



# Finite Element Stress Analysis of Airplane Seat

Serhat Erden<sup>1</sup> , Paşa Yayla<sup>1\*</sup> 

<sup>1</sup>Marmara University, Engineering Faculty, Department of Mechanical Engineering, İstanbul, Turkey

## Abstract

Finite element method (FEM) is frequently used in the seat industry, as well as in the aircraft seat industry, which is a sub-branch of it, especially in the last 10-15 years. Developments in finite element (FE) analysis have enabled safer and cheaper designs to be created in the seat industry. The accuracy of the finite element analysis performed while using this method is extremely important. For this reason, in creating the finite element model, some important parameters must be selected and processed correctly for the model to give the correct result. These parameters can be listed as element size, time scale, analysis type, and material model. The verification of the Finite element analysis (FEA) results is usually done using experimental methods. It is known that in the finite element analysis results almost equivalent to experimental results are obtained when the aforementioned parameters are modeled correctly. This study aims to perform static stress analysis and topology optimization of an airplane seat using the FEM. The static stresses and displacements created at the seat are calculated under simulated loading conditions. Thanks to the topology optimization study, the weight of the airplane seat is minimized by a 30% without sacrificing seat safety. A comparison of static stresses obtained from the FE and analytical models indicates a reasonable correlation, demonstrating confidence in our FE analysis.

**Keywords:** Finite element analysis, Aircraft seat, Topology optimization

## 1. INTRODUCTION

The plane seat is the seat where passengers sit during the traveling on the flight. Aircraft seats are generally positioned as row to row on the seat tracks in the aircraft. For airplane seats, there are some basic features. These features can be classified into two categories: Providing protection and enable to the seat. There are sub-categories of these two main features. Under the feature of enabling to seat, there are two sub-categories: Longtime seating under which consists of comfort and orthopedy and providing entertainment which can be provided by LCD monitors. Sub-categories of other main features are safety during taxi, take-off and landing and fireproof of seat material, and suitability for emergency exit.

Some types of aircraft seats currently in use are as follows; economy class, premium economy class, business class, and first-class aircraft seats [1]. These listed seats have some features that vary depending on the amount of basic fees, such as seat price and ticket price. The first of these features is the distance between two seats [2, 3]. In addition to these, some features such as monitor size, seat cushion quality, and seat recline size vary in these seats [4]. Caputo and his co-workers have developed hybrid analysis methodologies combining multibody modeling with traditional finite ele-

ment modeling for aircraft seat certification [5]. Dhole and his co-workers [6] and Bhonge and Lankarani [7] conducted a FE analyses in product the development process of an aircraft seat. There are also a number of regulatory bodies detailing the technical regulations of an airplane seat design, such as the European Union Aviation Safety Agency ([www.easa.europa.eu/](http://www.easa.europa.eu/)) and Federal Aviation Administration ([www.faa.gov/](http://www.faa.gov/)).

In this paper, an economy class seat is chosen to perform static analysis and topology optimization. Because economy class seats are the most commonly used in the industrial sense and have the most share in the market. In this context, firstly, the model of the selected seat, which is required for static analysis, is created, static stress analysis is performed and then topology optimization is conducted for this seat.

## 2. FINITE ELEMENT METHODOLOGY

The computer-aided drawing (CAD) model of the economy class seat used in this study was provided by TSI Aviation Seats Company. The FEM static analysis is conducted on this CAD model.

The basic logic in the FEM is to simplify a complex problem and solve it. In this method, the solution region is divided into several, simple, small, interconnected, finite element

\* Corresponding author  
Email: [pasaya.yayla@marmara.edu.tr](mailto:pasaya.yayla@marmara.edu.tr)



sub-regions. In short, the solution to the problem, which is divided into pieces connected by a large number of knots, can be made easily.

For example, the application of the FEM in structural analysis is as follows [8];

- The structure is divided into a number of elements (with elements containing node points).
- The behavior of physical sizes is defined for each element.
- Elements are connected through node points and an approximate system of equations is created for the whole structure.
- System equations are solved for unknown values in node points (for example, displacement).
- The desired values of the selected elements are calculated (for example, stresses).

While performing the FE analysis, the system is divided into small parts, which are called mesh, and meshes are connected to each other at their vertices, which are called node. The number of meshes can change according to the model and also the expected accuracy of the analysis.

