Structural Analysis and Optimization of Dental Chair

Hare Sachin Shridhar[1], Prof. B. S. Allurkar[2]

[1]PG Student, [2]Associate professor, P.G. Department of Manufacturing Processing Engineering, MBES College of Engineering, Ambajogai, Maharashtra, India.

---***---

Abstract - *Structural analysis of dental chair is an important aspect while studying patient's dental anatomy. It's a specially designed and equipped medical instrument which supporting a movement of patient and the instruments applied during the treatment. It's usually reclines and positions the patients whole body by driving pedestal mechanism. These mechanism sometimes provides axial as well as linear motion and are attached to a Dental Engine. This engine is a companion device that provides power, suction, and water also considering patients comfort during treatment.*

Dental chair are very useful assets in examining, extracting and doing surgeries driving with pedestal mechanism and utilizing hydraulic power ensuring smooth and quiet adjustment of all moving parts. Patients comfort are provided using headrest, backrest system with stylish, fully adjustable and comfortable cushions. To this end, dental chairs can contain a bewildering array of attachments, either on the chair itself or on the ubiquitous Dental Engine: spittoon bowls, suction tubes, pneumatic tubes to power various parts of the equipment used in cleaning and surgeries, etc.

Recent changes in dental chairs consist of metal and plastic bodies and also provides hygiene and ease to clean facilities to avoid any bacterial infusion to minimize the risk of infection. Particular sitting arrangement with leg rest folded away makes dentist and patients interact more comfortably during face-to-face consultation and treatment. Dental chairs also provides vertical and horizontal adjustment have the ability to adjust the height of the chair according to the operating requirements of the dentist or the patient. The current study focuses on the structural analysis and optimization of the lifting design of the dental chair according to the weight load and medical requirements to adjust the scissor link length for a stronger, safer and optimized design.

Key Words: Dental Chair, Dental Engine, recliner, articulated, FEA

1. INTRODUCTION

Oral health is an essential part of the general health care. There are many oral health issues that concern India despite having the maximum number of dental schools in the world. India lacks basic data to know the exact prevalence of oral diseases, which is the first requirement for setting any national policy or manpower allocation. India is a vast country, nations within nations, with great diversity in diet and behavior [1]. Reaching the dental care services to the lower and middle class people is really a challenging task as there is still lack of awareness among these peoples. Even if sometimes it reaches to these people many of these dental care centres are not having sufficient working facilities and services. Dental units are devices which are useful in examining and caring of teeth and mouth (like drilling, filling, cleaning and examination) [2]. Dental chair is an integral part of dental unit providing functionalities like patients leg and back inclination as well as rise and fall of chair and it can also have ability to move up and down using motor and hydraulic mechanisms [3].

This paper will be defined as optimization of dental chair structure as per the anthropology and medical requirement to adjust the scissor link length for stronger, safer and optimized design.

1.1 Dental Chair

The Dental Chair unit is specially designed to operate oral health instruments by easily moving the patient's whole body in up and down side so that doctor can easily position patient according to treatment (refer fig.1). The movement to dental chair is actually realized with actuators and pneumatics and provided at pedestal position. Now a days these mechanism can be controlled by remote. All devices and dental chair are integrated to form a dental engine providing power, suction, and water like mechanisms.

Fig -1: Dental Chair

This dental chairs is either allied with variety of attachments, or had a separate Dental Engine with spit bowls, suction tubes, and pneumatic tubes to power various pieces of equipment.

1.2 Paper Outline

This paper gives an idea about actual work done on linear static analysis of Dental chair system and optimization of scissor link lengths. We have provided respected CAD models and designs of experiment.

