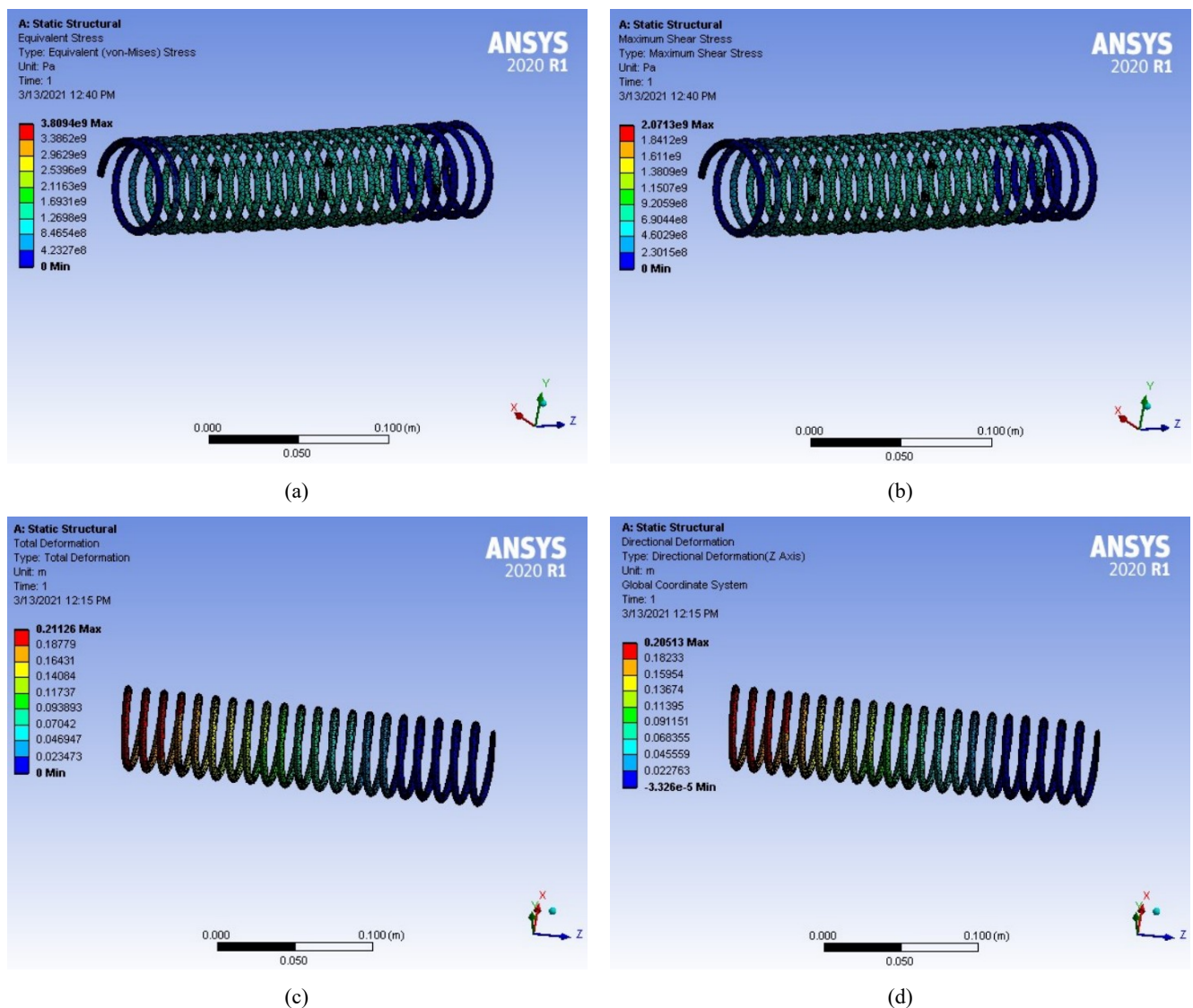
3.2 Spring

Spring is a flexible element in the mechanism that deforms depends on the trunk diameter and provides the frictional force, which is helpful to hold the mechanism with the trunk without any failure. From Figure 6, it can be observed that the maximum deformation of the spring is 20 cm and the stiffness of the spring 1,500 N/m. Moreover, the failure of the spring mainly depends upon the shear stress. The stresses obtained from Figure 6 are less than the failure limit of the spring. According to the static analysis, all the shear stress values in each coil are less than the allowable stress limit of the Cr-V alloy material. Table 3 shows the maximum and minimum stresses and deformation behaviour of the spring. The results obtained from the analysis will be safe and perform their function while climbing the mechanism on a trunk.