Different types of meshes are used when using finite element models. Some of them can be listed as solid mesh and shell mesh. In this study, solid mesh and shell mesh types are used. When choosing these mesh types, the dimensions of the part to be meshed are taken into consideration. If the part thickness is low and the difference between thickness and other dimensions is high, shell-type mesh is used. Shell-type mesh accepts the thickness of the material as if it were absent and only provides planar mesh. However, the part thickness is defined in the program to construct the model correctly. When this way is defined, the parts that use shell-type mesh are displayed in 2D on the mesh screen and 3-dimensional on the geometry screen.

Another mesh type, solid mesh, is used to define mesh by dividing the part directly into small pieces. In this way, the part to be meshed is divided into small elements in 3 dimensions. There are some points to be considered when choosing parts to use this mesh type. One of them is that there are not proportionally large differences between the dimensions of the part that solid mesh will be used.

In finite element analysis, another factor to be considered about meshing is the average element quality. This value can be observed in the program where finite element analysis is done after the mesh operation is finished. Having an average mesh level of 80% and above is one of the conditions required for the finite element analysis to be converted to the correct result. In models where the average element quality is lower than 80%, it may be necessary to try to increase the average element quality by applying mesh refinement processes to increase the accuracy of the results. The average element quality of the mesh used in this study is around 87%.

This ratio is sufficient to converge to the correct results.

One of the most important factors to consider when obtaining a solution using the FE analysis method is the mesh size. The mesh size used is important so that when the model is analyzed, it converges to the correct result. As the mesh size decreases, the number of elements used will increase as the model is divided into smaller pieces. When the number of mesh is increased, the accuracy of the analysis results also increases. However, this situation has an increasing effect on solution time and increasing solution time causes an increase in costs. So, there is an engineering optimization case between the number of mesh and solution time.

The mesh size used for the solution in this study is around 3-4 mm on average, depending on the parts in the model. In addition, the total number of mesh used in the model analyzed in this study is 333185.

### 3. STATIC ANALYSIS

Static analysis is a form of analysis performed by subjugating a previously created CAD data with geometry cleaning, meshing, modeling, and solution phases with the help of a FE program, by applying static load values to the created model and entering the necessary boundary conditions.

For this type of analysis, the most important idea is based on the assumption that time does not play an important role in the analysis and its influence on the results can be ignored. In this type of analysis, classical FE logic is used, the package program converges to the solution in line with the entered boundary conditions and the created model and finally gives the user stress and deformation values with colorful geometric graphics.

There are two main types of static analysis, namely linear static analysis, and non-linear static analysis. According to linear static analysis, there are some assumptions,

- Linear geometry
- Linear material
- No contact
- No internal effects
- No vibrations

If there is no linear relationship between the forces and deformations applied in a system, this system is called a non-linear system. Analysis of such systems is called non-linear analysis. There are some reasons for this nonlinearity. These can be listed as geometric nonlinearity, material nonlinearity, and contact effects [9].

As the linearity in the analysis has deteriorated, as in this study, a non-linear analysis was chosen as the analysis type. This is mainly due to the static friction contacts, non-linearity of the material, pre-tension application in the established model, etc. Thanks to non-linear static analysis, one can get not only the information about our targeted seat but also information about the condition of the seat under different

loading conditions. This gives us information about whether a CAD model created has the desired properties before the prototype production and whether it can withstand loading in the conditions in which it will be used.

#### 4. TOPOLOGY OPTIMIZATION

Topology optimization is a tool that allows us to reach the optimum geometry for the model desired to be created according to the determined boundary conditions and the specified purpose function. Thanks to the topology optimization, the designed models can save material and remain at a minimum weight [10].

In this study, one of the most important issue is the weight of the seat because in aviation industry, weight is an unwanted feature. Therefore, for our case we try to decrease weight of the airplane seat using the topology optimization tool of Ansys Workbench. Thanks to this tool we can optimize the geometry of the seat parts and this allows us to design a lighter seat with enough strength.

Because of the topology optimization performed in the analysis programs, the unnecessary part sections of the draft model are shown in blue, which indicates the lower stress value. These unnecessary sections are then removed by the designer and a new model is created. When the new model is created, it is possible to comment on the changes between the draft model and the new model, such as volume, weight and amount of material. In addition, it is possible to make an approximate estimate of the change only through the figure seen in the analysis program.

