MARMARA UNIVERSITY
FACULTY OF ENGINEERING

FINITE ELEMENT ANALYSIS AND TOPOLOGY OPTIMIZATION OF AIRPLANE SEAT

SERHAT ERDEN

GRADUATION PROJECT REPORT
Department of Mechanical Engineering

Supervisor
Prof. Dr. Paşa YAYLA

ISTANBUL, 2020

MARMARA UNIVERSITY
FACULTY OF ENGINEERING

Finite Element Analysis and Topology Optimization of Airplane Seat

by

Serhat ERDEN

27.05.2020, Istanbul

SUBMITTED TO THE DEPARTMENT OF MECHANICAL ENGINEERING IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE

OF

BACHELOR OF SCIENCE

AT

MARMARA UNIVERSITY

Signature of Author.. Serhat ERDEN
Department of Mechanical Engineering

Certified By..Prof. Dr. Paşa YAYLA
Project Supervisor, Department of Mechanical Engineering

Accepted By..Prof. Dr. Bülent EKİCİ
Head of the Department of Mechanical Engineering

ACKNOWLEDGEMENT

First of all, I would like to thank my supervisor Prof. Dr. Paşa YAYLA, and TSI Aviation Seats Company and all workers, especially R&D Manager Onur KESKİN and R&D engineers Zeynep Ebrar TEKİNGÜNDÜZ and Mert YILDIZ, for the valuable guidance and advice on preparing this thesis and giving me moral and material support. Also, I would like to thank my family who gave me strength and morale throughout my thesis.

In this thesis, a literature research was carried out on airplane seat and structural analysis and a preliminary design was created accordingly. Using this design, the necessary model for the structural analysis of the airplane seat was created and possible cases were analyzed also, topology optimization was done for rear leg of this aircraft seat.

May, 2020 **Serhat Erden**

TABLE OF CONTENTS

CHAPTER 4 RESULTS OF THE ANALYSIS

CHAPTER 5: CONCLUSIONS

ABSTRACT

Finite Element Analysis and Topology Optimization of Airplane Seat

Serhat ERDEN

The finite element method (FEM) is a method that 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 analysis have enabled safer and cheaper designs to be created in the seat industry. The aim of this study is to perform static analysis and topology optimization of an airplane seat using the finite element method (FEM). 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 correctly and processed correctly in order for the model to give the correct result. These parameters can be listed as element size, time scale, analysis type, material model. Verification of the 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 above mentioned parameters are modelled correctly.

ÖNSÖZ

Uçak Koltuğunun Sonlu Elemanlar Analizi ve Topolojisi Optimizasyonu

Serhat ERDEN

Sonlu elemanlar yöntemi (FEM), koltuk endüstrisinde, ve bunun bir alt kolu olan uçak koltuğu endüstrisinde, özellikle son 10-15 yılda sıkça kullanılan bir yöntemdir. Sonlu eleman analizindeki gelişmeler koltuk sektöründe daha güvenli ve daha ucuz tasarımların oluşturulmasını sağlamıştır. Bu çalışmanın amacı sonlu elemanlar yöntemi kullanılarak bir uçak koltuğunun statik analizi ve topoloji optimizasyonunu yapmaktır. Bu yöntemi kullanırken yapılan sonlu elemanlar analizinin doğruluğu son derece önemlidir. Bu nedenle, sonlu elemanlar modeli oluşturulurken, modelin doğru sonucu verebilmesi için bazı önemli parametrelerin doğru seçilmesi ve doğru şekilde modele işlenmesi gerekir. Bu parametreler eleman boyutu, zaman ölçeği, analiz tipi, malzeme modeli olarak sıralanabilir. FEA sonuçlarının doğrulanması genellikle deneysel yöntemler kullanılarak yapılır. Sonlu elemanlar analiz sonuçlarında, yukarıda belirtilen parametrelerin doğru bir şekilde modellenmesi durumunda deneysel sonuçlara hemen hemen eşdeğer olduğu bilinmektedir.

ABBREVIATIONS

FEA: Finite Element Analysis

CAD: Computer Aided Design

CAE: Computer Aided Engineering

FEM: Finite Element Method

EASA: European Union Aviation Safety Agency

FAA: Federal Aviation Administration

TtoL: Taxi Take-off Landing

LIST OF FIGURES

PAGE

LIST OF TABLES

PAGE

CHAPTER 1   INTRODUCTION

1.1. Introduction

The plane seat is the seat where passengers traveling on the plane sit for the duration of the flight. Aircraft seats are generally positioned as row to row on the seat tracks in the aircraft. For an airplane seats, there are some basic features. These features can be classified in two categories: Providing protection and enable to seat. There are sub-categories of these two main features. Under the feature of enabling to seat, there are two sub-categories: Long time seating under which consists of comfort and orthopedy and providing entertainment which can be provided by LCD monitors. Sub-categories of other main feature are safety during taxi, take-off and landing (TToL) and fire proof 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. These listed seats have some features that vary depending on the amount of basic fees, such as seat price, ticket price. The first of these features is the distance between two seats. In addition to these, some features such as monitor size, seat cushion quality, seat recline size vary in these seats [1].

In this study, an economy class seat was chosen to perform static analysis and topology optimization. Because economy class seats are the most 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, will be created and then topology optimization will be made for this seat.

Finally, the computer-aided drawing (CAD) model of the economy class seat used in this study was provided by TSI Aviation Seats Company for use in this study.

Figure 1.1. Epianka model economy class seat of TSI Aviation Seat

Figure 1.2. Royalux model business class seat of TSI Aviation Seat

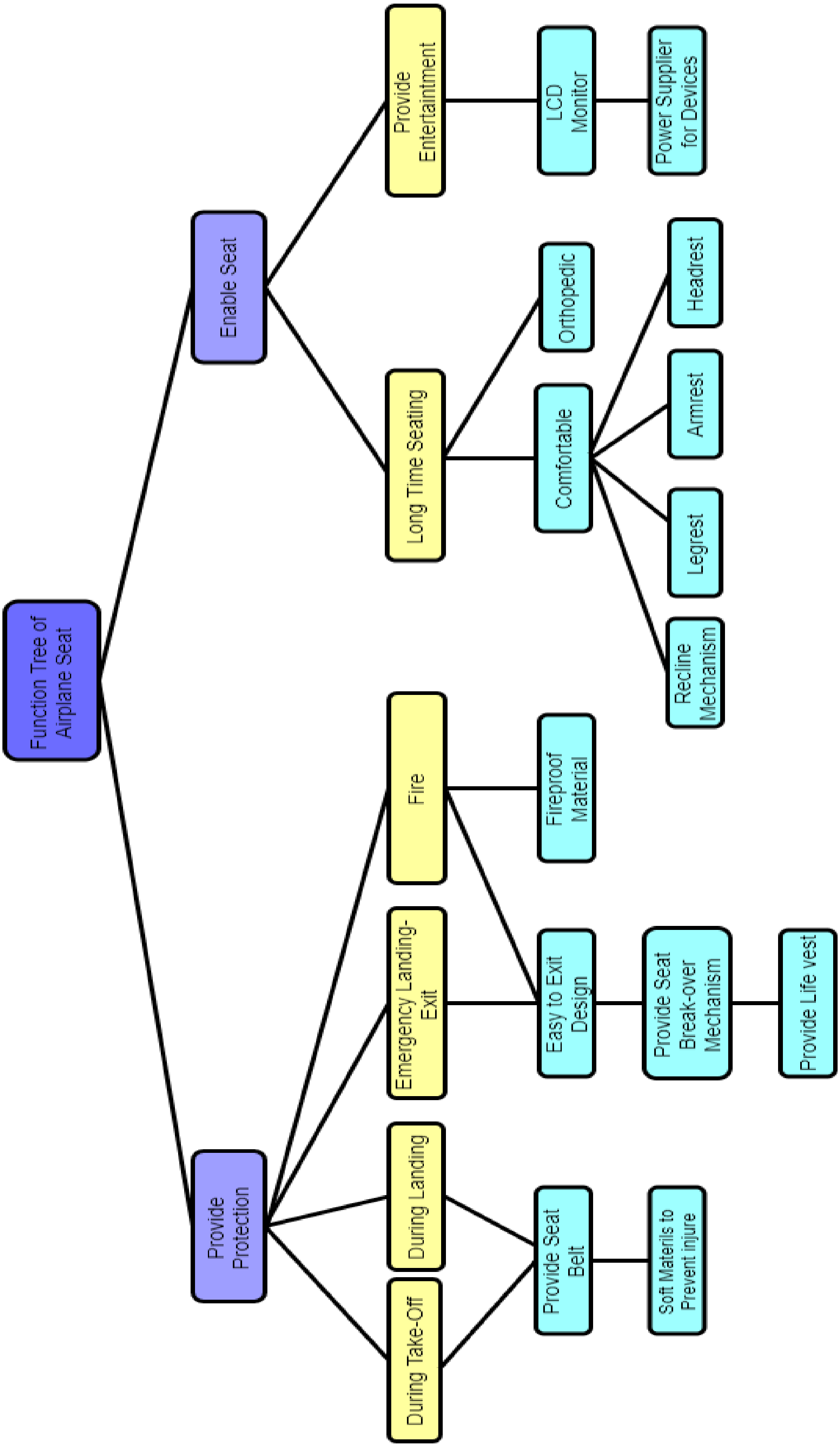

Figure 1.3. Function tree of the airplane seat

1.2. Literature Survey

Some books, articles and aviation standards and rules set by the aviation authorities were used to reference us in the writing process of this thesis. There are some primary authorities publishing specifications which are European Aviation Safety Agency(EASA), Federal Aviation Administration(FAA) and Sivil Havacılık Genel Müdürlüğü(SHGM). Some of these specifications are “Airbus Seat Specification” and “Boing Seat Specification” PDF books. These books are provided by airplane manufacturers to seat manufacturers.

Also, we had to determine which program will be used for the steps of the structural analysis which are the geometry cleaning, meshing, setup of model and solving. While there are lots of program which can perform these steps, but we determined to use Hyperworks tools which are the Hypermesh and Optistruct, because of the adaptability with TSI Aviation Seat Company.

We also decided to use the Ansys program to perform topology optimization.

Book of Fundamentals of Finite Element Analysis published by David V. HUTTON in June 2003 was determined to reference us about structural analysis which is main purpose of the thesis and to enable background about fundamentals of Finite Element Method (FEM). Additionally Practical Aspects of Structural Optimization, A Study Guide published by Altair University in June 2014 used for understanding of topology optimization. Also tutorials of Hyperworks tools are used to learn how to create model and get solving.

The use of finite element analysis has increased in the aviation and armchair industry, as in every sector that has developed in recent years. Accordingly, companies working to meet the requirements of civil and military aviation standards needed a verification phase before going into production after the design phase. While this verification phase was previously provided by testing on the prototype, it was later moved to the digital platform thanks to the developing software and computer hardware.

When previous studies in this area are examined,

In 2013, in the study carried out By Prasannakumar BHONGE, he worked on static and dynamic tests to verify the aircraft seat with the FEM method. At the end of the studies, static analyzes and dynamic analyzes were carried out in computer environment. These analyzes are divided into two as explicit and implicit in the methodological sense. Nevertheless, 14g, 16g dynamic conditions, which are the dynamic test loads determined by the aviation authorities, were analyzed. As a result of the analysis, the strength values of the seat were tested without physical testing [2].

As a similar study, a study was conducted to investigate the use of different materials on the feet of vehicle passenger seats made by Ferruh DÜVENCİ using the finite element method. As a result of the study, another material was used instead of the currently used material, and as a result, although a thicker material requirement was required, the total weight of the seat decreased due to the density of the material. In addition, if the new material is used, the total cost of the seat has decreased in terms of material price. As for this study, studies have been done in computer environment using FEM [3].

In another study in 2007, "In an article published jointly by PJ García NIETOA et al., An analysis of the backrest of an automobile seat was performed. As a result of the many analysis, an optimum geometry for the seat geometry remained within the desired strength limits and a lighter According to the information obtained from the results of the study, the seat frame weight decreased from 9.135 kilograms to 8.298 kilograms, which means that the seat weight decreased by 9.16% as a result of topology optimization [4].

In another study by M. R. SHIVAKUMAR et al. In 2016, the factors affecting the comfort in the driver's seat were handled by using a finite element analysis. The aim of the study is based on the principle of catching maximum comfort without leaving the regulations for the person sitting in the driver's seat. As a result of the study, it was obtained that the maximum passenger comfort was achieved by using 5 kPa foam stiffness and 6 mm * 3 wires. In this study, the principle of analysing the seat structure, the passenger and the surrounding environment with the finite element method was made, and the results were obtained by determining the loads coming to the passenger while driving [5].