Table 3 The maximum and minimum value of stresses and deformation behaviour of the spring

S. no.	Parameter	Critical section	Max. value	Min value
1	Normal stress	End of spring	380.94×10^6 Pa	152×10^6 Pa
2	Shear stress	End of spring	207.13×10^6 Pa	76.45×10^6 Pa
3	Total deformation	End of spring	211.26 mm	0.00 mm
4	Directional deformation (z-axis)	End of spring	205.13 mm	-0.0332 mm

Figure 6 The structural analysis of the spring, (a) von-misses stress (b) maximum shear stress (c) total deformation (d) directional deformation (see online version for colours)



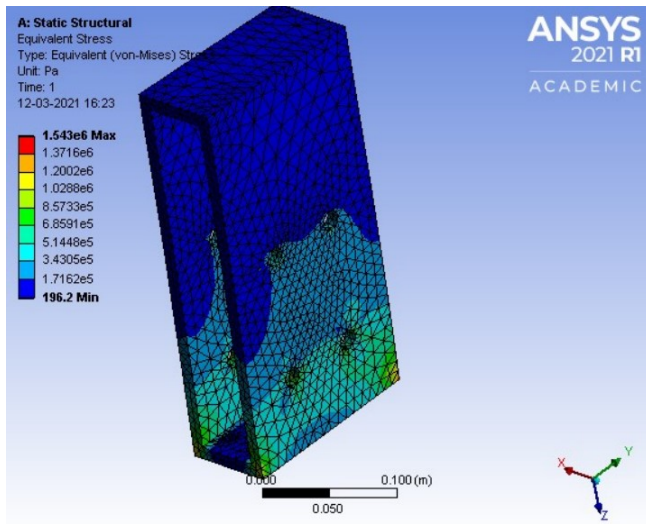
3.3 Rectangular bracket

It is a connection between the frame and the universal joint. To conduct the static analysis on the bracket, applied the boundary conditions like fixed at one end and free at the other end, and then applied primary and secondary force on the bracket, which is previously calculated. The results related to the static analysis of the rectangular bracket are shown in Figure 7. It can be observed that the eccentric load acting on a rectangle bracket with one end fixed and the other end free (that is, a strut beam) usually fails due to bending stress. One side of the rectangular bracket is bolted to the frame while conducting the analysis. The bending and shear stress values (ref. Table 4) at each point are less than the permissible stress limit of aluminium alloy. Therefore, the designed rectangle bracket is safe and rigidly connected to the frame and universal joint.

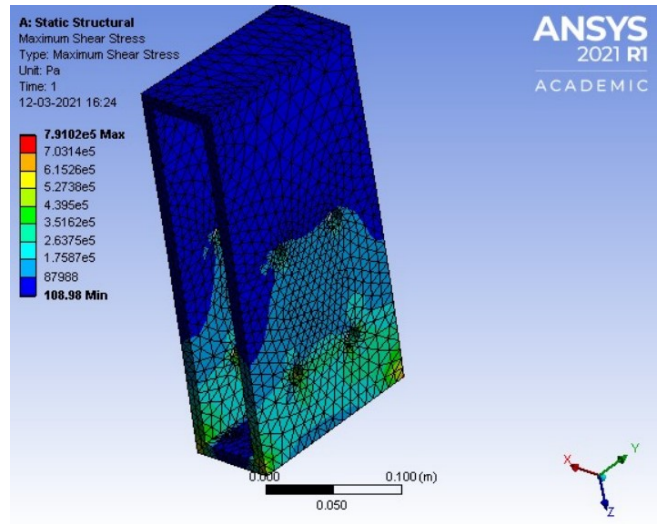
Table 4 The maximum and minimum value of stresses and deformation behaviour of the rectangular bracket