#### 5. MATERIALS SELECTION

Material selection is one of the most important steps in the design process. Because the strength, cost, weight and many other features of the system to be designed will vary depending on the choice of the material. To choose the right material, it is necessary to know the requirements that the system must fulfill.

In this study, Al7075 T651 aluminum alloy is selected for the seat base and spreader of the seat and Al6082 T651 aluminum alloy is selected for the seat frame.

High strength aluminum alloys give lightweight solutions to the designer, as its density is low. For this reason, it is a frequently used material in the aviation industry. Additionally, it has been predicted that aluminum meets the required strength conditions in the material researches. Besides, the aluminum alloy used in that study was determined as the optimum material for the model used in this paper by TSI Aviation Seats.

In addition, SHE PC ABS T85 polymer material is used for the plastic parts. In the FE model, most of the parts are metal but some of the parts are plastic. These plastic parts do not require load-carrying such as seat pan and clamp. So mechanical properties of used materials are depicted in Tables 1 to 6 [11].

**Table 1.** Mechanical Properties of Aluminum 6082 T651

Mechanical Properties	Metric
Tensile Strength	290 MPa
Yield Strength	250 MPa
Yield Strain	10.0 %
Modulus of Elasticity	79 MPa
Poisson's Ratio	0,35

**Table 2.** Mechanical Properties of Aluminum 7075 T651

Mechanical Properties	Metric
Tensile Strength	572 MPa
Yield Strength	503 MPa
Yield Strain	9.0 %
Modulus of Elasticity	71 MPa
Poisson's Ratio	0,33

**Table 3.** Mechanical Properties of SHE PC ABS T85

Mechanical Properties	Metric
Tensile Strength	238 MPa
Yield Strength	41 MPa
Yield Strain	4,7 %
Modulus of Elasticity	2300 MPa
Poisson's Ratio	0,35

**Table 4.** Mechanical Properties of Aluminum 6005 T6

Mechanical Properties	Metric
Tensile Strength	270 MPa
Yield Strength	225 MPa
Yield Strain	9.0 %
Modulus of Elasticity	69 MPa
Poisson's Ratio	0,33

**Table 5.** Mechanical Properties of 2024 T3

Mechanical Properties	Metric
Tensile Strength	536 MPa
Yield Strength	370 MPa
Yield Strain	16.7 %
Modulus of Elasticity	72,4 MPa
Poisson's Ratio	0,33

**Table 6.** Mechanical Properties of AISI Type 304 Stainless Steel

Mechanical Properties	Metric
Tensile Strength	505 MPa
Yield Strength	215 MPa
Yield Strain	16 %
Modulus of Elasticity	193 MPa
Poisson's Ratio	0,30

#### 6. BOUNDARY CONDITIONS

Boundary conditions are the forces that are required to solve a model or deformations associated with these forces. Boundary conditions are among the known values when building a model. Thanks to these conditions, the solution tool we use reaches the results by performing the analysis. When creating a FE model, at first we create the mesh model, then we define the contacts, at last, the right boundary conditions are inserted [12].

In our study, we have the boundary conditions, which are the constraints and contacts. At static FE analysis for an airplane seat, we assume that base of the seat is constant this means we define it as a fixed support in the analysis. Airplane seat consists of different components, so we have

to define contacts between them.

In the current study, HyperMesh is used as the pre- and post-processor for meshing the model and displaying the results of the FE analysis. In HyperMesh there are several contact types and we select the contact type, which is appropriate to model. In our model, there are three types of contact to define the model. The first one is the shell to shell contact; this contact is used to define the shell part and shell part contact and gap is defined according to part thickness. The second type of contact is solid to solid contact, this type of contact is used for the solid body contact to the solid body contact. The third type of contact is solid to shell contact type. This type of contact is used for the contact of the solid bodies with shell parts. Also, for this type gap is defined according to part thickness of the shell body [13]

The boundary conditions in the FE model prepared within the scope of this study can be summarized as follows;

1) Thanks to the constraints given to the front and rear legs of the seat, the parts of the seat fixed in the aircraft are introduced to the program.