1.3 Recent Survey on FEA

S. P. Chaphalkar, S. N. Khetre, A. M. Meshram had explained the modal analysis. This paper presents some basic concepts of modal analysis of transverse vibration of fixed free beam which is specially designed to absorb the shocks. It is described an experimental apparatus and the associated theory which allows to obtain the natural frequencies and modes of vibration of a cantilever beam. The concept of modal analysis plays an important role in the design of practical mechanical system. So, it becomes important to study its effects on mechanical system for different frequency domain i.e. low, medium and high frequency [4].

Agashi, Andre & Daywin, Frans & Gozali, Lina & Adianto, & Purna Irawan, Agustinus had proactively studied and presented a design of swivel chair for bending machine work station which will help the workers to easily do their cumbersome works and increase their productivity process. Due to long duration of standing work workers may experience some physical fatigues, the research is mainly focused to calculate and to reduce this fatigue with the help of swivel chair by considering the Nordic Body Map [5].

F Bham, H Perrie, J Scribante and C-A Lee have done audit on how paediatric dental chair sedation is useful in the current practice of dentist in South Africa. They have presented a report which shows a prospective, contextual and descriptive study design with randomly selected dental practitioners [6].

A. Bijalwan and A. Misra represented and compared traditional with wearable chairs which are used at compact work places. They have mainly focused on the mechanical design and finite element analysis (FEA) of the mechanism which are satisfying equilibrium and stability criterion and which are capable of reducing the physical fatigue among the workers in assembly line and factories [7].

From literature review, Design Optimization & Analysis of a Dental Chair for Linear Static Structural Analysis is extremely important for analysing weight and size of various components, designers have to change the different sizes of scissor length of Dental chair structure, this may lead to over design or under design.

2. PROBLEM DEFINATION AND FEA

As discussed above we have deliberated the need of structural optimization of dental chair lifting mechanism as per the customer requirements, here we would like to design the lifting system and based on the weight and length of the scissor link, we would like to analyze the frame for correct sizing of component by considering the defined constrained.

We also like to compare the weight, deflection, factor of safety for traditional and new design optimize vertically.

Partial differential equations are used to define complex physical phenomena in the form of numerical techniques in Finite Element Analysis (FEA) (also called Finite Element Method (FEM)). This method is widely used in many problem solving areas like, structural analysis, mass transport, fluid flow and electromagnetic potential.

We can use the FEM model to predict and find the behavior of system state or response to external loads according to the design variables applied under their operating conditions. Design study is a simulation of mechanical and thermal systems, to reduce the design cycle time, and to improve overall system performance for different scenarios.

Steps to follow in FEA process are as below:

Geometric representation creates the geometry of the problem by defining features of the system and considering different design variables and various environmental conditions. These are then analyzed and stored in a CAD database to create some useful graphics.

Element formulation this step develops the equations that describe the behavior of each finite element by using some mathematical numerical techniques by discretizing them into small elements. The material properties of each element are taken into account when formulating the governing element formulation equations. This involves obtaining a set of algebraic equations to solve displacement function within each element.

Assembly Gets a set of global equations for the entire model from the individual element equations. The loads and support (boundary) conditions are applied to the appropriate nodes of the finite element mesh. This assembly is done by interconnecting many smaller bodies or units also called finite elements at common points.

Solution of equations provides computation of the unknown values of the primary field variables and the solution for the unknown nodal degrees of freedom (or generalized displacements). These solutions are used then to evaluate additional, derived variables like element stresses, heat flow and reaction forces.

2.1 Components of FEA Model

We need a higher accuracy requirements by a lot more complex problem data. A FEA model is composed of several different components that together describe the physical problem to be analysed and the results to be obtained. The computational model designed for simulation using FEA includes various factors like element properties, discretized geometries, dynamics, general loads and boundary conditions, material properties, analysis type, and output requirements.

2.2 Discretized geometry

Discretised geometry is made up of small integrated finite bodies or elements. These elements and nodes describes the basic geometry of the physical structure being simulated or modelled.