S. no.	Parameter	Critical section	Max. value	Min. value
1	Normal stress	Fix support and holes	1.543×10^6 Pa	196.20 Pa
2	Shear stress	Fix support and holes	7.910×10^6 Pa	108.98 Pa
3	Total deformation	upper section	0.003806 mm	0.00 mm
4	Directional deformation (y-axis)	Upper section	0.003685 mm	0.00 mm

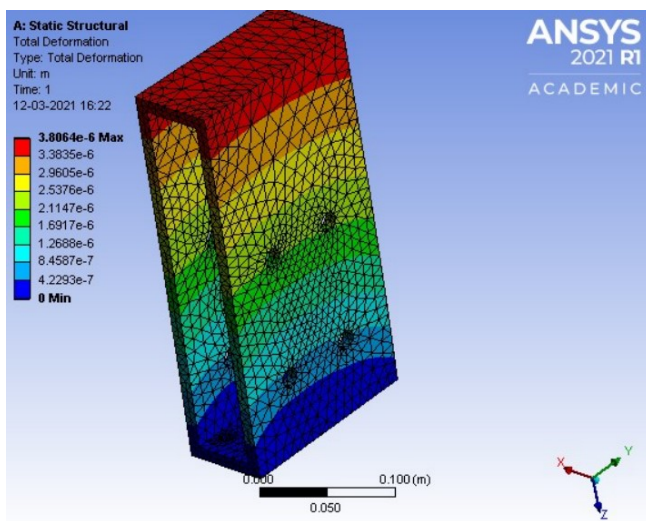
Figure 7 The structural analysis of the rectangular bracket, (a) von-misses stress (b) maximum shear stress (c) total deformation (d) directional deformation (see online version for colours)



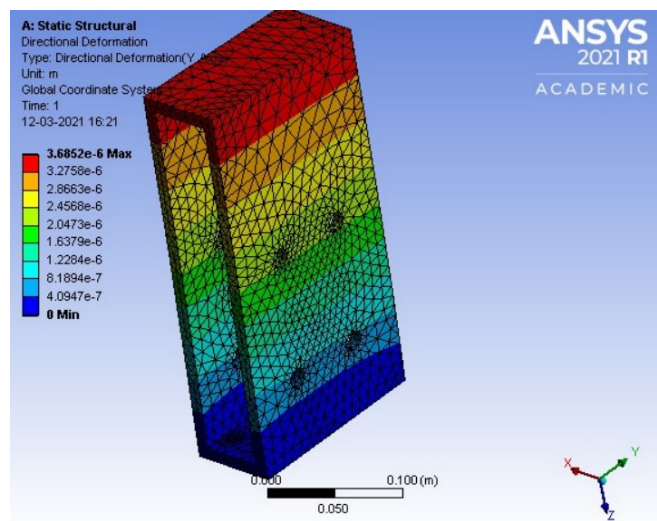
(a)



(b)



(c)



(d)

Figure 8 The structural analysis of the universal joint, (a) von-misses stress (b) maximum shear stress (c) total deformation (d) directional deformation (see online version for colours)