2) Thanks to the contacts defined between the parts, the interaction that will occur between the seat parts in the real case is defined in the model. The most used of these contact types is the node to the solid contact type. In addition, while defining these and other contact types, the static friction feature is activated and the static friction coefficient is taken as 0.2. The main places where this contact is used are:

- Between spreaders and spreader beams parts
- Between spreader beams and seat legs
- Between spreaders and seat pan clamp rings

3) Parts with fixed bolt connections are modeled as rigid bodies and defined in the program.

## 7. LOADING CONDITIONS

In the current work, since the deformations are not known, the known load values which are the mean passenger weight values for an airplane seat are entered as inputs. These values are determined by the aviation authorities. These loads are considered as the distributed load in the model. This means we assume that the weight of a passenger is distributed on the seat uniformly. This assumption does not cause any errors in the analysis results. In our particular case, a passenger weight is taken as 1000 N, and we totally apply 3000 N load for 3 packs seat, this is mainly due to the fact that according to the specifications the maximum static test load is 1000 N for the airplane seat.

In our model, we applied gravitational acceleration to observe the weight of the seat in the analysis. Additionally, to make a realistic model that is required for the exact result, we used the "add mass" in HyperMesh. Add mass is a technique that is used for the missing parts, and it makes the results more realistic. In our model, there are some missing

parts, such as the seat cushion, backrest, and seat belt [14].

In addition, pre-tension was applied to the between seat beams and spreader to model bolt connections.

## 8. SOLUTIONS PROCEDURE

At that stage, we explain how to solve the analysis and solution procedure. The FEA programs work on the base of an algorithm and according to this algorithm at first, we inset the CAD model from the CAD program to CAE program. Then the geometry cleaning works are done. Some of these works are small hole deleting, mid-surface works, edge deleting, surface deleting and etc. This stage is required for creating a fine mesh because many models geometries are not suitable for the fine meshing.

In continuation, meshing work is done per the part to be meshed. This stage is significantly important for the accuracy of the result. Because improper mesh structure gives incorrect stress and deformation values in analysis results. Also, mesh size is important because mesh size affects the precision and accuracy of the result. The proper mesh size varies according to the model to be analyzed, the computer used, and the sensitivity of the desired solution. In our static analysis, 333185 mesh was used.

Then, boundary conditions and loading conditions are applied to the model as described above.

After these pre-processing works, we run the static analysis in OptiStruct tool, which is mainly used in the topology optimization studies.

After executing the analysis with the FEA program, the intended results are obtained. The analysis time varies depending on the mesh size, the number of mesh, the type of elements used, the type of analysis selected (linear or nonlinear), and the performance of the computer. According to the results of the analysis, we can evaluate whether the CAD model meets the engineering requirements. Also, the total duration may vary slightly depending on the FEA program used. In this study, the execution performed in the HyperWorks program, which is used for static analysis, took approximately 2 hours. HyperWorks is CAE simulation commercial program used in modeling, analysis, visualization, and data management solutions for linear, nonlinear, structural optimization, fluid-structure interaction, and multi-body dynamic problems.

After completing the static analysis, the phase of topology optimization was started. At this stage, a model has been established for the rear leg of the seat and weight optimization work has been done for this part. The topology optimization model was established in the Ansys Workbench program. While the topology optimization model was being established, the force and moment values calculated in the previous static analysis were used. In addition, while this model was established, the mesh study, which is previously used in the static analysis, was used. In order to be able to perform to-

polynomial optimization, the stress values from static analysis must first be known. Therefore, when the established model is run, static analysis has started automatically. Afterward, the topology optimization phase was started. The duration and number of iterations of this stage vary depending on the size of the model, contact structures, and forces. In the topology optimization model established in this study, there are 6189 mesh and 10308 nodes.

## 9. RESULTS

After many attempts the model was installed, the errors in the model were fixed and the results were obtained. The stress and deformation values are obtained for the static analysis and they are presented in Figure 1 to 5.

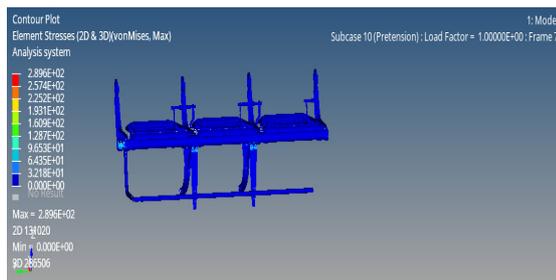


Figure 1. Stress values in front view according to analysis results.