In another study by A. SIEFERT et al. In 2007, the effects of the static and dynamic effects of the seat on the sitting comfort were examined and the factors affecting this were tried to be optimized virtually. As a result of the study, the following findings are listed;

Thanks to virtual optimization,

1) Expensive and late design modifications can be avoided,

2) Optimization of seat structure and seat connections can be provided,

3) Since the analyzes are carried out in parallel with the testing phase, when undesirable situations are encountered, a reduction in test costs can be achieved thanks to the possibility of early modification [6].

In another study conducted by Şeyda SAATÇI AYDINER in 2018, she worked on the optimization of the passenger seat leg with the finite element method. The main aim of the study is to evaluate different design products before physical tests and to determine the most economical and final design suitable for the regulations. In this context, different designs for the seat connection legs have been tried using finite element analysis and an optimum design has been tried to be reached. Again, in this study, as a result of the topology optimization for the passenger seat leg of a bus, a 3% decrease in seat weight was achieved [7].

In 2008, in a study conducted by Prasannakumar BHONGE and Hamid LANKARİNİ from Wichita State University, it was studied to verify the compliance of dynamic tests for the aircraft seat with SAE and FAE regulations by using the FEM method and computer verification. The main purpose of the study is to investigate whether it is possible to perform seat verification tests using computer aided models, not physically. As a result of the study, while drawing attention to the parallelism of the FEA results with the test results, some points that this method should be developed in the future are pointed out. These points are as follows [8].

1) Dummy validation

2) Material properties

3) Initial conditions

In a study conducted by Artanç BULGUR from Gebze High Technology Institute in 2006, the simulation of the seat fasteners during the crash test with the first speed conditions and dummy effect was studied. In simulations using Ansys and LS-Dyna programs, the condition of the seat in dynamic test conditions is simulated. The analysis was made linearly due to the lack of computer hardware of the thesis writer and as a result, only the strength values were observed, not the deformations. FEA results were evaluated on the basis of the analysis made on similar seats, since there is no one-to-one comparison chance since the analyzed seat has no test data. As a result of the evaluations, it was stated that the results of the analysis are of logical values and it was stated that computer aided simulations could be used instead of physical tests [9].

In the study of Güney GÖRGÜN, which was made by TSI Aviation Seats in 2019 within the body of Istanbul Technical University, it was studied on the modelling and analysis of aircraft seat backrest using composite material and using finite element analysis method. The main purpose of the study is to lighten the back of the seat made using composite material and to reduce flight costs. As a result of the study, it has been mentioned that, according to the strength values verified using the finite element method, a plane designed with the use of composite materials has a weight reduction of 135-153 kilograms for an airplane with 300 seats. In the same study, it was pointed out that the one-year fuel expense for the 1-kilogram flight weight of the airline companies was around 130-150 $ and the financial importance of the weight reduction study carried out in this study [10].

As a result of the literature study, similar studies previously conducted in the armchair industry were examined using finite element analysis. It has been observed that the analysis and topology optimization studies provided as a result of the examinations gave positive results. In addition to these, ideas about the methodology to be followed in this thesis have been obtained.

1.3. Aim of the Study

Development in FEA methodology helps airplane industry in designing and simulation of seats and other interiors more economically and confidently. Objective of this study is to perform structural analysis and topology optimization for airplane seat by using FEA techniques and is divided in the following four categories.

•Comparison of yield strength of selected material with obtained stresses result of the static analysis.

•To check the result of maximum deflection in the performed structural analysis.

•To determine optimum geometry using topology optimization and reduce the total weight of the aircraft seat with FEA program.

•Checking the seat’s compatibility with aviation standards.

CHAPTER 2 FEM ANALYSIS

2.1. Finite Element Methodology

The Finite Element Method (FEM) was first developed for stress analysis of aircraft bodies, in 1956, and has been used for the solution of applied sciences and engineering problems in the next decade. In the following years, these methods and solution techniques were developed rapidly and today, it has become one of the best methods used for the solution of many engineering problems.

The basic logic in the finite element method is to simplify a complex problem and solve it. In this method, the solution region is divided into several, simple, small, interconnected, finite element sub regions. In short, the solution of the problem, which is divided into pieces connected by a large number of knots, can be made easily.
For example, the application of the finite element method in a structural analysis is as follows:
The structure is broken into pieces. (With elements containing node points)

- The behaviour 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) [11].

While we perform the FE analysis, we divide the system into small parts which called mesh and meshes are connected to each other at their vertices which called node. Number of meshes are changing according to model and also expected accuracy of the analysis.

Different types of mesh structures are used when using finite element models. Some of them can be listed as solid mesh, shell mesh. In this study, solid mesh and shell mesh types were used. When choosing these mesh types, the dimensions of the part to be meshed are taken into consideration. If the piece thickness is low and the difference between thickness and other sizes 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 in order 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 parts 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 mesh 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 important factors to consider when making a solution using the finite element 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 number of mesh is increase, accuracy of the analysis results also increases. However, this situation has an increasing effect on solution time and increasing solution time causes increase in costs. So, there is an engineering optimization case between 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.

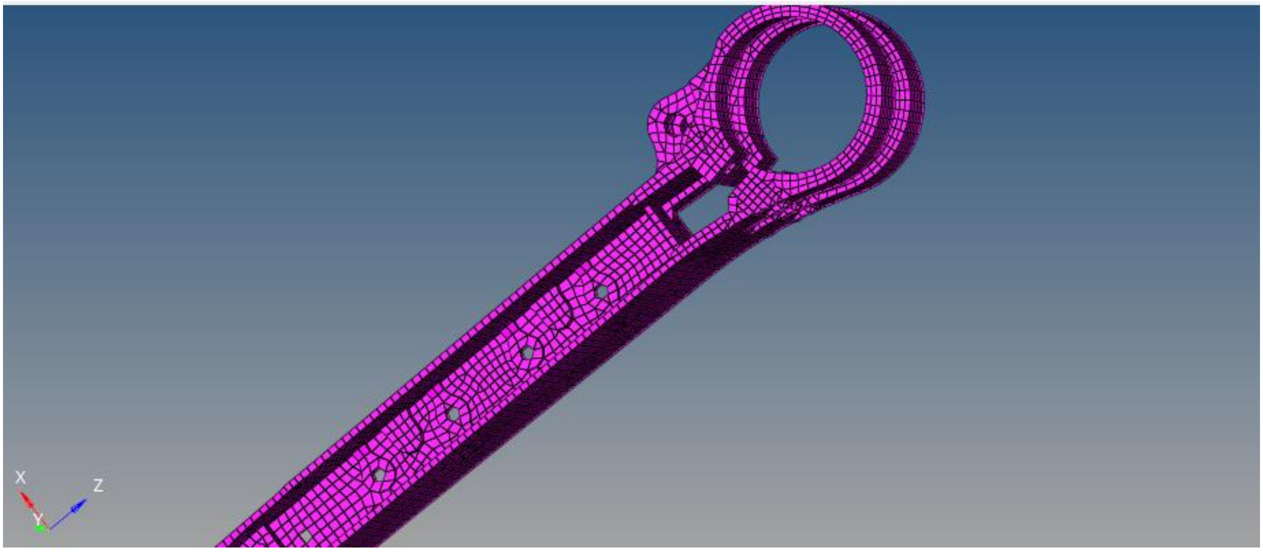

Figure 2.1. Solid mesh example for seat leg part.

2.2. Description of The Program Used in the Study

In this study, the CAD model created in the Catia V5 program by TSI aviation seats was used. In the analysis process, different tool of Hyper works package program were used at static analysis stage, such as hyper mesh and OptiStruct. Hyperworks is a Computer Aided Engineering package program which used for the static, dynamic, impact simulations, topology optimization and also flow simulations. In this context, while Hypermesh is used in geometry clean-up and mesh stages, OptiStruct, which is the static solver of the same package program, was used to solve the created static analysis model. Finally, the results, which were obtained by using the OptiStruct program, used for the topology optimization. In this study, Ansys Workbench FEA program used for the topology optimization. Due to the number of mesh and model size, a powerful computer processor was needed throughout the work.

2.3. Static Analysis

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

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

There are 2 main types of static analysis, one of them is linear static analysis and the other one is non-linear static analysis. According to linear 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 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 [12].

In this study, non-linear analysis was chosen as the analysis type. Because for some reason, the linearity in the analysis has deteriorated. Some of these reasons are as follows: static friction contacts, non-linearity of the material, pre-tension application in the established model, etc. Thanks to non-linear static analysis we can get the information about the our seat. We can get 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.

2.4. 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 in addition, the designed model remains at a minimum weight [13].

In our study one of the most important case is the weight because in aviation industry, weight is unwanted feature. So 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 file and this allows us to design a lighter seat with enough strength.

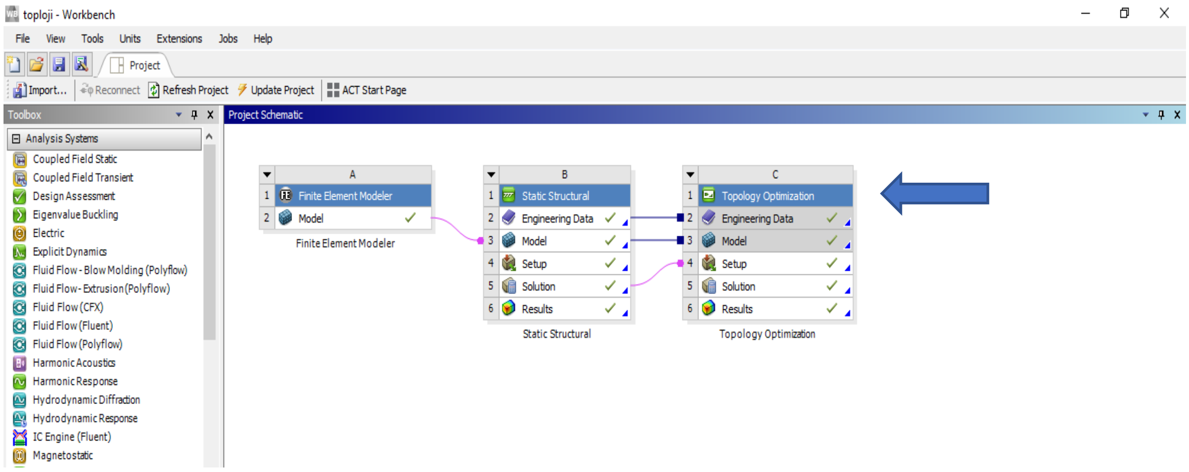

Figure 2.2. Topology optimization setup of Ansys Workbench.

As a result of the topology optimization performed in the analysis programs, the unnecessary parts 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. Since the right to make changes to the draft data for the sake of confidentiality in this study only belongs to TSI Aviation Seats, no changes will be made to the draft CAD data after the topology optimization study. This situation will not prevent the completion of the study as mentioned above.

CHAPTER 3 PROBLEM DEFINITION

3.1. Description of the Problem

The first CAD model created when designing a product is not the first prototype produced. Because prototype costs are high and it is necessary to decrease this as much as possible in order to produce an economical design. To achieve this goal, a previously produced CAD model is analyzed with the help of CAE simulation programs. In addition, weight and material optimizations of the product to be produced can be made using these package programs.

In this study worked on the static analysis and topology optimization for an airplane seat. The main purpose of the study is to verify the CAD model with high accuracy before producing prototypes using FEA programs and to reduce the weight with topology optimization method in order to make the airplane seat to be more competitive and economical design.
During the verification, stress values obtained from the program, the yield strength of the materials and the requirements required by the aviation authorities were taken into consideration.

3.2. Materials Selection

Material selection is one of the important steps for a design. Because the strength, cost, weight and many more features of the system to be designed will vary depending on the choice of the material. In order to choose the right material, it is necessary to know the requirements that the system must fulfill.

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

Aluminum gives 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. In addition, aluminum alloy which used in that study was determined as the optimum material for the model used in this thesis by TSI Aviation Seats.

Also, SHE_PCABS_T85 material used for the plastic parts. In this model, most of the parts are metal but some of the parts are plastic which parts that do not require load carrying such as seat pan and clamp. So mechanical properties of used materials are as follow, [14].