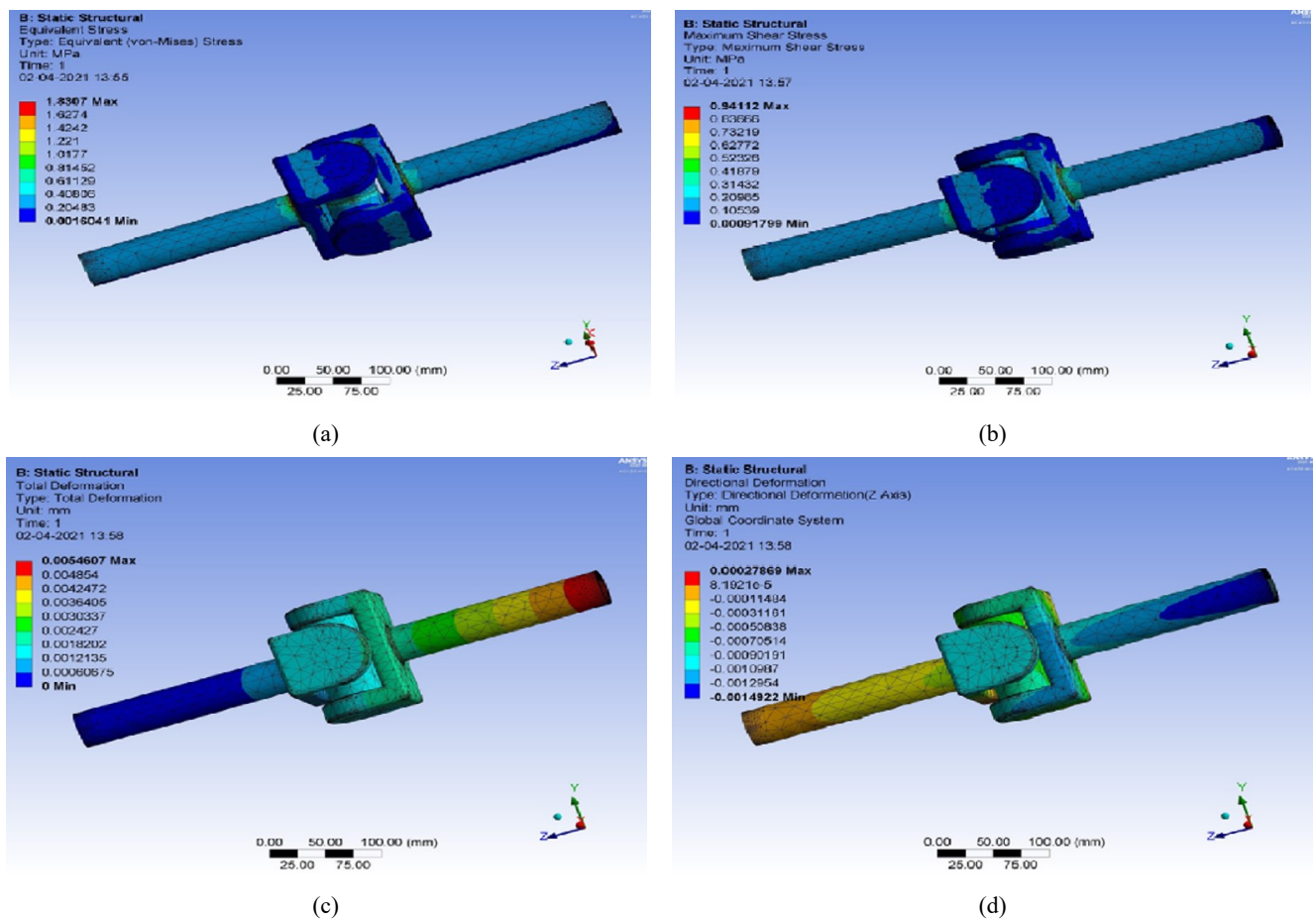


Table 5 The maximum and minimum value of stresses and deformation behaviour of the universal joint

S. no.	Parameter	Critical section	Max. value	Min. value
1	Normal stress	Holes in jaw section	1.837×10^6 Pa	1,604.10 Pa
2	Sear stress	Holes in jaw section	0.911×10^6 Pa	971.99 Pa
3	Total deformation	Rod in jaw	0.00149 mm	0.00546 mm
4	Directional deformation (z-axis)	Rod in jaw	0.0002786 mm	0.00 mm

3.4 Universal joint

It is connected between the upper and lower frames, and it will allow providing the moment on the curvature surface of the trunk of the tree. The boundary conditions of the universal joint are one end fixed and the other end subjected to a load of 250 N. The results (ref. Figure 8) show the maximum stress and deformation of the universal joint at various conditions. It has been observed that the developed stresses and deformation of the universal joint will connect the upper and lower frames without failure. The jaw section

of the universal joint where the spider joint is connected achieved maximum shear stress due to stress concentration, which is called critical sections. Table 5 shows the maximum and minimum stresses and deformation behaviour of the universal joint.