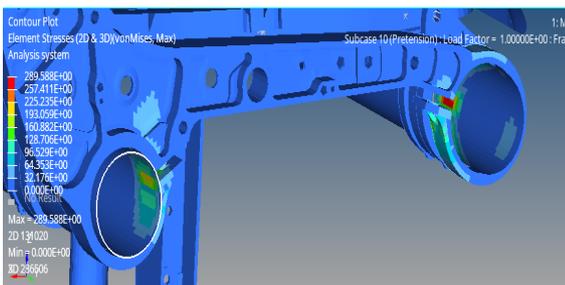


Figure 2. Stress values in side view.

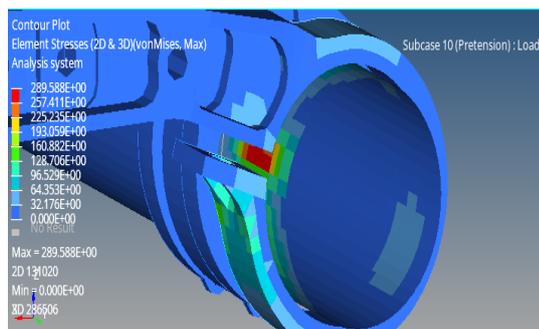


Figure 3. Stress values in detailed side view according to analysis results.

When the stress and deformation values were evaluated, it was seen that there were 30-60 MPa stress values in the seat structure in general. Furthermore, it was found that the maximum von Mises stress was calculated to be about 290 MPa. This stress is at the junction of the spreader. According to the results of the static analysis, these values are predictable and at the expected levels.

When the deformation values were examined, it was observed that the deformations remained at low levels due to the rigid structure of the seat. Accordingly, as a result of the static analysis, the maximum deformation value was observed

to be around 0.5 mm.

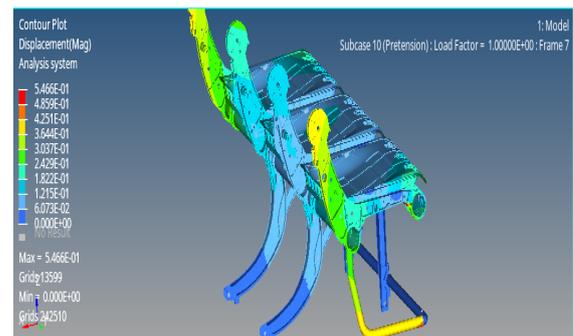


Figure 4. Displacement values in the back-side view according to analysis results.

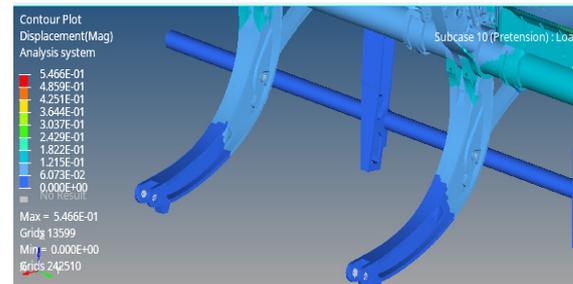


Figure 5. Displacement values in a detailed back view for the rear leg according to analysis results.

It is predictable and expected that the designed seats have such low deformation values in static analysis. Because the designed aircraft seats are subjected not only to static tests but also to 14g and 16g dynamic tests. To pass these tests, the seat structure must be ultra-rigid, stiff, and high strength.

The results obtained in the topology optimization study are presented in Figures 6 to 9.

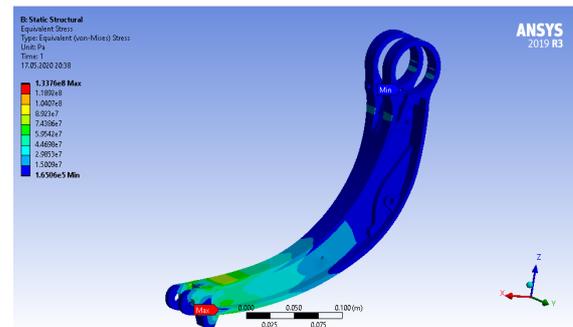


Figure 6. Static condition results for topology optimization of rear-seat leg.

