Chapter 6 Redesign of a fatigue machine guide plate based on topology optimization

Capítulo 6 Rediseño de una placa guía de una máquina de fatiga con base en optimación topológica

SOTO-MENDOZA, Gilberto†*´, MARTÍNEZ-GARCÍA, José´, EDMUNDO-MASTACHE, Jorge´ and HERNÁNDEZ-GÓMEZ, Luis Héctor´´

´Tecnológico Nacional de México. Tecnológico de Estudios Superiores de Jocotitlán. Department of Mechatronics Engineering. Toluca-Atlacomulco Highway KM 44.8, Ejido de San Juan y San Agustín, Jocotitlán, 50700 Mexico.

´´Instituto Politécnico Nacional. ESIME Zacatenco Unit. Graduate Studies and Research Section. Building 5, 3rd. floor. Professional Unit Adolfo López Mateos. Col. Lindavista. Alc. Gustavo A. Madero. 07738 Mexico City. Mexico.

ID 1st Author: *Gilberto, Soto-Mendoza* / **ORC ID:** 0000-0001-7357-9445, **CVU CONACYT ID:** 635154

ID 1st Co-author: *José, Martínez-García* / **ORC ID:** 0000-0002-7797-1062, **CVU CONACYT ID:** 612069

ID 2nd Co-author: *Jorge, Edmundo-Mastache* / **ORC ID:** 0000-0001-6104-6764, **Researcher ID Thomson:** H-1187-2018, **CVU CONACYT ID:** 544943

ID 3rd Co-author: *Luis Héctor, Hernández-Gómez* / **ORC ID:** 0000-0003-2573-9672, **CVU CONACYT ID:** 5107

DOI: 10.35429/H.2021.9.1.97.113

G. Soto, J. Martínez, J. Edmundo and L. Hernández

* gilberto.soto@tesjo.edu.mx

A. Ledesma (Coord.). Engineering Science and Technology. Handbooks-©ECORFAN-México, Estado de México, 2021.

Abstract

Machinery components are subjected to dynamic loads. In particular, the fatigue machines must be designed for these types of conditions. On the other hand, the industry demands that it is sought to consume the least amount of raw material for its construction, that is, to optimize. In general, optimization tasks have been carried out mostly by trial and error. In the present work, a redesign of a guide plate of a fatigue machine was carried out based on Topology Optimization. For this purpose, Static Structural, Topology Optimization, Fatigue and Modal Analysis were carried out. With this, a new design is obtained with a reduction in its raw material of 61%. The component was designed for infinite life so that it will not compromise its structural integrity throughout the life of the equipment operation.

Analysis, MEF, Natural frequency, Fatigue

Resumen

Los componentes de las máquinas están sometidos a cargas dinámicas. En particular, las máquinas de fatiga deben ser diseñadas para este tipo de condiciones. Por otro lado, la industria exige que se busque consumir la menor cantidad de materia prima para su construcción, es decir, optimizar. En general, las tareas de optimización se han llevado a cabo mayoritariamente por ensayo y error. En el presente trabajo, se realizó un rediseño de una placa guía de una máquina de fatiga basado en la Optimización Topológica. Para ello, se ha realizado un análisis estructural estático, de optimización topológica, de fatiga y modal. Con ello se obtiene un nuevo diseño con una reducción en su materia prima del 61%. El componente fue diseñado para una vida infinita, de manera que no comprometa su integridad estructural a lo largo de la vida de operación del equipo.

Análisis, MEF, Frecuencia natural, Fatiga

6.1 Introduction

An issue of relevance for mechanical components that are subjected to loads that change over time is fatigue. Fatigue occurs in a component when it is subjected to variable loads, which can be dynamic (uniformly varying loads) or random (seismic loads, wind loads, etc.). Fatigue failures represent a very high economic cost. These costs come from fatigue failures of land vehicles, trains, aircraft, bridges, cranes, offshore oil well structures, as well as a variety of machinery and equipment. They also involve human lives (Budynas & Nisbett, 2019), (Ugural, 2015). A fatigue testing machine allows measuring the fatigue resistance property of the material. Knowing this property, the number of cycles the material can be subjected to while retaining its structural integrity can be determined. Fatigue tests can be tension, compression, bending, torsion or a combination of stresses (Fatigue Test, 2021).

Today, machinery designers are required to minimise the cost of their designs. This is in order to be more competitive in the marketplace. It is a difficult challenge. One way to reduce cost is to remove material, to design leaner components. However, this has as a consequence an impact on the strength of the mechanical elements that make it up, its natural frequency is reduced, among others. The traditional way to optimise the product is to do it by trial and error. Such a technique is costly and time-consuming (Chen & Liu, 2018). With technological development, software has emerged that allows the mechanical behaviour of machines to be simulated. It is essential, that the designer has the scientific knowledge and is able to manipulate these programs to be much more efficient and competitive.