These finite elements are interconnected at some common points to form the physical structure of the design which are discrete in nature and which in turn share some common properties and behaviour. Node coordinates and element connectivity—that is, which nodes belong to which elements—make up the geometry of the model. The set of all elements and nodes in a model is called a mesh. In general, the mesh is only an approximation of the actual geometry of the structure.

The type of element and the total number of elements used in the mesh affects the results obtained by the simulation.

The greater the mesh density is directly proportional to the accuracy and the results obtained from the simulation. As the mesh density increases, the analysis results converge to a unique solution, and the computer time.

2.3 Element properties

The selection of proper element and order (e.g. linear, parabolic, and cubic) is very necessary in the problem data simulation. The choice of the element defines the function of displacements between the interconnected points. The validity of simulation results in feature libraries is limited by the availability of accurate material data.

Complex models can be easily understood with 3D models which gives better illustration of deformation. 3D simulation model is that the each and every part is meshed with 3D objects. The similarity when modelling with 1D element, 2D element, and 3D element is that they are able to generate the same result parameters. 3D solid elements only accounts for translational displacements.

The simulation of individual finite element is displayed in fig. 2 and generated simulation of lifting chair by applying elements properties and material data on given dental chair model is shown in fig. 3.

In this paper we have taken some common measurement of dental chair which are sown in fig. 4. Different common element type is shown in fig. 5

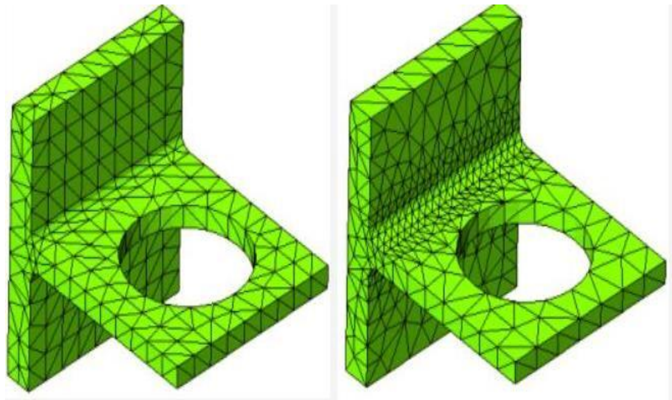

Fig -2: CAD Model divided in Small Pieces (Elements)

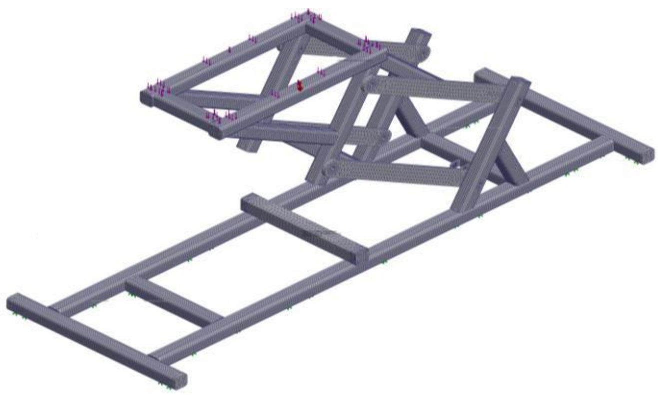

Fig -3: CAD model 3D simulation

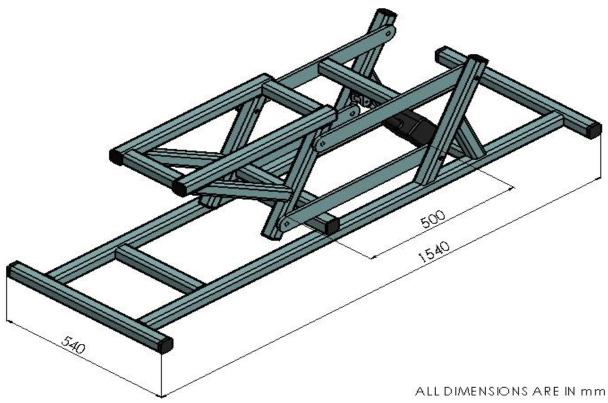

Fig -4: Dimension of Lifting Mechanism in Dental Chair

2.4 Material Data

Material data is an important factor while creating a high accuracy simulation. To achieve the high accuracy for complex structures we required high quality material data. All materials behaves differently according to different environmental conditions applied on it and possess different boundary conditions to each environmental factors. Variations and number of variations of material properties can give more accurate results and simulation performance.