The topology optimization of this study was repeated 3 times with different objective functions. In this context, while performing topology optimization work, the loads in the natural conditions of the part to be optimized were made and the mass reduction was determined as the optimization objective function. As a result of the topology optimizations, designs have been obtained for a potential weight reduction study that can be done for the rear leg of the seat.

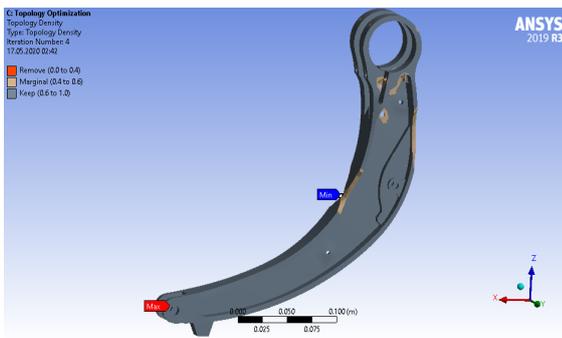


Figure 7. Topology optimization result with 90% to retain of rear-seat leg.

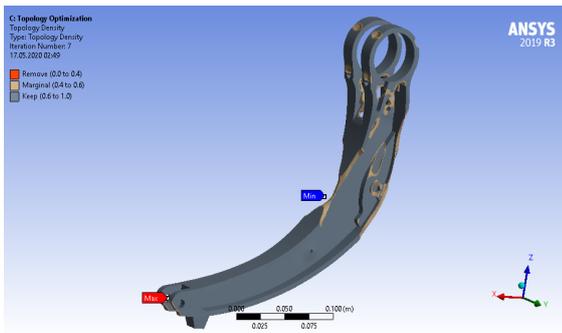


Figure 8. Topology optimization result with 80% to retain of rear-seat leg.

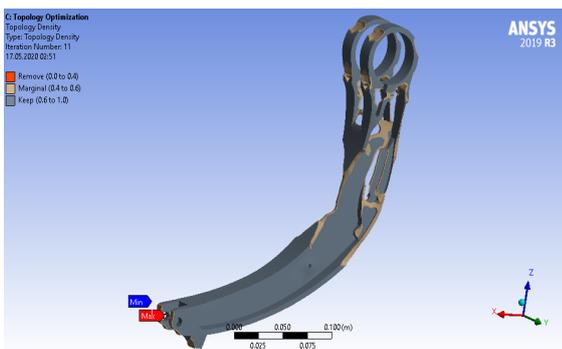


Figure 9. Topology optimization result with 70% to retain of rear-seat leg.

Thanks to the computer-aided analysis, the static analysis was performed and the stresses created in the seat were obtained and then topology optimization was performed. While establishing the topology optimization model, the forces obtained from the full-model analysis were taken as a basis. Besides, topology optimization was made for the behavior of the rear seat leg under harder conditions by adding to the forces obtained from the previous model. The stresses and deformations obtained as a result of these analyzes are at predictable and expected levels.

### 10. VERIFICATION OF RESULTS

The results of the finite element analysis, the correct element type, size, and appropriate solution methods are selected to be highly compatible with the test data. However, for the FE analysis to be validated, it must be tested and verified.

The analysis results made within the scope of this study have not been verified by physical testing. However, the results obtained from the FEM analyzes are compatible with the test results for other seat models. In addition, the stress and deformation values obtained from the FEM analysis are at

the expected levels.

In addition, to check the accuracy of the static analysis results obtained from the current study, some analytical calculations were made. In this context, the stress calculation for the front leg of the seat is as follows;

Firstly, since the seat legs have an almost symmetrical arrangement, the approximate calculation of the load on the front legs of the seat is as follows;

$$Force\ for\ front\ legs \approx \frac{300 \times 9.81}{2} \approx 1471.5\ N$$

Considering that the load applied for 3 seats is applied from three different points;

$$Force\ for\ per\ 3\ point \approx \frac{1471.5}{3} \approx 490.5\ N$$

According to these results, the forces and reaction forces applied to the seat leg are as follows;

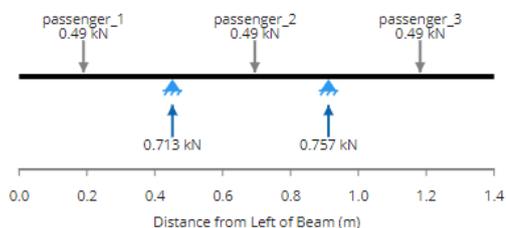


Figure 10. The forces acting on the seat legs and reaction forces.