The programs that have been developed are mostly based on the Finite Element Method. They can be used to analyse the mechanical behaviour of complex geometries that cannot be obtained or would be time-consuming with traditional techniques. Additionally, analyses can be coupled to consider various phenomena. In this paper, a methodology is presented to optimise a mechanical component subjected to variable loads by means of numerical simulation. This involves static-structural analysis, topology optimisation studies, fatigue analysis and modal analysis in order to ensure its structural integrity during operation by obtaining a design with a lower weight than that proposed in the traditional way.

Computer Aided Design and Computer Aided Engineering programmes will be used for this purpose. These programmes have taken on another dimension in the training of today's engineers and have generated a link with industry that demands ever faster responses. There is a model focused on the cooperation of industry and universities to standardise and certify human resources (Lukač, 2011).

In addition, surface meshing and sub-modelling will be used to reduce the consumption of computational resources required for simulation. These techniques are very useful when academic licences are available that are limited in the number of nodes or elements that can be solved.

Section two gives a general description of the type of fatigue machine where the component to be optimised is located, as well as its overall dimensions. Then, in section three, the methodology followed to optimise the component is presented. In this section, only a general explanation is given; in the subsequent sections, each of its stages is covered.

It is essential to understand the theoretical basis of the phenomenon to be studied and not just see the programme as a black box. You must have the engineering knowledge to know what is intended to be obtained from the simulation and to verify that the results are reliable. The fact is that there may be several particular cases in which the results do not match reality and you may fall into error due to lack of experience. Topic four presents the theoretical basis of the studies carried out.

Section five deals with the numerical simulation. It starts with a structural analysis coupled to the topology optimisation study. With the stl model delivered by the optimisation, a redesign of the original part is created and the static simulation is performed again. Given that the component is subjected to fatigue loads, this analysis is carried out with the aim of achieving an infinite life. To ensure that the stress results do not vary with mesh size, a mesh sensitivity analysis is carried out and to reduce computational cost, the sub-modelling technique is used.

In section six, the results of deformation, mass, fatigue and natural frequencies are analysed. The original guide plate component is compared with the proposed new design. Finally, conclusions are drawn.

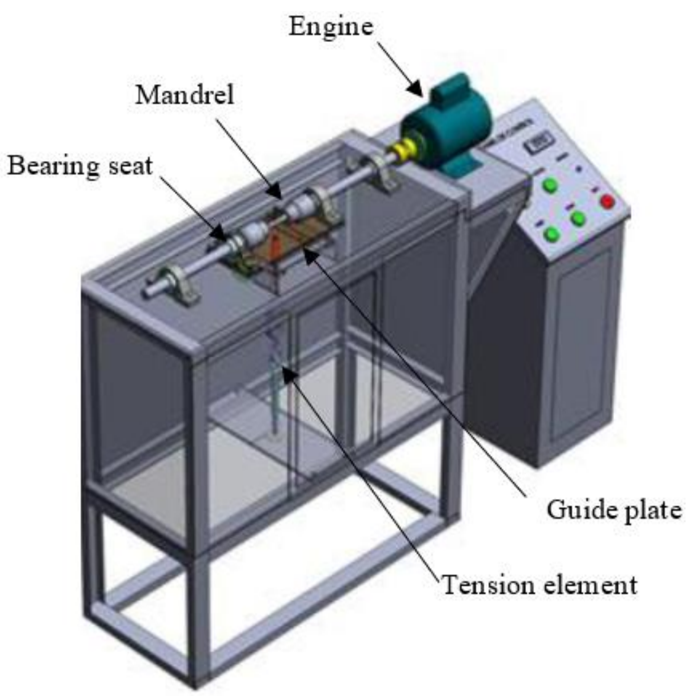
6.2 Description of the problem

One of the needs of educational institutions is to have infrastructure in their laboratories to complement the training of engineers, giving the opportunity to perform experiments. Materials testing is a key area. Among the tests performed is fatigue strength. This material property is essential for the design of machinery.

Many institutions lack such equipment. Some others have encouraged thesis work in the design of such academic equipment to fill this need. Some examples are Universidad de los Andes (Londoño, 2019), Escuela Superior Politécnica De Chimborazo (Castagneto, 2020), Universidad Católica Santo Toribio de Mogrovejo (Olivera, 2019), among others.

The common design form of these teams is shown in Figure 6.1.