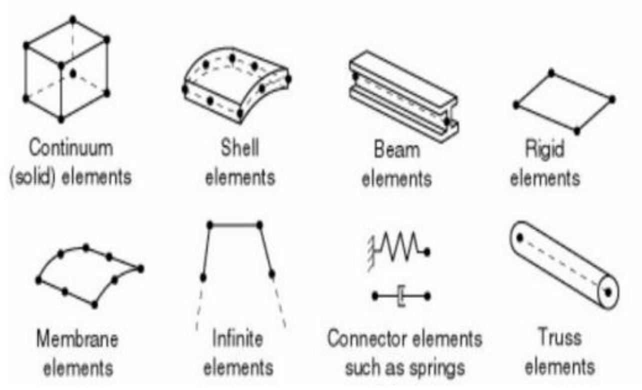

Fig -5: Common element families

2.5 Loads and boundary conditions

In addition to the modeling method and the application of loads and boundaries conditions, 1D element, 2D element and 3D element are also different in terms of structural deformation. Stress is created in physical structure by applying loads and the technique is continued gradually by a prescribed amount (nonzero displacements) by considering the boundary conditions which affects the physical structure of the complex body.

The most common forms of loading include:

- Point loads
- Distributed tractions on surfaces
- Pressure loads on surfaces
- Distributed edge loads and moments on shell edges
- Thermal loads
- Body forces, such as the force of gravity
- In a static stress analysis adequate boundary conditions must be used to prevent the model from moving as a
- Rigid body in any direction; otherwise, unrestrained rigid body motion causes the simulation to stop prematurely.
- The potential rigid body motions depend on the dimensionality of the model.

2.6 Analysis type

The perfect analysis type for simulation can be decided by evaluating the applied type of load on structure, the inclusion of inertial effects and material properties which is the first step in any FEA method. Some of the common analysis types include linear/nonlinear static/dynamic, electrical, multiphysics, buckling, heat transfer, fatigue, and optimization.

Static analysis or quasi-static analysis is done by applying the load slowly on the structure for slower deformation of the body. We may say that the inertia force is negligible in this case which leads no impacts no vibration. Dynamic analysis is nothing but the loads changing per unit time or frequency. For example, you perform a dynamic analysis to simulate the effect of an impact load on a component or the response of a building during an earthquake.

Modal analysis is mostly used to identify the natural frequencies in the examining body structure. It is a powerful tool that allows engineers to design a product to avoid excitation to match the natural frequencies of the structure, thereby eliminating or minimizing excessive vibration.

A nonlinear structural problem is one where the stiffness of the structure changes as it deforms. Most of nonlinear behaviour analysis is observed in all physical structures which has the large deformation problems. Linear analysis is a convenient approximation that is often adequate for design purposes. It is clearly inadequate for many structural simulations including manufacturing processes such as forging or stamping; crash analysis; and analyses of rubber components such as tires or engine mounts.

2.7 Software Tool Could Be Used: - Solidworks And Simulation

Dassault Systems SOLIDWORKS Corp. offers complete 3D software tools that let you create, simulate, publish, and manage your data. SOLIDWORKS products are easy to learn and use and work together to help you design products better, faster, and more cost-effectively.

The SOLIDWORKS focus on ease-of-use allows more engineers, designers and other technology professionals than ever before to take advantage of 3D in bringing their designs to life.