In addition, the moment acting on the seat legs is as follows;

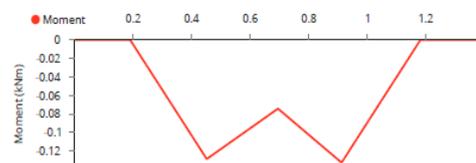


Figure 11. Moment on seat legs.

Due to the reaction forces and moment, the stress on the seat leg was calculated as follows;

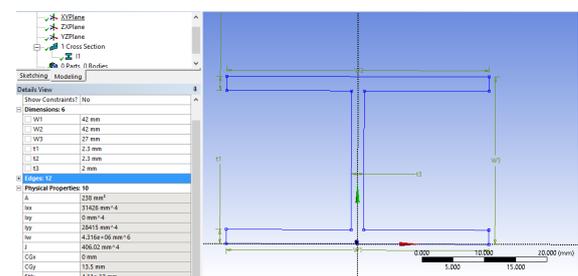


Figure 12. The structure and dimensions of I cross-section profile of the seat leg.

Maximum moment= 132 N.m=132000 N.mm

Maximum force= 757 N

y (the perpendicular distance from the neutral axis) = 13.5 mm

I (Moment of inertia of the seat leg) = 28415 mm<sup>4</sup>

Cross-sectional area of the seat leg = 238 mm<sup>2</sup>

$$\begin{aligned}\sigma_{\text{total}} &\approx \sigma_{\text{compressive}} + \sigma_{\text{bending}} \\ &\approx \frac{F}{A} + \frac{M \cdot y}{I} \\ &\approx \frac{757}{238} + \frac{132000 \cdot 13.5}{28415} \\ &\approx 65.9 \text{ N/mm}^2 \text{ (MPa)}\end{aligned}$$

According to the analytically calculated stresses on the front leg of the seat, the result is approximately 65.9 MPa. In addition, the stresses obtained in the current FEM analysis results are around 30-60 MPa for the leg region of the seat.

When the analytical results are compared with the results obtained in the current analysis, it is understood that the maximum stresses are very close to each other.

## 11. CONCLUSION

In this work, the static stress analysis of an aircraft seat is studied. The work is mainly concentrated on model building, static analysis, and topology optimization studies and their results.

In this study, firstly, the plane seat model to be studied was provided by TSI Aviation Seats Company. In the continuation of the study, model preparation studies have been initiated for the seat to be analyzed statically. Primarily, in order to examine the seat parts with the FEA method, a mesh study was performed. This stage is one of the most important stages for model preparation. Because the mesh quality and the chosen mesh size are very important for the prepared model to converge to a result, more importantly, to converge to a correct result. After the appropriate mesh parameters were set for the analyzes, the boundary conditions and determined loads were arranged for the seat. As a next step, the materials planned to be used in the designed seat were processed into the program, and material was assigned to the seat parts. Then, the methods required for the solution are defined to the model. Finally, the model was run and the results were obtained.

### 11.1. For Static Analysis

The results obtained from the static analysis were evaluated. The calculated stress and strain values remained within the expected limits for static analysis. When these results are evaluated numerically, it is seen that it is generally around 30-60 MPa. In addition, the highest stress expected is around 290 MPa. Considering the yield strength and tensile strength of the material used, it was observed that the stress values were much lower than the strength values. Based on these results, it was concluded that the designed chair was successful under static analysis conditions and could pass the static tests.

### 11.2. For Topology Optimization:

Topology optimization is very important for aircraft seats because every flight-related part weight means cost in the

aviation industry. In this context, each weight reduction work performed, which within the required strength limits of the seat, means a financial gain.

For this purpose, a topology optimization study was conducted for the rear seat leg, which is one of the important parts that determine the seat weight. The topology optimization was repeated with 3 different objective functions. As a result of the topology optimization studies, it has been understood that the strength of the CAD model at hand can remain the same when 30% of the existing material is removed.