Table 3.1. Mechanical Properties of Aluminum 6082 T651

Mechanical Properties	Metric
Hardness, Vickers	95
Tensile Strength, Ultimate	290 MPa
Tensile Strength, Yield	250 MPa
Elongation at Break	10.0 %

Table 3.2. Mechanical Properties of Aluminum 7075 T651

Mechanical Properties	Metric
Hardness, Vickers	175
Tensile Strength, Ultimate	572 MPa
Tensile Strength, Yield	503 MPa
Elongation at Break	9.0 %

Table 3.3. Mechanical Properties of SHE_PCABS_T85

Mechanical Properties	Metric
Hardness, Vickers	-
Tensile Strength, Ultimate	238 MPa
Tensile Strength, Yield	41 MPa
Elongation at Break	160 %

Table 3.4. Mechanical Properties of Aluminum 6005 T6

Mechanical Properties	Metric
Hardness, Brinell	85 HB
Tensile Strength, Ultimate	270 MPa
Tensile Strength, Yield	225 MPa
Elongation at Break	9.0 %

Table 3.5. Mechanical Properties of AW 2024 T3

Mechanical Properties	Metric
Hardness, Brinell	-
Tensile Strength, Ultimate	536 MPa
Tensile Strength, Yield	370 MPa
Elongation at Break	16.7 %

Table 3.6. Mechanical Properties of AISI Type 304 Stainless Steel

Mechanical Properties	Metric
Hardness, Brinell	123
Tensile Strength, Ultimate	505 MPa
Tensile Strength, Yield	215 MPa
Elongation at Break	70.0 %

Table 3.7. Material Table

Material	**Used Part**
Aluminum 6082 T651	Baggage bar
Aluminum 7075 T651	Seat legs, Spreaders, Leg connecting part
SHE_PCABS_T85	Seat pans, Seat pan clamp rings
Aluminum 6005 T6	Spreader connecting part
AW 2024 T3	Spreader beams
AISI Type 304 Stainless Steel	Bar to leg connector

3.3. Boundary Conditions

Boundary conditions are the forces that 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 the first we create the mesh model, then we define the contacts, in continue, boundary conditions are inserted [15].

At that 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 analysis. Also airplane seat consists of different components so we have to define contacts between them. In Hypermesh there are several contact types and we select 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 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 used for the solid body contact from the solid body contact. The last one is the solid to shell contact type. This type contact is used for contact of the solid bodies with shell parts. Also, for this type gap is defined according to part thickness of the shell body [16]

The boundary conditions in the finite element model prepared within the scope of this study can be examined 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 real case is defined in the model. The most used of these contact types is the node to 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 connection are modeled as rigid body and defined in the program.

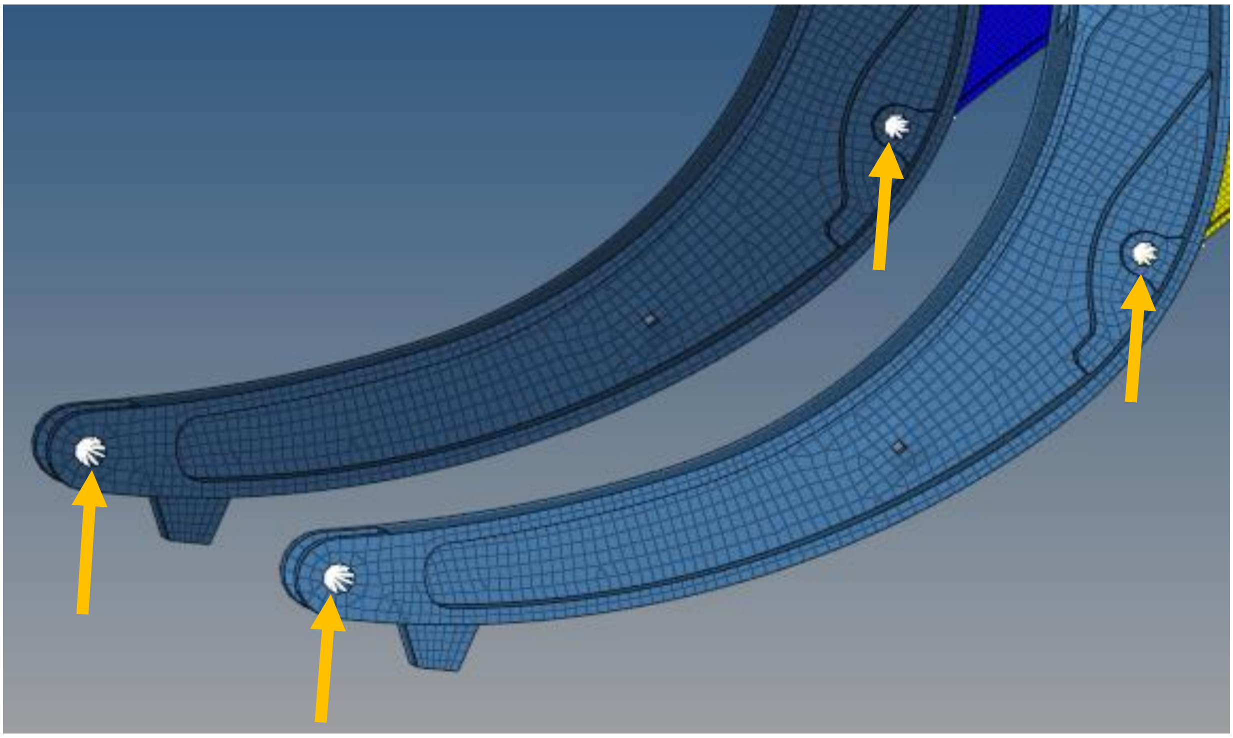

Figure 3.1. Fixed support and bolt connection boundary conditions, defined on the rear legs of the seat.

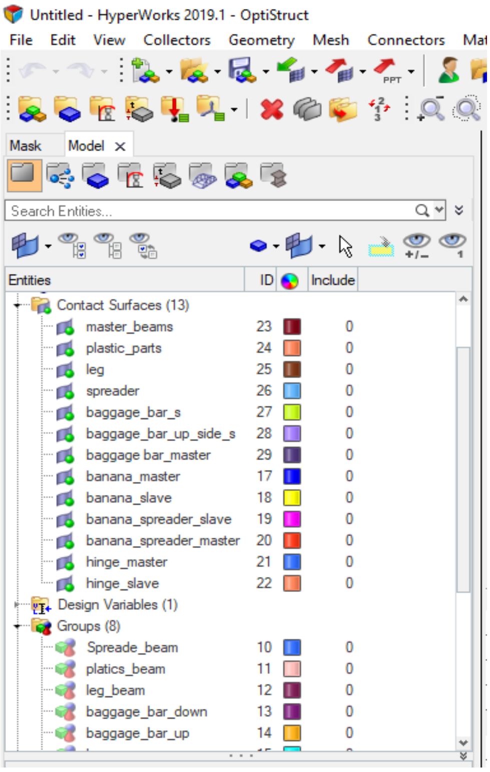

Figure 3.2. Some contacts that used in this study.

3.4. Loading Conditions

At that study we do not know the deformations so will inserted the known load values only which are the mean human weight values for an airplane seat. These values determines by the aviation authorities. These loads gives as the distributed load in the model. This means we assume that weight of a human distributed on the seat equally. This assumption does not cause any errors on the analysis results. In our particular case human weight taken as 100 kg, and we totally applied 300kg weight for 3 packs seat, because according seat specs the maximum static test load is 100kg for the airplane seat.

In our model we applied the gravitational acceleration, to observe the weight of the seat in analysis. Additionally, to make a realistic model which is required for the exact result, we used the “add mass” in Hypermesh. Add mass is a technique that used for the missing parts and it makes the results realistic. In our model there some of the missing parts are like this, seat cushion, backrest, seat belt [17].

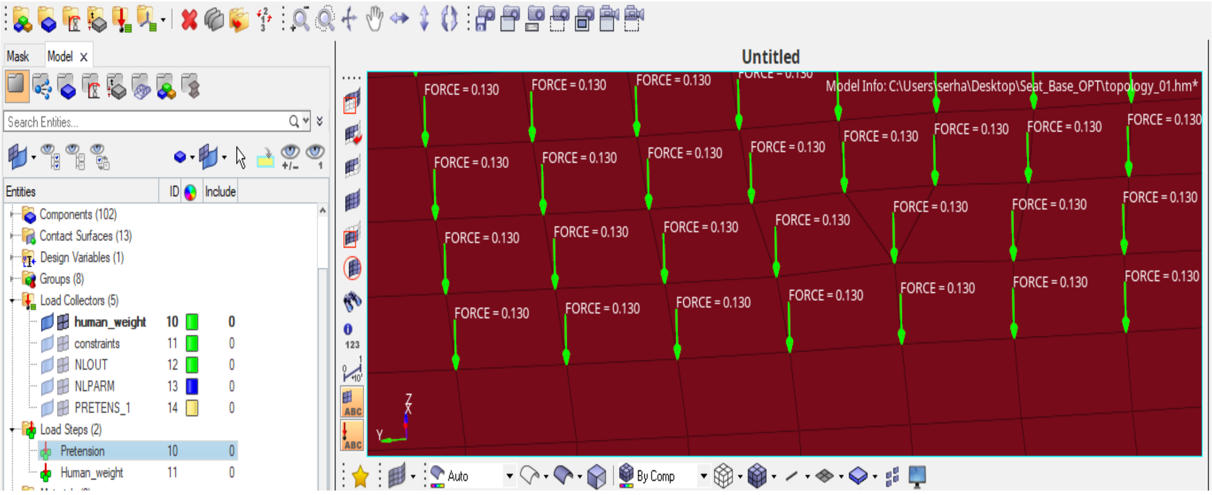

Figure 3.3. Distributed load for each node in Hypermesh.

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

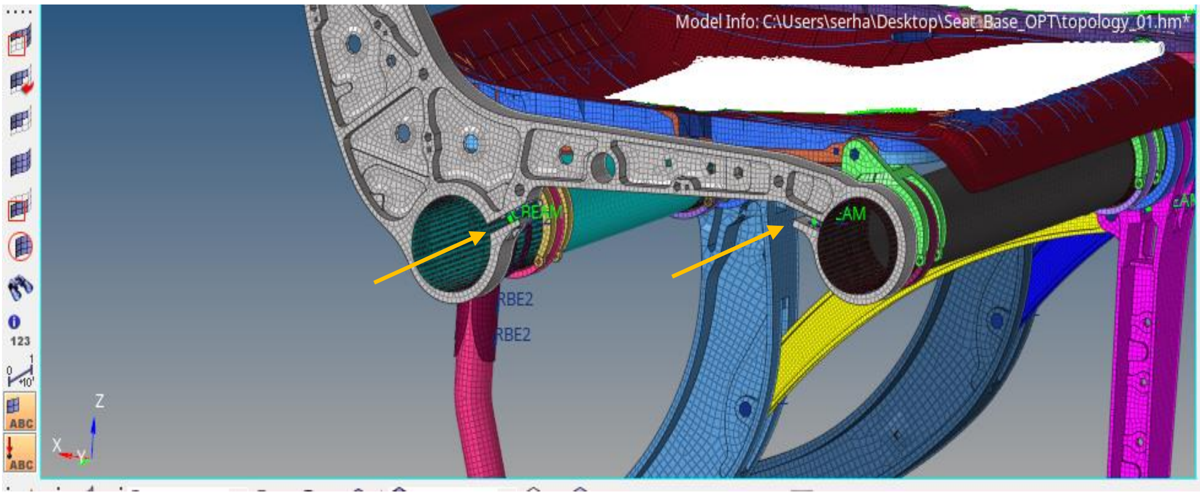

Figure 3.4. Pre-tension between seat beams and spreader in Hypermesh.

3.5. Solutions Procedure

This is the final step of the problem definition chapter. At that step, we explain the how to solve the analysis and solution procedure. FEA programs works an algorithm and according to this algorithm at the first we inset the CAD model from the CAD program to CAE program. Then, geometry cleaning works are done. Some of these works are like this; small hole deleting, mid-surface works, edge deleting, surface deleting and etc. This stage required for the create a fine mesh because in many models geometry is not suitable for the meshing.

In continuation, meshing work is done in accordance with 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 as it 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 this model, which was established for static analysis, 333185 mesh was used.

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

After all these works we run the static analysis in OptiStruct.