Table 6 The maximum and minimum value of stresses and deformation behaviour of the spider

S. no.	Parameter	Critical section	Max. value	Min. value
1	Normal stress	Connecting section	0.337×10^6 Pa	584.61 Pa
2	Sear stress	Connecting section	0.3798×10^6 Pa	334.43 Pa
3	Total deformation	Connecting section	0.0002581 mm	0.00 mm
4	Directional deformation (z-axis)	Connecting section	7.25×10^{-6} mm	-2.46×10^{-4} mm

3.5 Spider

A spider is a mechanical element that connects both jaws and transfers the load from one jaw to the other jaw in a universal joint. It is failed primarily due to shear stress in the critical section that fit into the jaw hole. Figure 9 shows

the stress and deformation at various points of the spider using static analysis. The results related to the maximum stress and maximum deformation at the critical section are lower than the permissible limit given in Table 6.

3.6 Frame

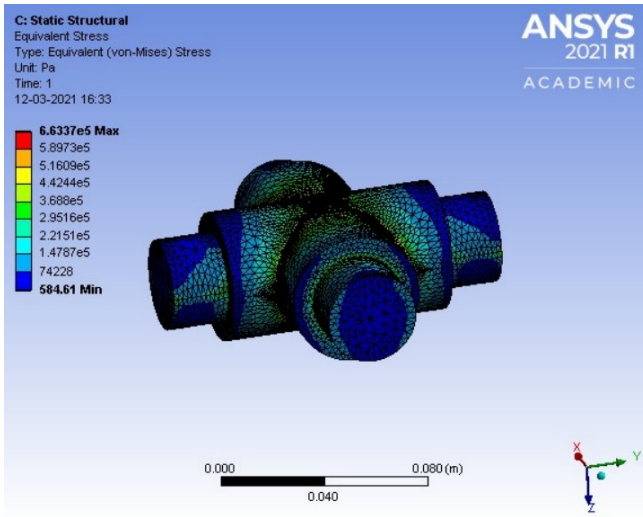
The frame provides rigid support for the other all mechanical components. There are two frames one is an upper frame, and the second one is the lower frame. While climbing the mechanism, two eccentric vertical and horizontal forces are acting on the frame. To conduct the static analysis, the two vertical and horizontal forces are fixing the base. The results related to the static analysis are shown in Figure 10. It can be observed that the maximum shear stress and deformation values are less than the frame

failure limit, which is mentioned in Table 7. Moreover, the proposed design can support the various components of the mechanism without any failure.

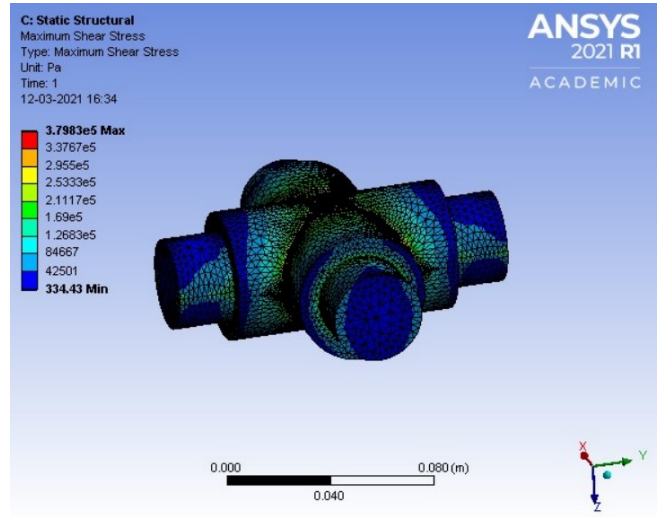
Table 7 The maximum and minimum value of stresses and deformation behaviour of the frame

S. no.	Parameter	Critical section	Max. value	Min. value
1	Normal stress	Base of bracket	1.257×10^6 Pa	473.82 Pa
2	Sear stress	Base of bracket	6.322×10^6 Pa	262.08 pA
3	Total deformation	Base of bracket	0.00277 mm	0.0 m