The correlation of current FE and analytical results are relatively in good agreement and demonstrated that the FE methodology presented here can be effectively used in the aircraft seat design. Considering these results, it can be said that with the topology optimization study, 30% weight and material savings can be made for the seat back leg.

The CAE design tools used in this study demonstrated the importance of effective usage of these tools in designing critical engineering structure not only save the weight but also reduce the product development time, not to mention much insight into the seat design.

## 12. DISCUSSION

Within the scope of this study, FE stress analyzes were conducted and the results were successfully obtained for an airplane seat. A comparison of the FE analysis results with analytical results indicates a reasonable correlation, strengthening confidence in FEM. However, some analysis stages could be done more comprehensively and additional studies could be made for further analysis.

In this context, the loads determined as the boundary condition in the topology optimization were an acceptance. This acceptance was made by adding some loads in addition to the loads coming from statistical analysis. In the previous stage, before topology optimization, if dynamic analysis was made in addition to static analysis, more accurate boundary conditions could be determined for topology optimization. Topology optimization has also provided approximate results, but as pointed out before, additional work can be done to obtain this result more accurately.

Another issue is about a possible new design that can be made after topology optimization work. In this context, after the topology optimization study is completed, a new CAD model can be created by transferring the obtained results to a CAD program. This means that an engineering-design iteration can be implemented.

It is worth pointing out that, the results obtained from the current FE analysis need to be verified by the physical tests. It is a well-known fact that, although the current study has yielded reliable and accurate results, they should be verified with physical tests before it turns into a final product. In this context, it is more reliable to verify the current FE analyzes results with suitable physical testing methods.

## ACKNOWLEDGEMENTS

The authors would like to thank TSI Aviation Seats Company for their valuable contribution during the preparation of this article.

## REFERENCES

- [1] Sriram, T. C. (2018). Effect of Anthropometric Variability on Middle-Market Aircraft Seating. *International Journal of Aviation, Aeronautics, and Aerospace*, 5(1), 7. 10.15394/ijaaa.2018.1208
- [2] Miller, E. L., Lapp, S. M., Parkinson, M. B. (2019). The effects of seat width, load factor, and passenger demographics on airline passenger accommodation. *Ergonomics*, 62(2), 330-341. 10.1080/00140139.2018.1550209
- [3] Chang, Y. C., Chen, C. F. (2012). Meeting the needs of disabled air passengers: Factors that facilitate help from airlines and airports. *Tourism Management*, 33(3), 529-536. 10.1016/j.tourman.2011.06.002
- [4] Airplane Seat Types, <https://www.aircraftcompare.com/blog/types-of-airplane-seats/> (Access date: 16.01.2020)
- [5] Caputo, F., De Luca, A., Marulo, F., Guida, M., Vitolo, B. (2018). Numerical-experimental assessment of a hybrid FE-MB model of an aircraft seat sled test. *International Journal of Aerospace Engineering*, 2018. 10.1155/2018/8943826
- [6] Dhole, N., Yadav, V., Olivares, G. (2012). Certification by analysis of a typical aircraft seat. *National Institute for Aviation Research*, 1-12.
- [7] Bhonge, P., Lankarani, H. (2008). Finite element modeling strategies for dynamic aircraft seats (No. 2008-01-2272). *SAE Technical Paper*. 10.4271/2008-01-2272
- [8] Hutton, D. V. (2004). *Fundamentals of Finite Element Analysis*, 1st Ed., McGraw Hill Higher Education, USA.
- [9] Dede, G. (2016). *Development of Seat Design and Simulation of Seating Systems' Tests According to European and Us Regulations For Seats*. MS thesis. Çukurova University Institute of Natural and Applied Sciences, (2016).
- [10] Altair University (2014). *Practical Aspects of Structural Optimization, A Study Guide*.
- [11] Callister W. D. and Rethwisch, D. G. (2009). *Materials Science and Engineering*, 8. Edition, John Wiley and Sons.
- [12] Url, < <http://www.value-design-consulting.co.uk/boundary-conditions.html> (Access date:20.01.2020)
- [13] HyperMesh Tutorials, <https://altairhyperworks.in/edu/contest/aoc/2013/tutorials-and-downloads.html#.XioHF8gzblU> (Access date: 20.01.2020)
- [14] Kelleci, Z. E., (2020). Personal Meeting, TSI Aviation Seats Company