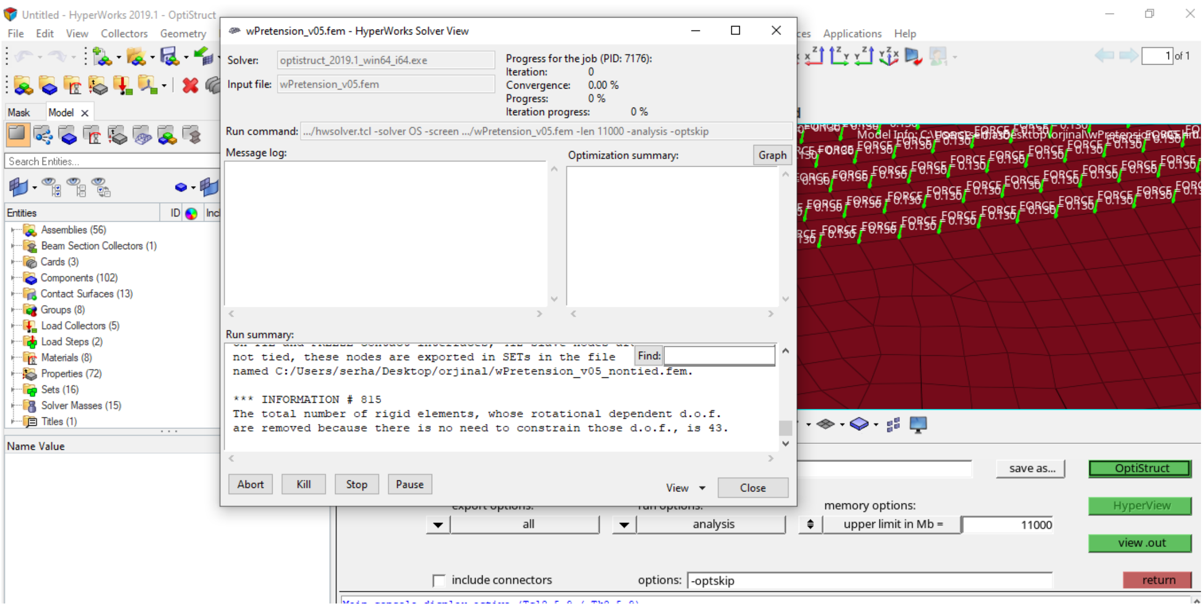

Figure 3.5. Analysis window with OptiStruct solver.

Analysis done by the FEA program and it gives us the results. 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.

The solution time may vary depending on the processing power of the computer and the number of elements in the model installed. Also, the total duration may vary slightly depending on the FEA program used. In this study, the run performed in the Hyperworks program, which used for static analysis, took approximately 2 hours.

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 previously used in static analysis, was used.

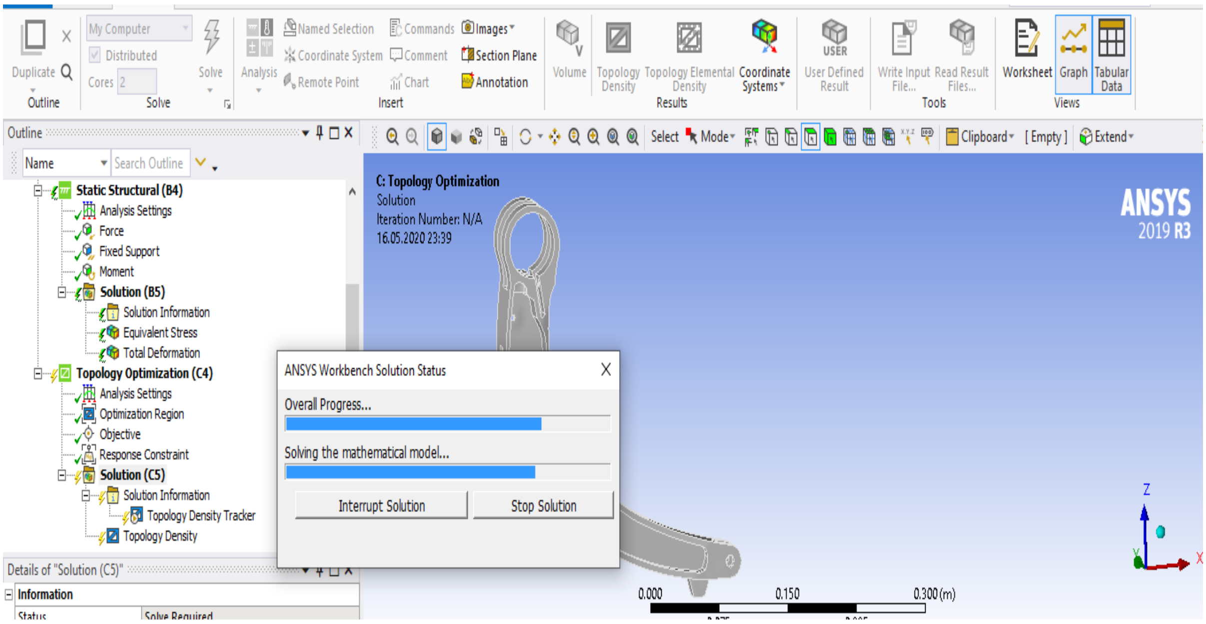

Figure 3.6. Static analysis process of the seat rear leg in Ansys Workbench.

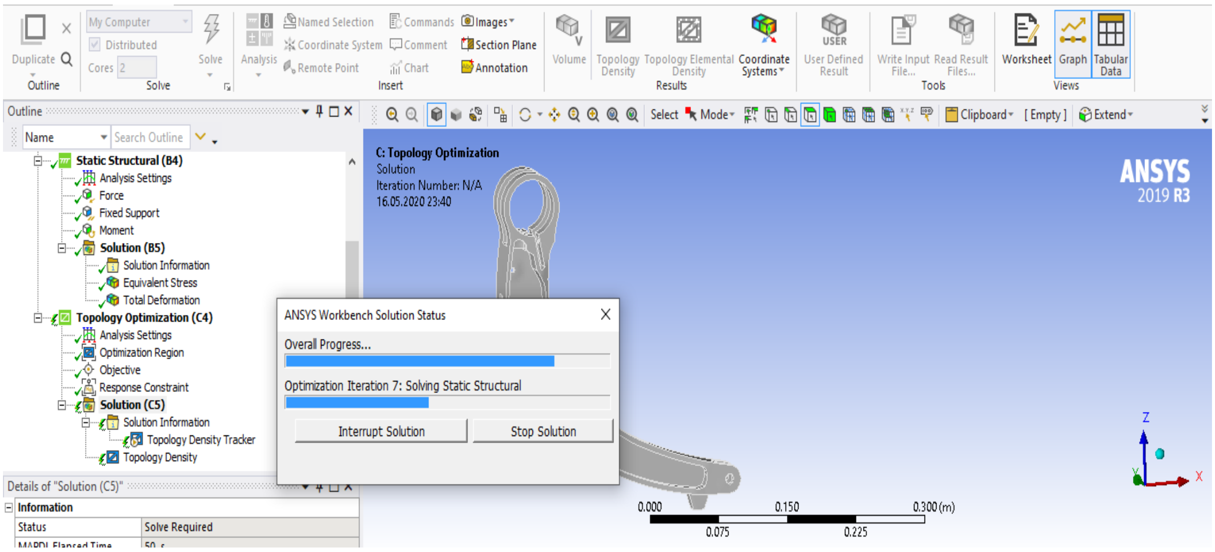

Figure 3.7. Topology optimization process of the seat rear leg in the Ansys Workbench.

As can be seen in the figures in previous page, in order to be able to perform topology optimization, the stress values from static analysis must first be known. Therefore, when the established model is run, static analysis has started automatically. Afterwards, 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.

CHAPTER 4 RESULTS OF THE ANALYSIS

4.1. Analysis of the FEM Results

After many attempts after the model was installed, the errors in the model were fixed and the results were obtained. The stress and deformation values obtained as a result of the static analysis are as shown in the figures below.

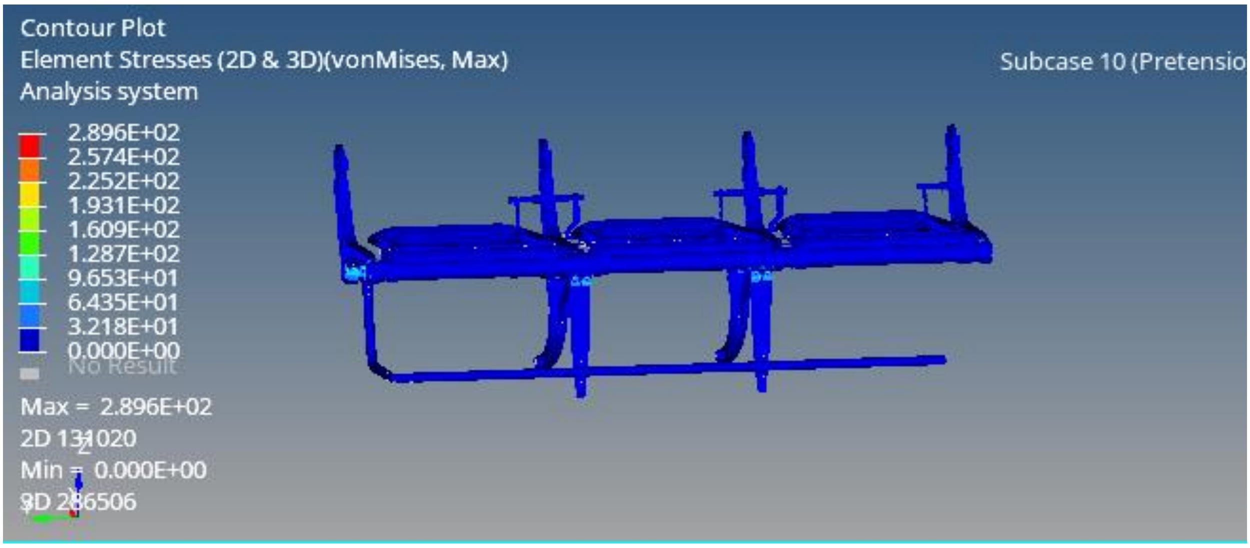

Figure 4.1. Stress values in front view according to analysis results.

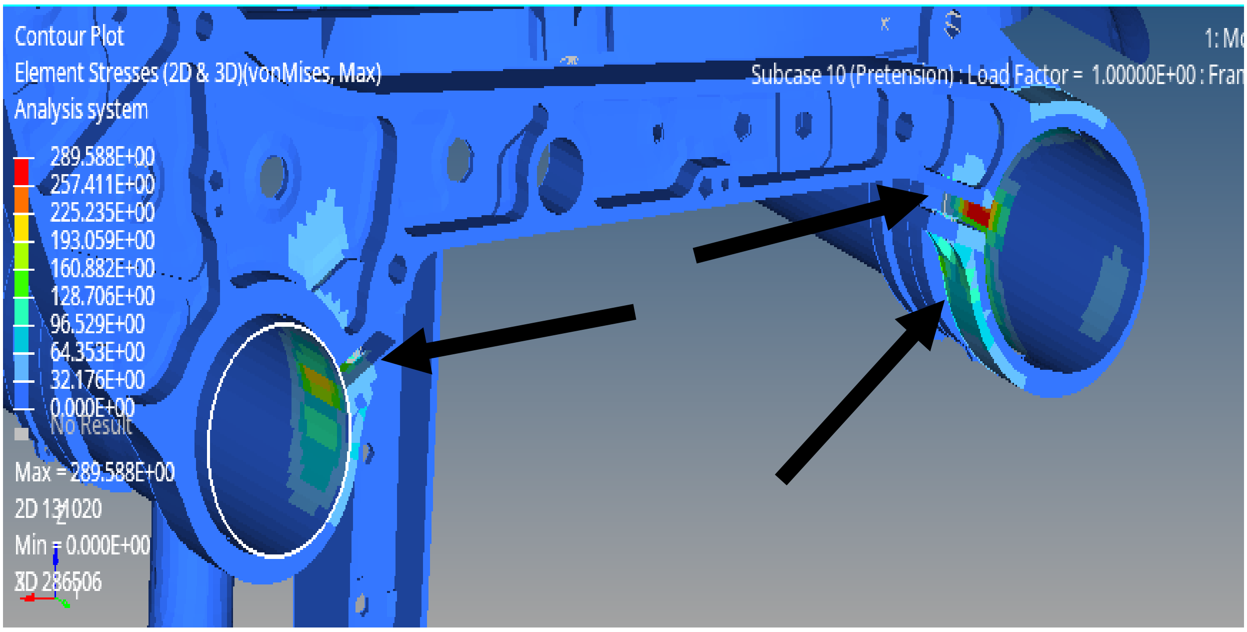

Figure 4.2. Stress values in side view according to analysis results.