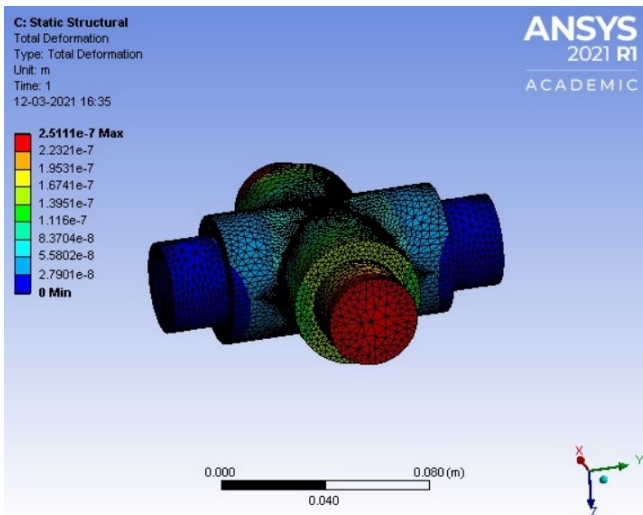
Figure 9 The structural analysis of the spider, (a) von-misses stress (b) maximum shear stress (c) total deformation (d) directional deformation (see online version for colours)



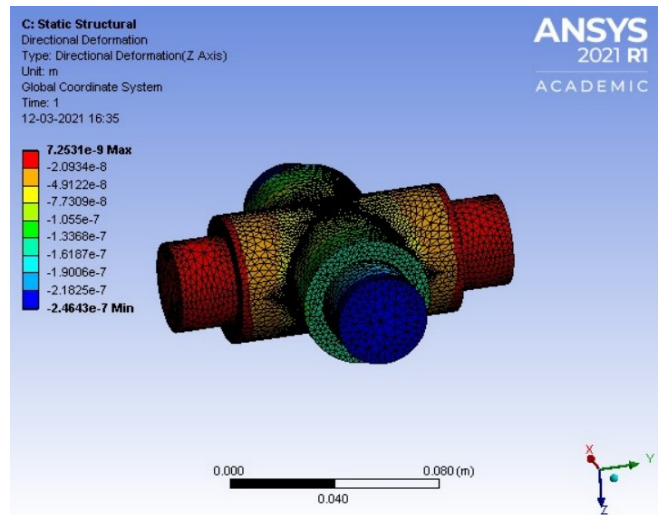
(a)



(b)