SOLIDWORKS Simulation Standard is an intuitive virtual testing environment for static linear, time-based motion, and high-cycle fatigue simulation. It delivers a concurrent engineering approach, helping you know if your product will perform properly and how long it will be lasting the design phase, SOLIDWORKS Simulation Professional enables you to

Optimize your design, determine product mechanical resistance, product durability, topology,

Natural frequencies, and test heat transfer and buckling instabilities, it can also perform sequential multi-physics simulations.

3. GEOMETRIC DETAILS

3.1 Traditional Design

Traditional design of dental chair geometry is shown in the following fig. 6. We can clearly see the arrangements of folding and base for flexible movement of the load being applied on the chair.

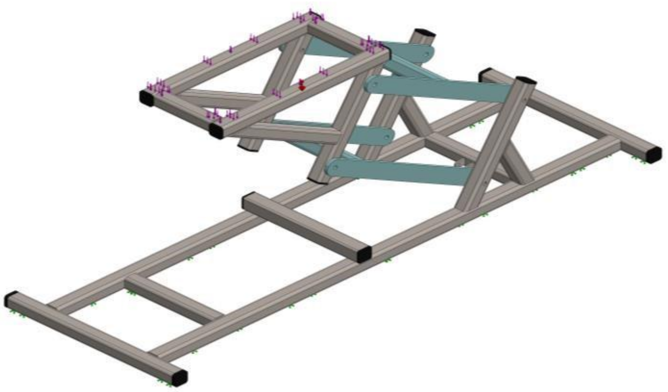

Fig -6: Geometry Details Structure Frame Simplified (Upper and Lower Scissors highlighted in Blue color)

Components used in the dental chair simulation are:

1) Base Frame
2) Upper Scissor
3) Lower Scissor
4) Chair Support
5) Motor
6) Piston
7) Couch

3.2 Modified Design

Dental chair assembly with the scissor length of 300 mm is shown in fig. 7. Stress applied is shown with blue bars on assembly.

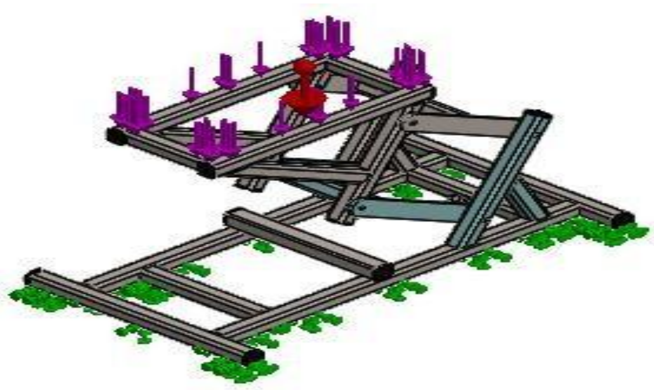

Fig -7: Dental chair assembly: scissor length (300mm)

Table -1: Mesh Information

Mesh type	Solid Mesh
Mesher Used:	Standard mesh
Automatic Transition:	Off
Include Mesh Auto Loops:	Off
Jacobian points for High quality mesh	16 Points
Element Size	0.005 m
Tolerance	0.00025 m
Mesh Quality	High

Table -2: mesh information – details

Total Nodes	1006190
Total Elements	522719
Maximum Aspect Ratio	18.27
% of elements with Aspect Ratio < 3	99.2
Percentage of elements with Aspect Ratio > 10	0.00153
Percentage of distorted elements	0
Time to complete mesh(hh;mm;ss):	00:01:17
Computer name:	DESKTOP

The finite element properties used in this project is described in the table 1 and table 2 which in turn also shows the mesh formation analysis with the total elements.

3.2.1 Resultant Forces

Stress analysis is an important part of the FEA which examines the problem body at its extent level. This is done by applying external force on the examining body and the statistics of applied stress on dental is shown in Table 3, 4 and 5. The resulting simulation is shown in figure 8.