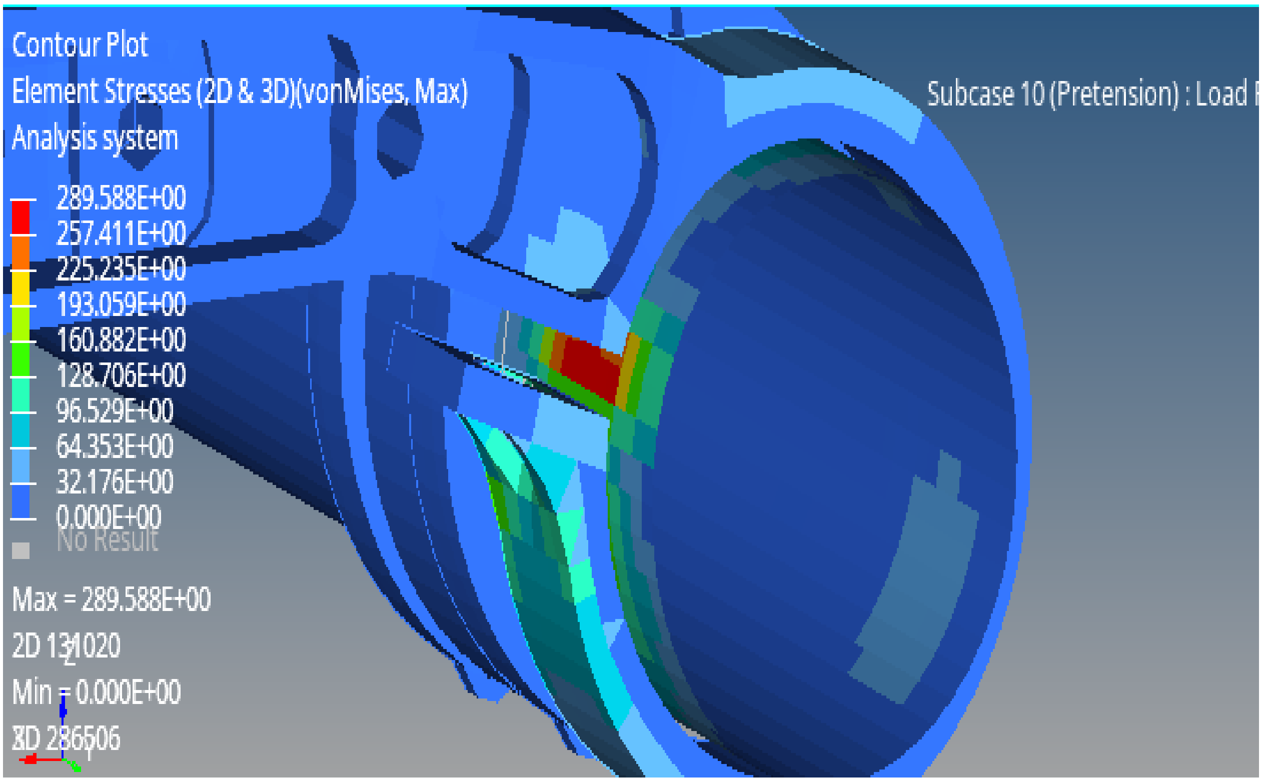

Figure 4.3. Stress values in detailed side view according to analysis results.

When the stress and deformation values were evaluated, it was seen that there was 30-60 MPa stress values in the seat structure in general.

In addition, it was found that the maximum 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.

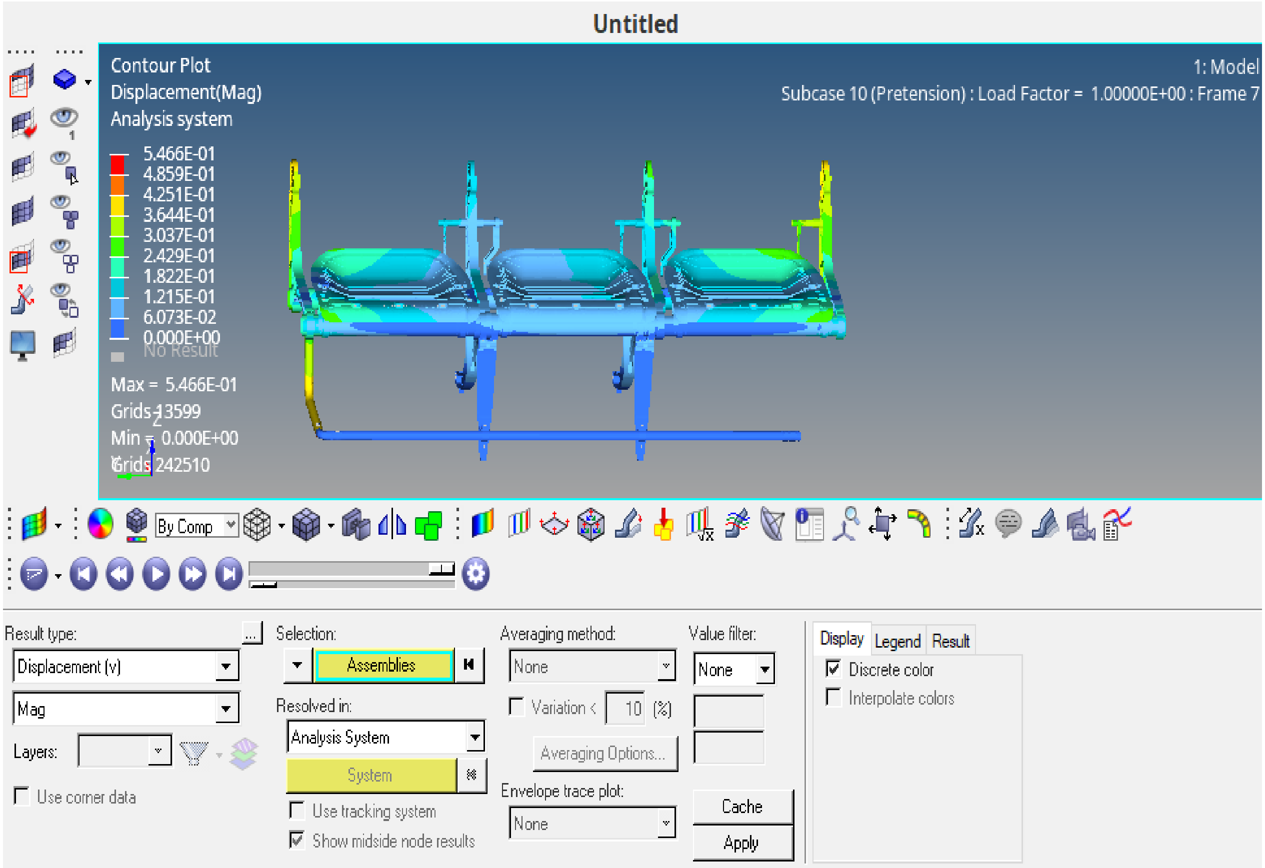

Figure 4.4. Displacement values in front view according to analysis results.

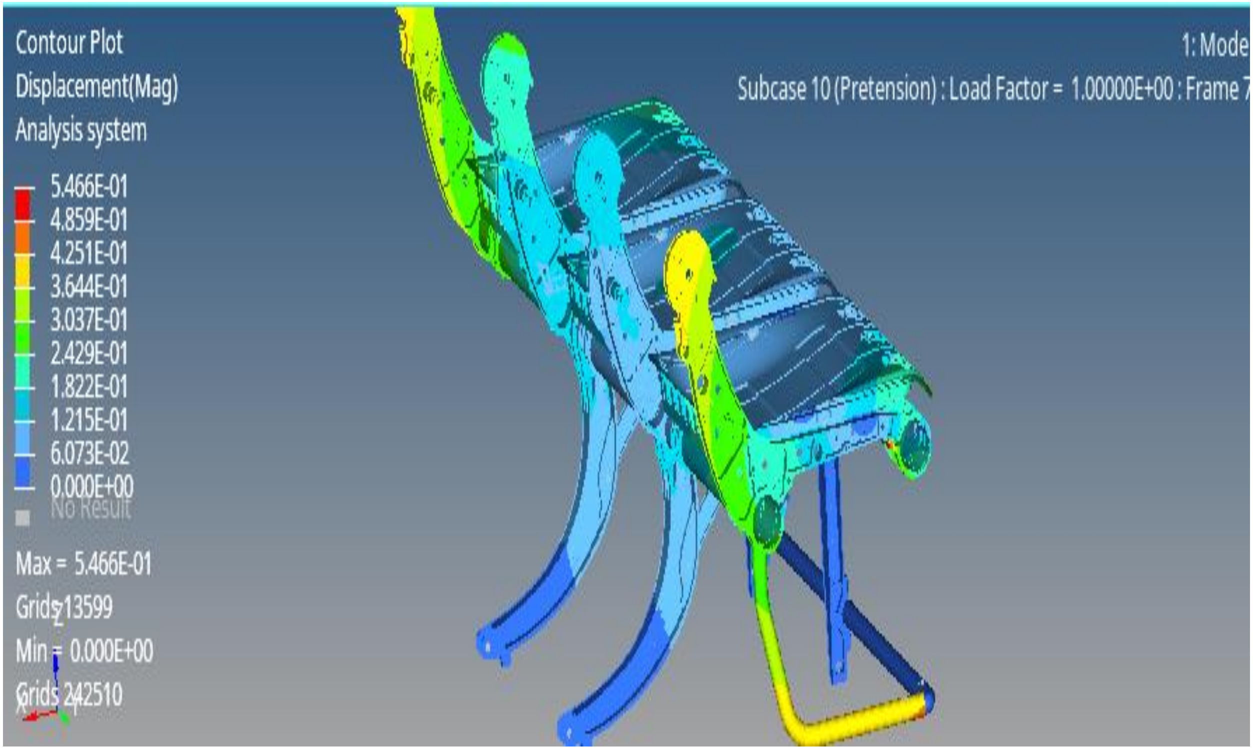

Figure 4.5. Displacement values in back-side view according to analysis results.

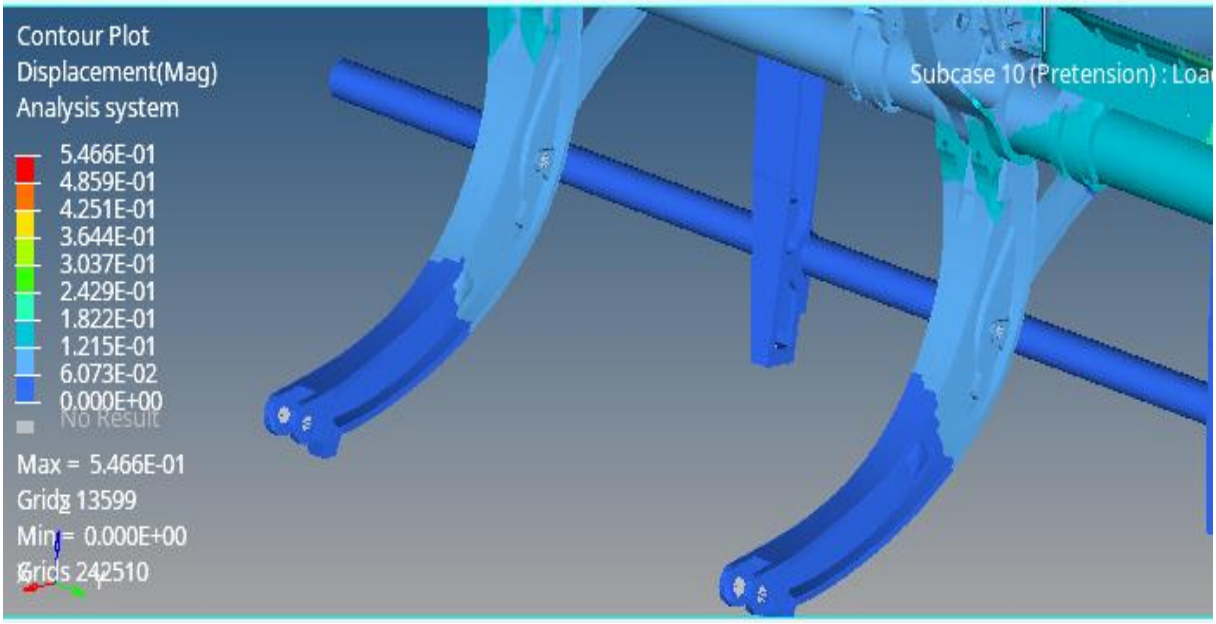

Figure 4.6. Displacement values in detailed back view for the rear leg according to analysis results.

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.

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. In order to pass these tests, the seat structure must be ultra-rigid and high strength.

The results obtained in the topology optimization study are as in the figures below.