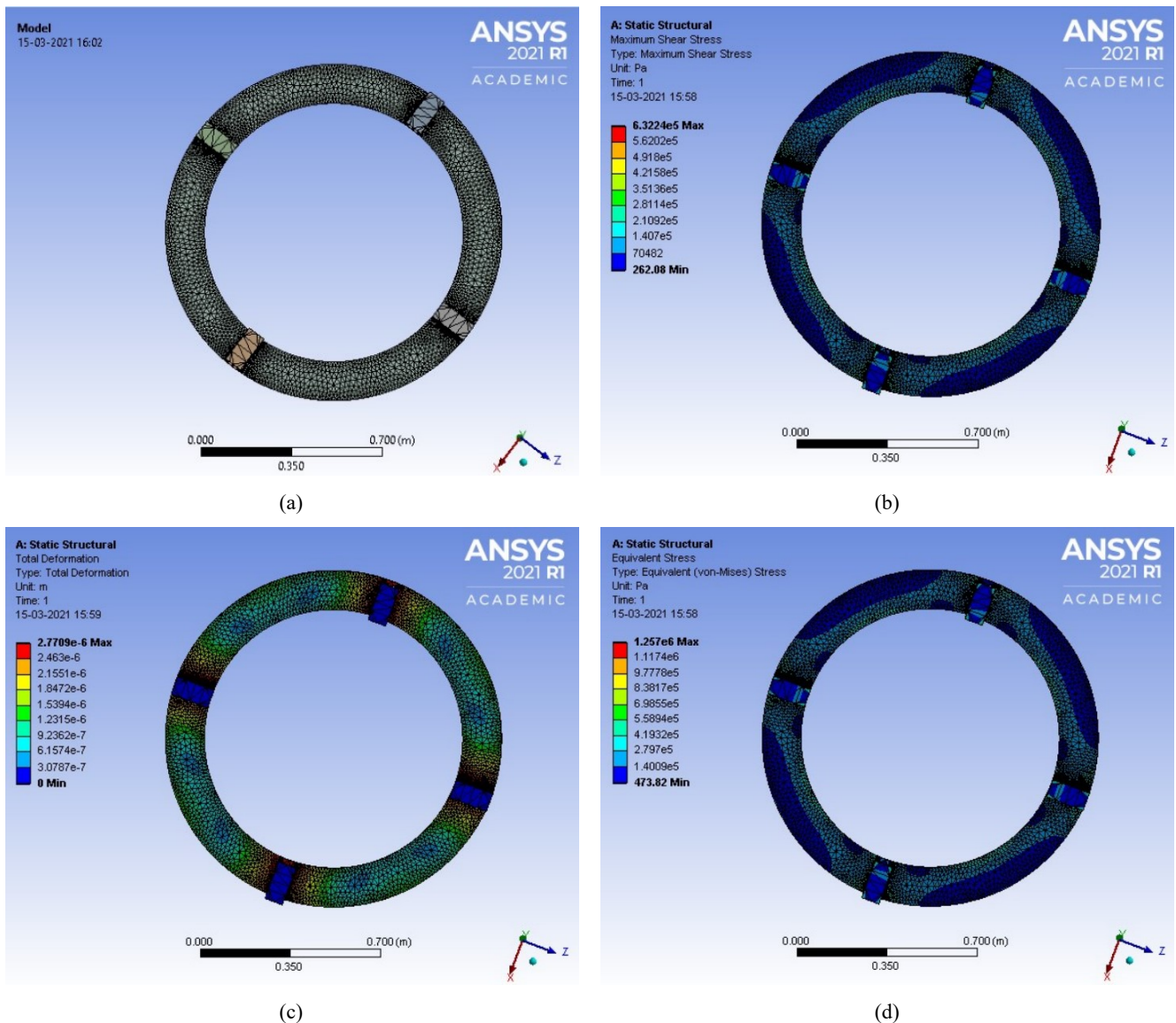


(c)



(d)

Figure 10 The structural analysis of the frame, (a) von-mises stress (b) maximum shear stress (c) total deformation (d) directional deformation (see online version for colours)



4 Conclusions

The authors designed a coconut tree climbing mechanism for cutting the coconuts without any human intervention in this work. A CAD model has been developed in CATIA with proper dimensions. Structural analysis has been conducted on various components of the mechanism using ANSYS 2021 to understand the deformation behaviour and determine the various stresses developed on the design. After conducting the structural analysis, it has been observed that the stresses and deformation developed by the individual components of the mechanism are less than the failure limit. Hence, the proposed design is structurally safe under esteemed loading conditions. The authors will also conduct the dynamic analysis on the developed mechanism while climbing the mechanism on a coconut tree, which we consider as a future work.

Acknowledgements

The authors are thankful to DST Govt. of India under SYST Scheme for funding the project number SP/YO/2019 /1052(G)

References

- Abbasi, A.A., ul Hasan, E. and Khan, A. (2018) 'Static force, torque and structural analysis of 5R robotic arm – simulation and practical implementation', in *2018 4th International Conference on Control, Automation and Robotics (ICCAR)*, IEEE, pp.65–70.
- Anand, K., Nithin, M. and Sandeep, A. (2021) 'Numerical and experimental analysis of torsional stress of traditional and modified steering yoke', *Science Direct Materials Today: Proceedings*, Vol. 46, pp.7099–7104.