Table -3: Reaction forces

Selection set	Units	Sum X	Sum Y	Sum Z	Resultant
Entire Model	N	0.00 0946 283	2,394 .6 8	9.0 658 7e-05	2 , 39 4.68

Table -4: Study Results

Name	Type	Min	Max
Stress1	VON: von Mises Stress	2.673e-05N/mm^2 (MPa) Node: 172327	1.482e+02 N/mm^2 (MPa) Node: 382682

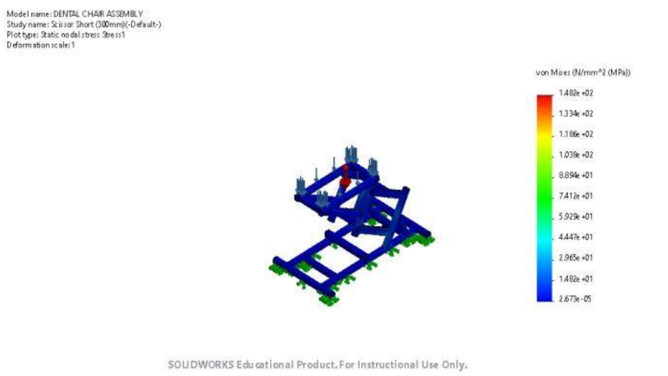

Fig -8: Dental Chair (Scissor Short) - Stress-Stress1

Table -5: Strain applied in dental chair problem data

Name	Type	Min	Max
Strain1	ESTRN: Equivalent Strain	2.688e-10 Element: 34482	4.673e-04 Element: 187995

The Displacements is used to calculate strain and stress in each element at the nodes surrounding the element. The displacement parameters are shown in Table 6 and respected simulation is presented in fig. 9 and 10.

Table -6: Displacement parameters

Name	Type	Min	Max
Displacement1	URES: Resultant Displacement	0.000mm Node: 9344	0.522mm Node: 707483

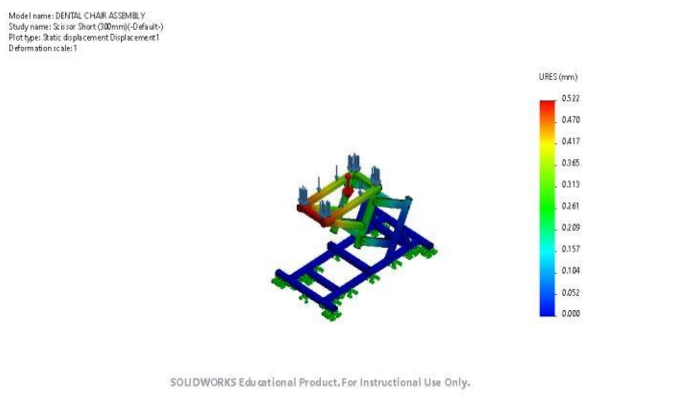

Fig -9: Dental Chair (Scissor Short) - Displacement-Displacement1

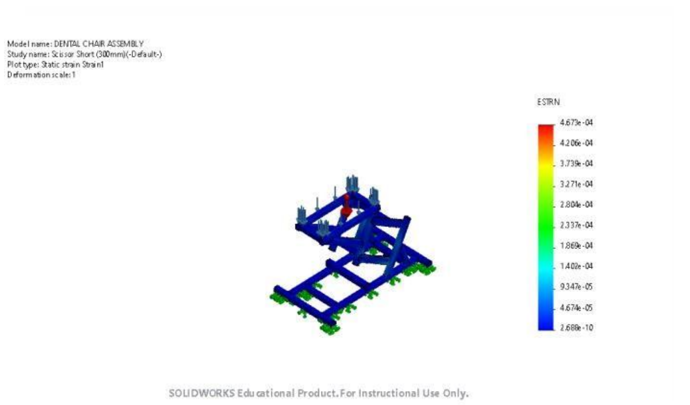

Fig-10: Dental chair (Scissor Short) Strain-strain1

To evaluate whether the given structure is fit for the intended purpose we have to study FE analysis results with comparison, this is called allowables. The ratio of (Allowable/FEA Result) is called Factor of Safety (FoS). FoS parameter is defined in Table 7 and simulation is given in fig. 11.