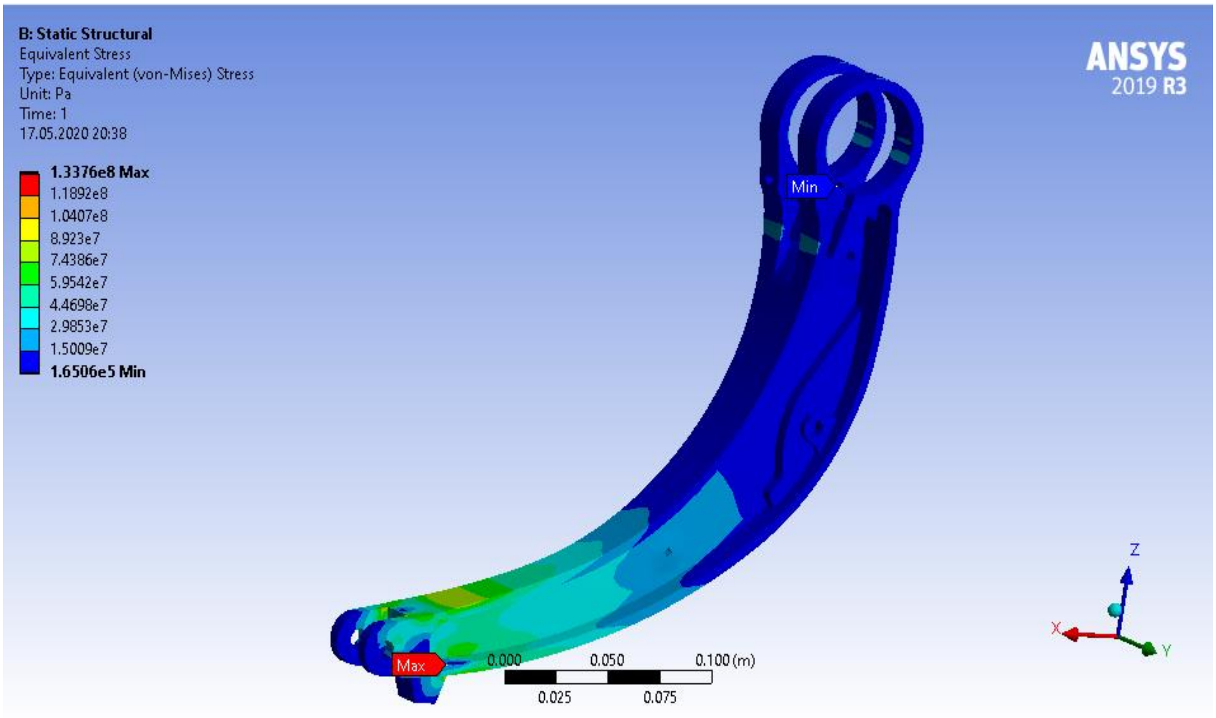

Figure 4.7. Static condition results for topology optimization of rear seat leg.

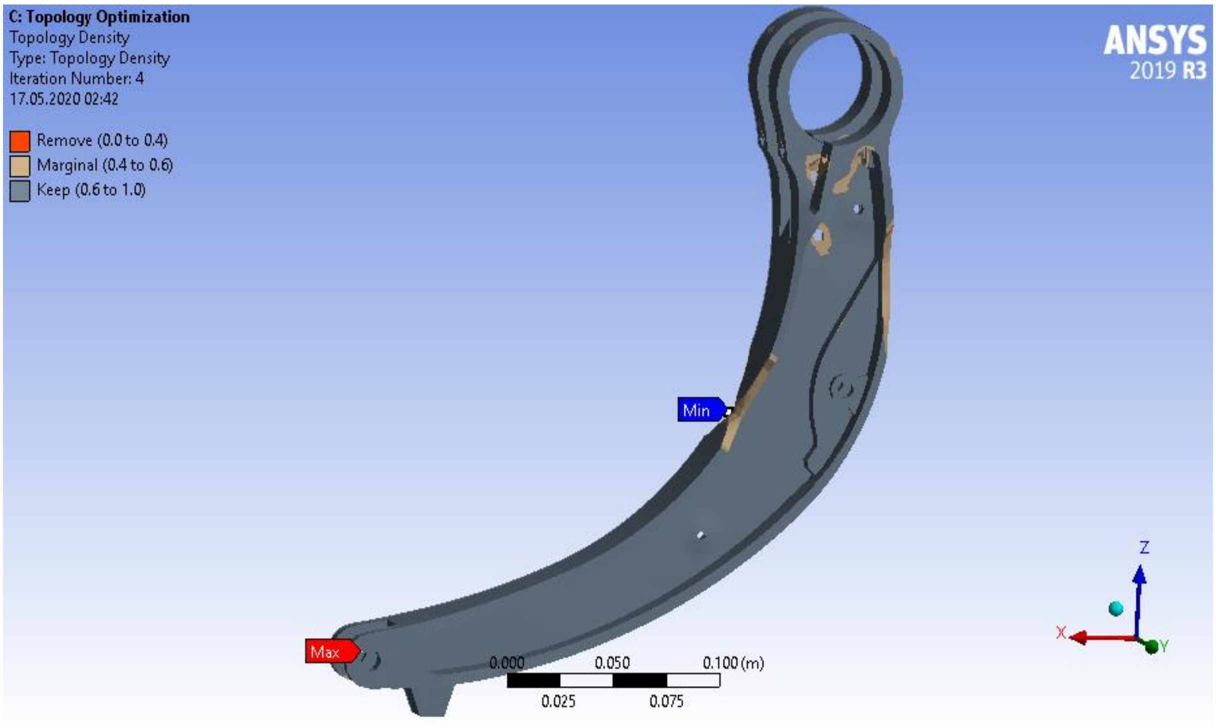

Figure 4.8. Topology optimization result with 90% to retain of rear seat leg.

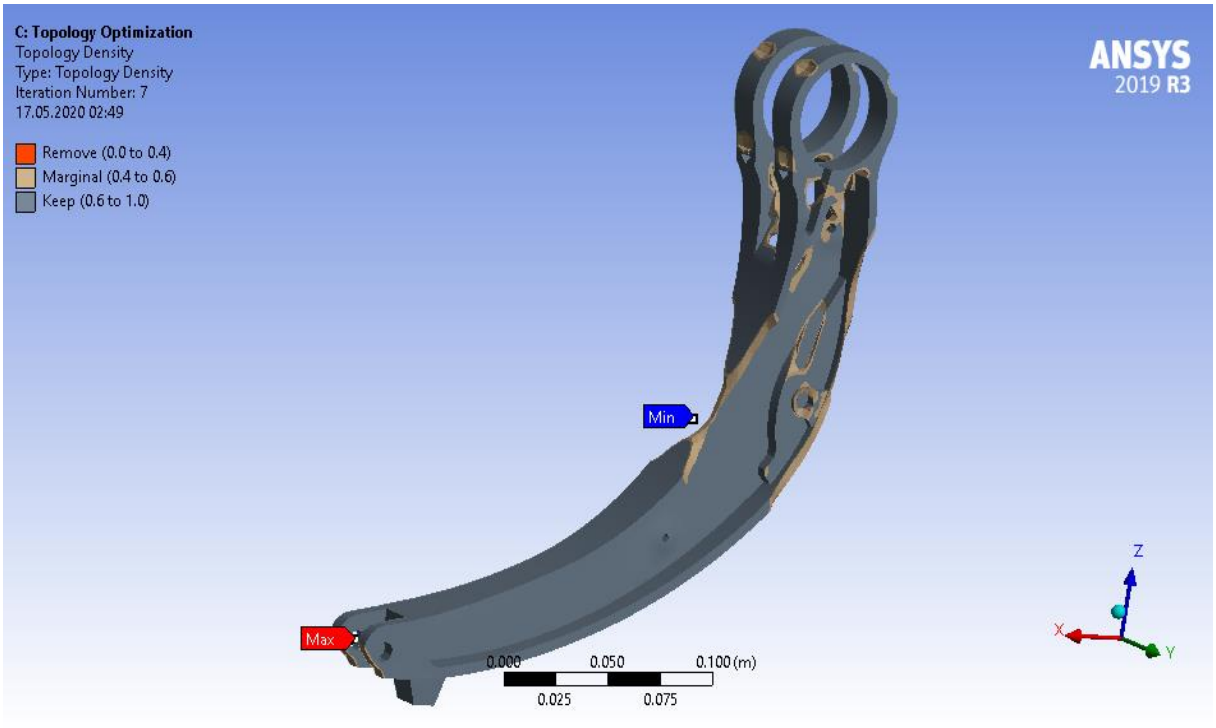

Figure 4.9. Topology optimization result with 80% to retain of rear seat leg.

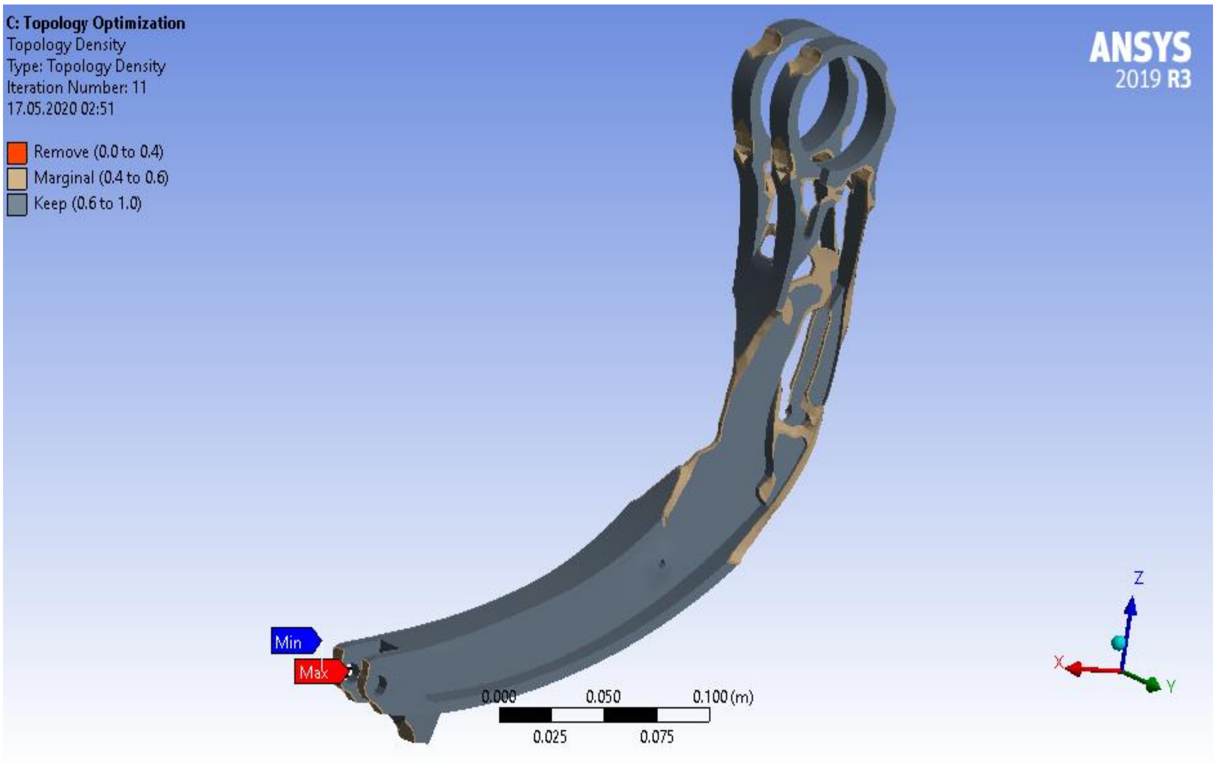

Figure 4.10. Topology optimization result with 70% to retain of rear seat leg.

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 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.

Thanks to the computer-aided analysis, the static analysis was performed and the stresses formed in the model established 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 basis. In addition, topology optimization was made for the behaviour of the rear seat leg under harder conditions by adding to the forces which obtained from the previous model. The stresses and deformations obtained as a result of these analyzes are at predictable and expected levels.

4.2 Comparison and Discussion of the Results

When all the obtained results were evaluated, it was seen that the calculated stress values were around the expected and predictable levels. In the full-model static analysis, the highest stress value occurring within the seat parts is, about 290 MPa. In addition, stress on other parts is in the 30-60 MPa range. When these values are analyzed, it is seen that they are close to the values calculated in other seat analyzes conducted previously within the TSI Aviation Seats Company.

Considering the materials used in this seat which has highest stress value, the tensile and yield strengths are known to be around 500 MPa. With this analysis, it is anticipated that the Epianka model aircraft seat is suitable for static tests and it can successfully pass the static loading test.

On the other hand, the strength values of aluminum alloy which used in other parts, with lower stress value are known around 290 MPa. In this context, 30-60 MPa stresses occurring in other parts are quite normal for static analysis and it is thought that these parts can pass static tests successfully.

When the topology optimization made within the scope of this study was evaluated, it was observed that the stress values were higher (around 134 MPa levels, it can be seen in the next page) since additional loads were made to the loads from the results of the static analysis. The highest stress value is around the expected area and close to the expected values. In the topology optimization made within the scope of these results, 3 different versions of the optimized seat leg models were obtained under these loads.

These optimized models are obtained by reducing the mass according to the intensity of the stress values. When the optimization results are analyzed, it is seen that mass reduction can be made from less critical regions.