- Bedacht, D., Buennagel, C., Monir, S., Uria, I. et al. (2020) 'Numerical investigation and static structural analysis of deep groove ball bearings using ANSYS FEA', *International Workshop on Electric Drives: MPEI Department of Electric Drives 90th Anniversary (IWED)*, Moscow, Russia.
- Bhole, K.S. and Fegade, V.T. (2015) 'Nonlinear static finite element analysis and material optimization of connecting rod', *International Journal of Engineering Research & Technology (IJERT)*, Vol. 3, No. 1, pp.1–5.
- Chethan, K.N., Zuber, M., Shenoy, S. and Kini, C.R. (2019) 'Static structural analysis of different stem designs used in total hip arthroplasty using finite element method', *Heliyon*, Vol. 5, No. 6, pp.1–8.
- Chung, G-J., Kim, D-H., Shin, H. and Ko, H-J. (2010) 'Structural analysis of 600Kgf heavy duty handling robot', *IEEE Conference on Robotics, Automation and Mechatronics*.
- Dong, X., Su, J., Wang, Y., Chen, X. and Lu, S. (2016) *Finite Element Analysis and Vibration Control of the Substation Equipment Water Washing Robot with Hot-line*, IEEE.
- Dubey, A.P., Pattnaik, S.M., Banerjee, A., Sarkar, R. and SaravanaKumar, R. (2016) 'Autonomous control and implementation of coconut tree climbing and harvesting robot', *Procedia Computer Science*, Vol. 85, pp.755–766.
- Fernandes, R., El-Borgi, S., Ahmed, K., Friswell, M.I. and Jamia, N. (2016) 'Static fracture and modal analysis simulation of a gas turbine compressor blade and bladed disk system', *Advanced Modeling and Simulation in Engineering Sciences*, Vol. 3, No. 1, pp.1–23.
- Gui, P., Tang, L. and Mukhopadhyay, S. (2018) 'A novel robotic tree climbing mechanism with anti-falling functionality for tree pruning', *Journal of Mechanisms and Robotics*, Vol. 10, No. 1.
- Gzal, M., Groper, M. and Gendelman, O. (2017) 'Analytical, experimental and finite element analysis of elliptical cross-section helical spring with small helix angle under static load', *International Journal of Mechanical Sciences*, Vol. 130, pp.476–486.
- Jahan, N., Amin, M.N., Hossain, M.I. and Sheikh, M.S.I. (2018) 'Development of a low cost coconut tree climber for small farmers in Bangladesh', *International Journal of Engineering Research & Technology*, Vol. 7, No. 3, pp.242–248.
- Jaikumar, U., Joseph, S., Preman, P.S., Unnikrishnan, C., Jitha, K.J. and Joseph, C.J. (2017) 'Design and development of sitting type coconut palm climbing device 'Kera Suraksha Coconut Climber'', *Journal of Tropical Agriculture*, Vol. 54, No. 2, pp.136–143.
- Kawasaki, H., Murakami, S., Kachi, H. and Ueki, S. (2008) 'Novel climbing method of pruning robot', *SICE Annual Conference*, 20–22 August, pp.160–163.
- Kumar, P. and Matawale, C.R. (2020) 'Analysis and optimization of mono parabolic leaf spring material using ANSYS', *Materials Today: Proceedings*, Elsevier, Vol. 33, pp.5757–5764.
- Li, M-C., Wang, L-R., Chen, D-M., Tao, J-J. and Chen, D. (2013) 'The static analysis of wearable lower extremity exoskeleton based on ANSYS workbench', in *2013 IEEE 9th International Conference on Mobile Ad-hoc and Sensor Networks*, IEEE, pp.510–514.
- Pan, X., Wang, H., Hu, M. and Tian, Y. (2017) 'Static stiffness modeling of the prismatic joint of a modular reconfigurable robot', *IEEE International Conference on Robotics and Biomimetics*, Macau SAR, China.
- Shanmugasundar, G., Sivaramakrishnan, R., Meganathan, S. and Balasubramanid, S. (2019) 'Structural optimization of an five degrees of freedom (T-3R-T) robot manipulator using finite element analysis', *Applied Materials Research*, Vol. 16, pp.1325–1332.
- Skybaa, R., Hrccka, S., Smetankaa, L. and Majchraka, M. (2019) 'Strength analysis of slewing bearing', *Transportation Research Procedia*, Vol. 40, pp.891–897.
- Sonowal, P., Pandey, K.M. and Sharma, K.K. (2021) 'Design and static analysis of landing gear shock absorber of commercial aircraft', *Materials Today: Proceedings*, Elsevier, Vol. 45, pp.6712–6717.
- Tang, Z. and Sun, J. (2011) 'The contact analysis for deep groove ball bearing based on ANSYS', *Procedia Engineering*, Vol. 23, pp.423–428.
- Tavakoli, M., Lopes, P., Sgrigna, L. and Viegas, C. (2015) 'Motion control of an omnidirectional climbing robot based on dead reckoning method', *Mechatronics*, Vol. 30, pp.94–106.
- Wang, D., Huang, Z., Zi, B., Pang, J., Zhang, H. and Zheng, L. (2019) 'Simulation and analysis of mechanical characteristics of a 6-DOF spray-painting robot', *China IEEE International Conference on Mechatronics and Automation*, Tianjin.