Table -7: Factor of safety parameter

Name	Type	Min	Max
Factor of Safety1	Automatic	1.970e+00 Node: 382682	1.092e+07 Node: 172327

Fig -11: Dental Chair (Scissor Short) – Factor of safety – Factor of Safety1

4. COMPARE RESULT

The fig. 12, 13 and 14 compares the finite element analysis with different parameters of displacement, stress and FoS by simulation. The model validation obtains by comparing the stress analysis by FEM. The maximum stress value is colored in red area in the FEM simulation.

4.1 Compare Result Displacement, Stress, FoS: Case 1

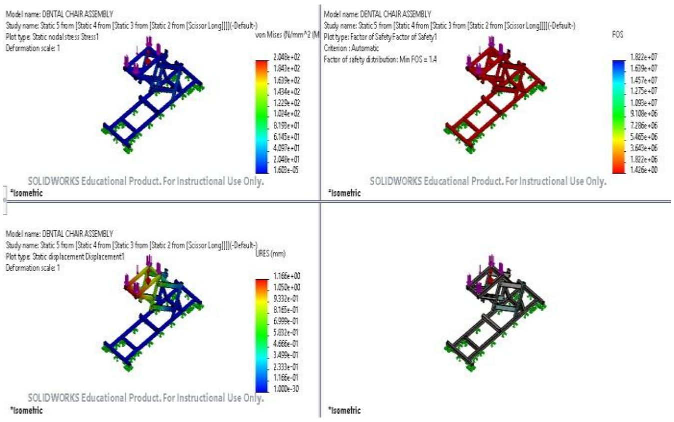

Fig -12: Compare Result for Scissor Link Length (500 mm)

4.2 Compare Result Displacement, Stress, FoS: Case 2

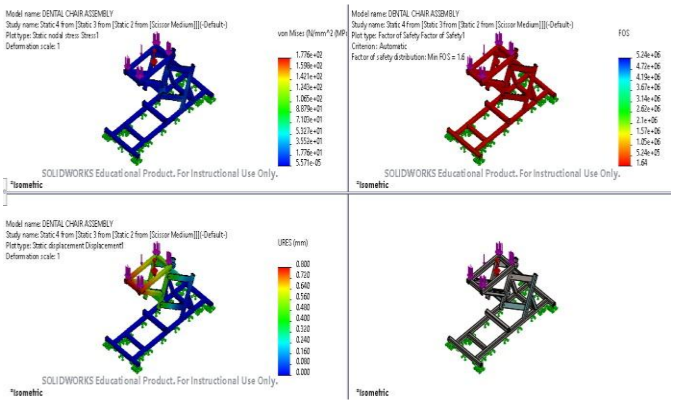

Fig -13: Compare Result for Scissor Link Length (400 mm)

4.3 Compare Result Displacement, Stress, FoS: Case 3

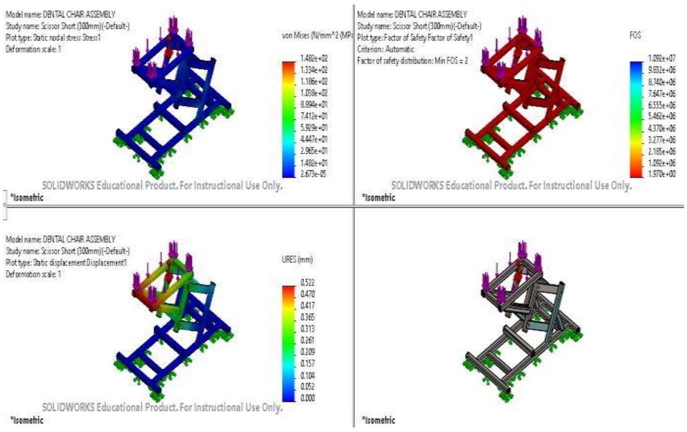

Fig -14: Compare Result for Scissor Link Length (300 mm)