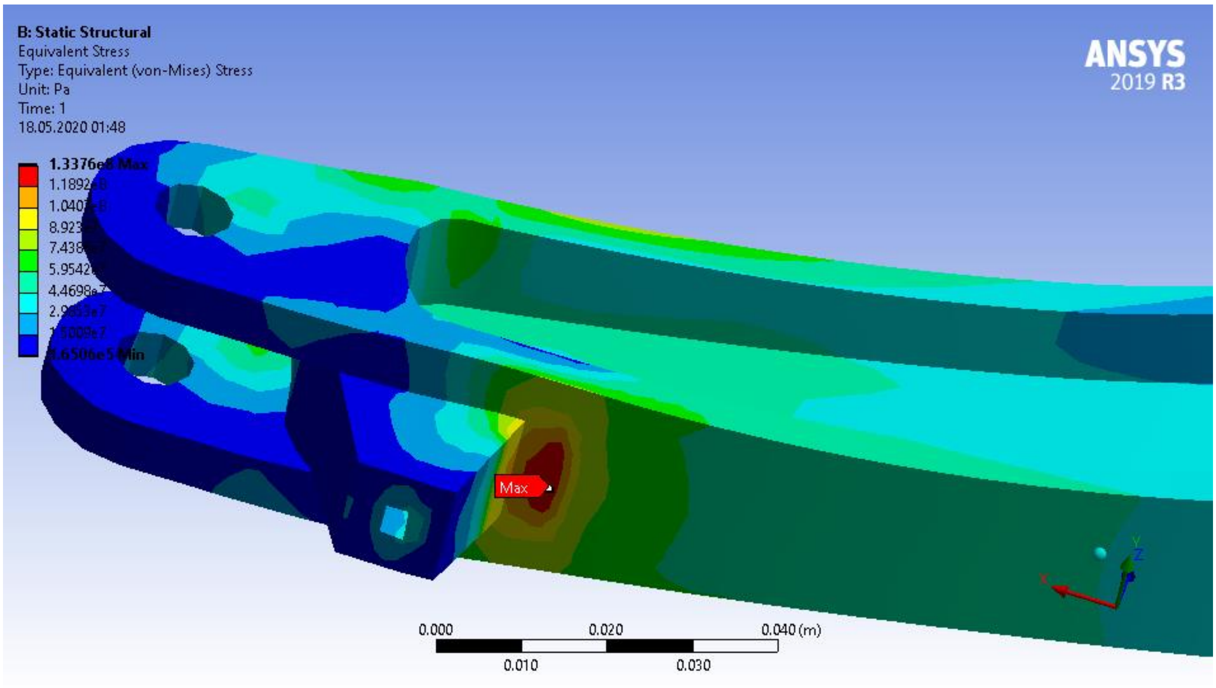

Figure 4.11. Maximum obtained stress value in rear seat leg in topology optimization study.

In the topology optimization study conducted in this study, it has been mentioned before that additional loads were made to the static analysis loads. The reason for this is that the designed seats are exposed to more loading and stress in dynamic analysis. For this reason, topology optimization was made by making additional loads to converge to the results in dynamic analysis conditions. In this way, realistic results were obtained from the topology optimization.

4.3 Potential Improvements

When the static analysis and topology optimization in the scope of this study is reviewed, it is observed that some improvements can be made.

Primarily, the seat model used in static analysis could be chosen as a full detailed model. For example, seat cushions, backrest zone, backrest frame and other seat accessories could also be included in the model. However, since the commercial and intellectual rights of the model used in this study belong to TSI Aviation Seats, only the seat platform was shared for use in this study in accordance with data privacy principles. The effect of unshared seat parts on the results of static analysis and topology optimization which performed in this study is not significant.

The mesh quality in the model used was sufficient for the model to give correct results. However, by choosing a higher quality mesh and smaller element size, the model could be converted to more accurate results. However, as the smaller element size is chosen, since the number of elements will be available, the solution time will be longer and the computing power of the computer needed for the solution must be higher.
This situation can be considered as another potential improvement.

Finally, loads in topology optimization made in this model are an acceptance. Although this acceptance is made by estimating the loads that the seat may be exposed to under dynamic analysis conditions, dynamic analysis of the seat is required to get completely correct results. However, this method was chosen because the dynamic analysis to be performed is not covered by this study.

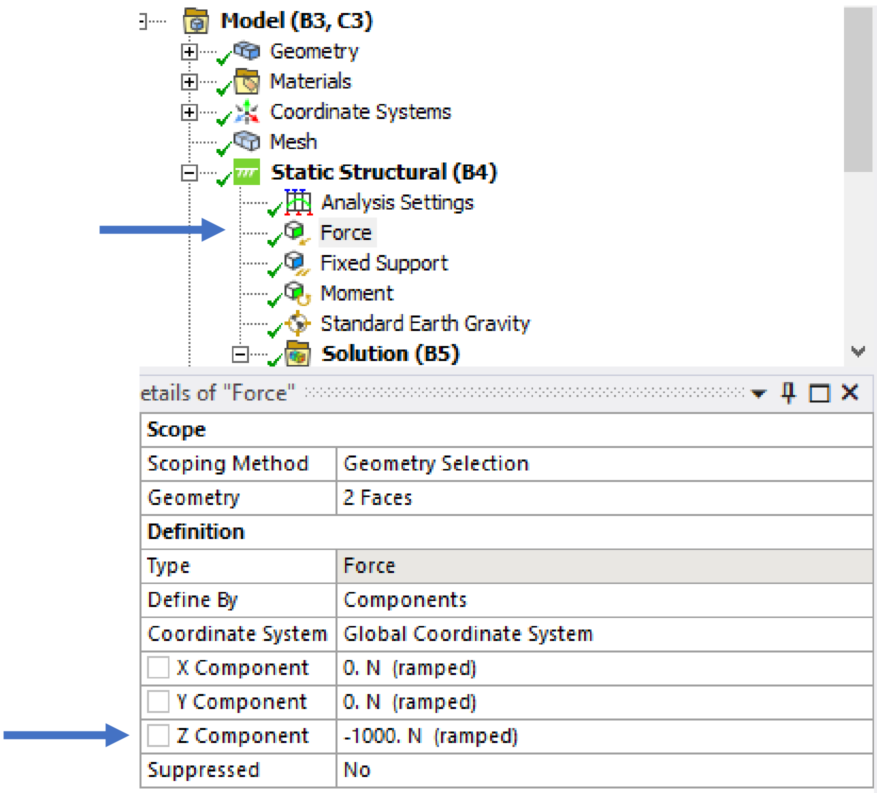

Figure 4.12. Applied force to the rear seat leg for topology optimization model.

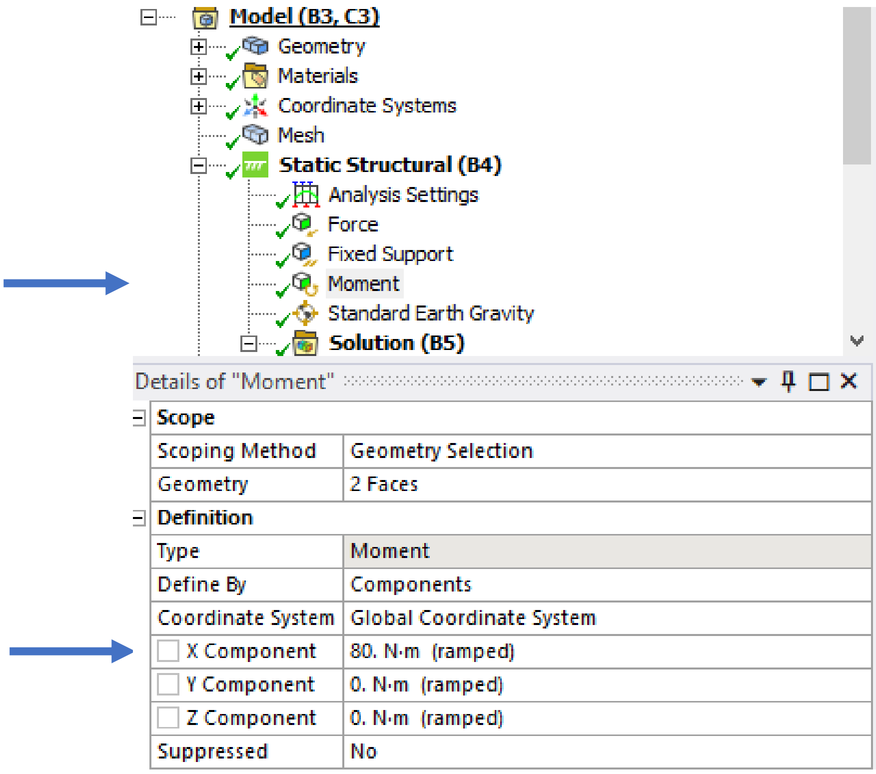

Figure 4.13. Applied moment to the rear seat leg for topology optimization model.

4.4 Verification

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, in order for the finite element 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 testing. However, the results obtained as a result of the analyzes are compatible with the test results for other seat models at TSI Aviation Seats company. In addition, the stress and deformation values in the results obtained from the analysis are at the expected levels.

In addition, in order to check the accuracy of the static analysis results made within the scope of this 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;

$$Force\ for\ front\ legs \approx \frac{300 * 9.81}{2}$$

$$\approx 1471.5\ \text{N}$$

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

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

$$\approx 490.5\ \text{N}$$

According to these results, the forces and reaction forces applied to the seat feet are as follows.

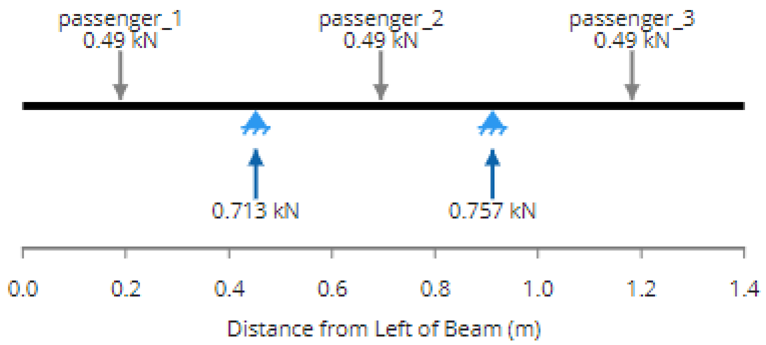

Figure 4.14. The forces acting on the seat legs and reaction forces.

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

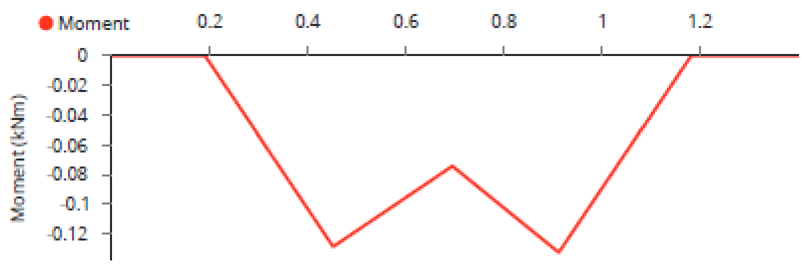

Figure 4.15. Moment on seat legs.

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

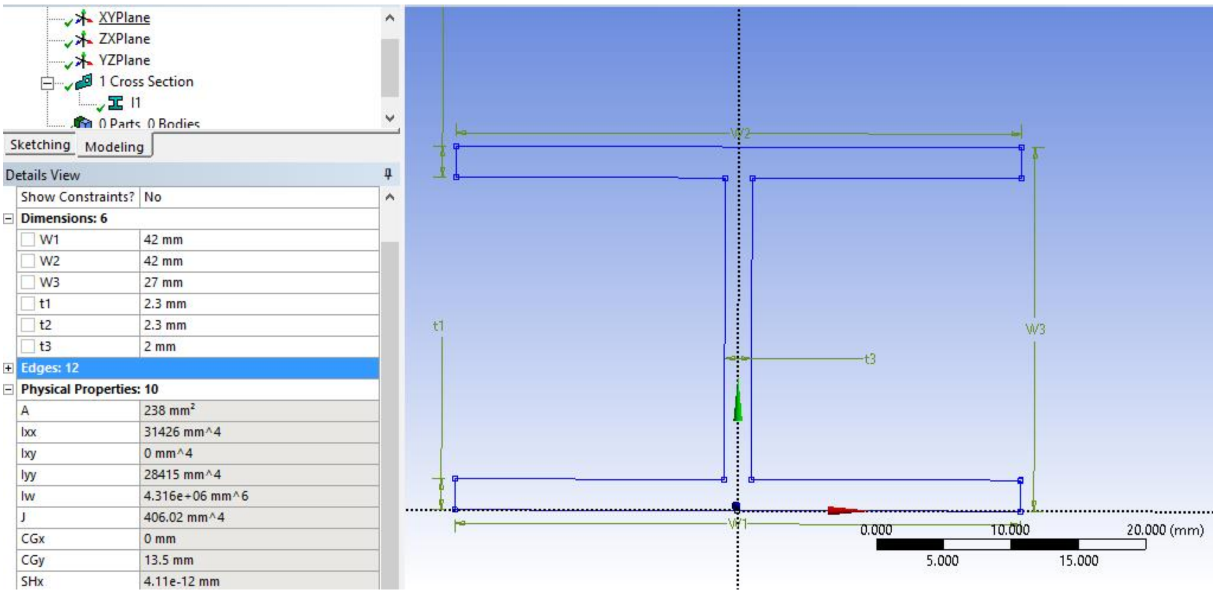

Figure 4.16. The structure and dimensions of the I profile of the seat leg.

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^4

Cross sectional area of the seat leg = 238 mm^2

$\sigma_{total} \approx \sigma_{compressive} + \sigma_{bending}$

$$\approx \frac{F}{A} + \frac{M * y}{I}$$

$$\approx \frac{757}{238} + \frac{132000 * 13.5}{28415}$$

$\approx$ 65.9 N/mm^2 (MPa)

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 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 analysis, it will be understood that the maximum stresses are very close to each other.

CHAPTER 5. CONCLUSIONS

5.1. Conclusion of the Study

This project has been completed with the literature review, model building, static analysis and topology optimization studies and their results. In this context, the subject of this project, which is the how a designed aircraft seat will behave in a static situation, has been studied.

In this study, firstly, the plane seat model to be studied was provided by TSI Aviation Seats company. After that, as a result of the literature review, similar previous studies on this subject were examined and comprehensive information about the subject was obtained. 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, 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 to be made in this study, the next step was passed. At these stages, the boundary conditions and determined loads were made for the seat. Later, 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 method are defined to the model. Finally, the model was run and the results were obtained.

For static analysis:

The results, which obtained from the static analysis, were evaluated. Stress and strain values obtained remained within the expected limits for static analysis. When these results are evaluated numerically, it is seen that it is generally around 30-60MPa. In addition, the highest stress expected is around 290MPa. 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 static tests.

For topology optimization:

After this part, the second part of this project, the topology optimization phase, has been started. Topology optimization is very important for aircraft seats, because every flight-related 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 performed 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.

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.

5.2. Recommendations for Further Work

Within the scope of this study, some analyzes were made and the results were successfully obtained. However, some of the stages could be done more comprehensively and additional studies could be made to these stages.

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 said 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 started. However, for the seat model provided by TSI Aviation Seats, this potential development is beyond the scope of the study, as it is not authorized to create a new CAD data.

In addition, in this study, the performed computer-aided finite element analysis could not be verified by the test method. It is a well-known fact that, although the study has yielded reliable and accurate results, it should be tested and verified before it turns into a final product. In this context, it is more reliable to verify the analyzes made in future studies, if there are suitable physical conditions, by testing method.

5.3. Evaluation of the Current Work from MÜDEK Perspective

5.3.1. Economic Analysis

One of the main objectives of engineering studies is to provide economic benefits. Thanks to the topology optimization analysis that is within the scope of this study, the existing rear seat leg has been reduced by approximately 26%. This means 146-gram reduction for a seat leg. Considering that there are 2 seat rear legs in an airplane seat, it can be said that a weight reduction of approximately 23 kilograms was made for a 240-person airplane thanks to the topology optimization. Considering that the 1-year fuel expense of the airline companies for a 1kg weight is about $ 140, the economic profitability of the work will be understood.

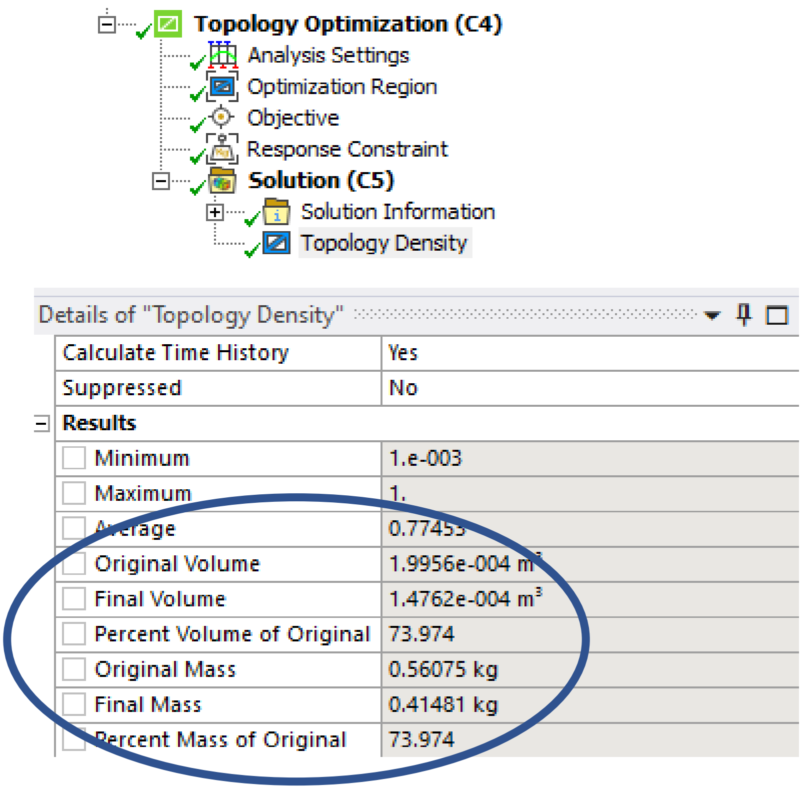

Figure 5.1. Topology optimization mass reduction results.

5.3.2. Real Life Conditions

The studies carried out within the scope of this thesis have been verified by analysis in computer environment, and in order to understand its compatibility in real life conditions, the results of the analysis must be verified by performing a physical test. In this context, judgments that can be made for real life conditions will be based on the accuracy of the analysis. The analyzes made within the scope of this study were made under the supervision of the thesis advisor and expert engineers, and it is believed to give high accuracy results.

However, as stated above, it would be more correct to subject the analyzed seat and the topology-optimized seat leg to physical testing before they turn into products.

When the figure below is examined, the design and production stages can be better understood [18].

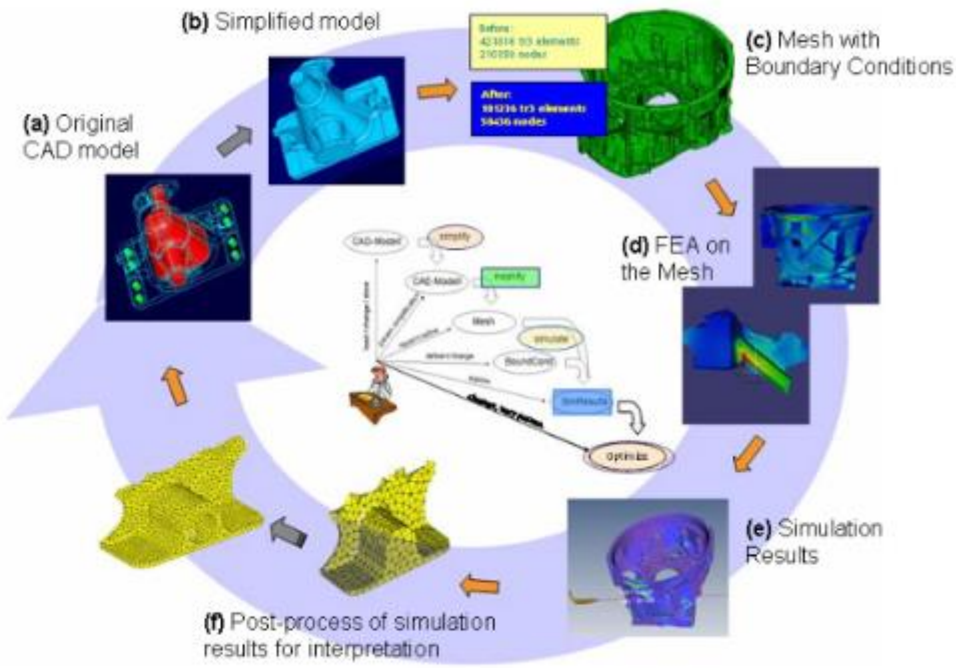

Figure 5.2. CAD-CAE-Production cycle.

5.3.3. Producibility

An important detail for a product designed or analyzed is the manufacturability of this product.

Because, if the product designed / analyzed under the existing production conditions cannot be produced, it will not provide any economic benefit, and engineering studies will also result in financial damage.
One of the topics covered in the content of this thesis is topology optimization. The graphic results obtained as a result of the topology optimization study should be taken into CAD environment and reviewed. In this way, a design that is suitable for topology optimization outputs and can be produced can be obtained. However, professional CAD and CAE engineers are required for these studies.

Topology optimization within the scope of this study was completed within the results obtained in the FEA program. The steps mentioned above must be completed to ensure manufacturability.

5.3.4. Constraints

The works carried out within the scope of this study are subject to some limitations. CAD models used in all of the first studies were taken from TSI Aviation Seats for academic purposes. Therefore, it is not possible to use the model or study outputs used in this study for commercial purposes.
In addition, the aircraft seat, which is the subject studied in this thesis, cannot be attached directly to the plane after the analyzes and development tests. After the computer-aided analysis and physical development tests carried out during the development phase, it is mandatory to test the seat in the test centers approved by the international aviation authorities (FAA, EASA etc.). The designed seat will receive the "onboard" approval after passing these tests, also known as certification tests.
In this context, considering the commercial evaluation of the study in this thesis, this situation should be taken into consideration.

REFERENCES

[1] Airplane Seat Types, <https://www.aircraftcompare.com/blog/types-of-airplane-seats/>, taken in 16.01.2020

[2] Bhonge, P.S., A methodology for aircraft seat certification by dynamic finite element analysis. Diss. Wichita State University, (2008).

[3] Düvenci, F. Araç yolcu koltuk ayaklarında farklı malzemelerin kullanımının sonlu elemanlar yöntemi ve testler ile mekanik etkisinin karşılaştırılması. MS thesis. Uludağ University, (2017).

[4] García Nieto, P. J., Vilan, J. V., Coz Diaz, J. D., "Analysis and study of an automobile rear seat by FEM." International Journal of Computer Mathematics 86.4 (2009): 640-664.

[5] Shivakumar, M. R., Kamat, A., & Prakash, G. S "Comfort Design And Analysis Of Driver Car Seat Using Finite Element Analysis." (2016).

[6] Siefert, A., S. Pankoke, and H-P. Wölfel. "Virtual optimisation of car passenger seats: Simulation of static and dynamic effects on drivers' seating comfort." International Journal of Industrial Ergonomics 38.5-6 (2008): 410-424.

[7] Aydıner, Ş.S., Yolcu koltuğu ayağının sonlu elemanlar yöntemi ile optimizasyonu. MS thesis. Uludağ University, (2018).

[8] Bhonge, P., & Lankarani, H., Finite element modeling strategies for dynamic aircraft seats. No. 2008-01-2272. SAE Technical Paper, (2008).

[9] Bulgur, A., Koltuk bağlanti elemanlarinin mukavemetinin; ilk hiz koşullari ve dummy etkisi ile crash testi esnasinda simülasyonu. MS thesis. Gebze Institute of Technology, (2006).

[10] Görgün, G., Uçak Koltuğu Kompozit Sirtlik Sisteminin Modellenmesi Ve Sonlu Elemanlar Analizi. MS thesis İstanbul Technical University, (2019).

[11] Hutton, D. V., Fundamentals of Finite Element Analysis, 1. Edition, Mc Graw Hill Higher Education, USA.

[12] Dede, G., 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).

[13] Altair University, Practical Aspects of Structural Optimization, A Study Guide, 2014.

[14] Callister W. D. and Rethwisch, D. G., Materials Science and Engineering, 8. Edition, WILLEY, USA.

[15] Url, < http://www.value-design-consulting.co.uk/boundary-conditions.html, taken in 20.01.2020

[16] Hypermesh Tutorials, < https://altairhyperworks.in/edu/contest/aoc/2013/tutorials-and-downloads.html#.XioHF8gzbIU > taken in 20.01.2020.

[17] Kelleci, Z. E., (2020), Personal Meeting, TSI Aviation Seats Company.

[18] Catalano, C. E., Camossi, E., Ferrandes, R., Cheutet, V., & Sevilmis, N. A product design ontology for enhancing shape processing in design workflows. Journal of Intelligent Manufacturing, (2009).