5. CONCLUSIONS

This project work gives an idea about structural strength of dental chair lifting structure, based on the above study, structural analysis results for Case-1: Scissor Link Long (500 mm), Case-2: Scissor Link Medium (400 mm), and Case-3: Scissor Link Short (300 mm) were obtained for comparative study, to conclude the safe and optimal design.

In Case-1: Scissor Link Long (500 mm), the obtained maximum stress was 204.8 MPa, with maximum deflection of 1.166 mm and Factor of safety 1.4 with assembly weight of 102.19 kg. For, Case-2: Scissor Link Medium (400 mm), the obtained maximum stress was 177.6 MPa, with maximum deflection of 0.799 mm and Factor of safety 1.6 with assembly weight of 98.93 kg. For, Case-3: Scissor Link Short (300 mm), the obtained maximum stress was 148.2 MPa, with maximum deflection of 0.522 mm and Factor of safety 2 with assembly weight of 95.67 kg.

The comparative study results show that there is,

- 3.2 % of weight optimization in Case-2 than Case-1, that is directly linked to the material cost.
- 17.65 % greater factor of safety in Case-2 than Case1, safer design.
- 6.37 % of weight optimization in Case-3 than Case1, that is directly linked to the material cost.
- 30% greater Factor of safety in Case-3 than Case-1, safer design.

The optimized design for the dental chair lifting structure is observed in Case-3, with Scissor link length as 300 mm and achieving lowest deflection and Von. Mises stress with highest factor of safety. Hence, the short scissor length up to 300mm gives the safe and optimal design for the Dental Chair.

6. FUTURE SCOPE

There future scope of research work may be as follows:

We can include the mechanism to adjust the inclination of head rest and foot rest pad of the dental chair.

Further we can do analysis for linear dynamics to check the real-world scenario.

REFERENCES

[1] Batra P, Saini P, Yadav V. Oral health concerns in India. J Oral Biol Craniofac Res. 2020 Apr-Jun; 10 (2):171-174.

[2] W. Zhang and S. Cui, "The research and design of orthodontic platform framework," in 4th International Conference on Mechatronics, Materials, Chemistry and Computer Engineering (ICMMCCE 2015), 2015, pp. 700–703, doi: 10.2991/icmmcce-15.2015.141.

[3] P. W. Kang, J. H. Kim, H. Kim, and J. R. Morrison, "Improving dental service via communication during treatment," 8th Int. Conf. Serv. Syst. Serv. Manag. - Proc. ICSSSM'11, pp. 3–8, 2011, doi: 10.1109/ICSSSM.2011.5959428.

[4] S. P. Chaphalkar, Subhash. N. Khetre, Arun M. Meshram, (2015), Modal analysis of cantilever beam Structure Using Finite Element analysis and Experimental Analysis, American Journal of Engineering Research, Volume 4, Issue-10, pp-178-185.

[5] Agashi, Andre & Daywin, Frans & Gozali, Lina & Adianto, & Purna Irawan, Agustinus. (2022). THE DESIGNING OF SWIVEL CHAIR FOR BENDING MACHINE WORK STATION AT PT GRAHA ADIKARYA LOGAM.

[6] Bham F, Perrie H, Scribante J, Lee CA. Paediatric dental chair sedation: An audit of current practice in Gauteng, South Africa. S Afr Med J. 2015 Jun;105(6):461-4.

[7] Bijalwan, Ashutosh & Misra, Anadi. (2016). Design and Structural Analysis of Flexible Wearable Chair Using Finite Element Method. Open Journal of Applied Sciences. 06. 465-477.