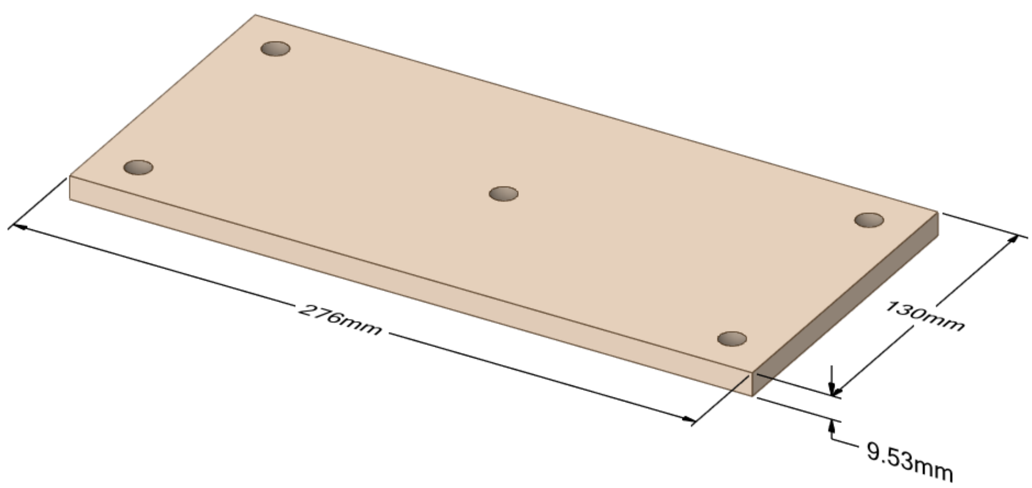
Figure 6.1 Fatigue machine



Source: (Simbaña & Chango, 2012)

Machine components must be designed according to the operating conditions. The case study requires a fatigue analysis and a modal analysis because it is subjected to dynamic loads. The idea of building a component using as little material as possible without losing its functionality involves several challenges, although numerical simulation tools help to this end with a shorter execution time than the traditional trial-and-error way. This paper presents the redesign of a fatigue machine guide plate based on topology optimization. **Figure 6.2** shows the overall dimensions of the guide plate of the case study.

Figure 6.2 Guide plate dimensions



6.3 Methodology

The design of a machine depends to a large extent on the creativity of the designer, on the ideas he or she has to perform the desired function. The following methodology is focused on optimising components that are subjected to variable loads and are intended to be lightweight. The stages for this purpose are:

1. Create a crude model

2. Determine the operating conditions of the component

3. Structural simulation

4. Simulation from a topology optimization perspective

5. Creation of the new model on the basis of topology optimization
6. Structural analysis of the new design
7. Fatigue study
8. Modal analysis
9. Analysis of results

The following section begins with the theoretical basis for the type of analysis required for the case study.

6.4 Theoretical basis

6.4.1 Structural analysis

The use of the Finite Element Method (FEM) has grown in engineering. It allows structural analysis of complex geometries among other types of analysis. In general, its stages are as follows:

- The model (geometry) is created in some Computer Aided Design (CAD) software.
- The mechanical properties of the material are defined.
- The model is discretised (mesh generation).
- Boundary conditions are applied.
- The solve simulation is run.
- Finally, the required results such as deformation, stresses, etc. are sought (post-processing).

For a linear structural analysis we have the following equation:

$$[K]\{D\} = \{F\} \tag{1}$$

Where:

{D} = Displacement vector

{F} = Force vector

{K} = Stiffness matrix of the structure

In the case study, several structural analyses are carried out. It is assumed that the design will work in the linear zone of the material, so Equation 1 is the one that the program will be solving.

6.4.2 Optimisation

The requirement to obtain a strong mechanical component using the least amount of material in order to make it lighter and/or reduce costs has led to the development of new technologies. The typical way to perform optimisation is through trial and error, but this procedure is deficient because of the cost and time consumed to execute it. Nowadays, numerical simulation has become a powerful tool in engineering and thanks to more powerful computers, a large number of calculations involving these techniques can be solved. Numerical optimisation compared to the traditional technique is more productive and economical (Chen & Liu, 2018).

There are two perspectives in numerical optimisation. The first is topology optimization and the second is parametric optimisation. The designer must optimise from different perspectives considering the process involved in the design itself.

The aim of topology optimization is to find the ideal distribution of the defined material in a given space considering the loads and boundary conditions. With this it is possible to obtain a good initial design concept. This type of optimisation should be used in the early stages of design.

Parametric optimisation, on the other hand, focuses on determining the shape and dimension of the structure in question. The design variables are usually length, thickness, etc. and the state variables are stress, deformations, etc.

6.4.3 Fatigue analysis

Fatigue occurs in a component when it is subjected to variable loads, which can be dynamic (uniformly varying loads) or random (seismic loads, wind loads, etc.). Among the serious accidents due to fatigue failure is the British Comet Jet in 1954, this caused the Comet flights to be suspended and the production of the British jet was stopped (The Aircraft Accidents That Revolutionised Aircraft Design - BBC News World, 2014). Fatigue study is essential in the design of mechanical components subjected to varying loads because of the economic impact of failure.

There are different approaches to fatigue study. The method of interest for this study is the stress-life method. This method is the most traditional, it is easy to implement and there is a lot of experience. However, it is only applied to elastic stresses, it is limited to low and high cycle stresses. In low cycle applications it is less accurate. It relies on stress-life (S-N) curves (Budynas & Nisbett, 2019). The case study is high cycling.

6.4.3.1 Fatigue terminology

Among the terminology found in fatigue we have:

Stress range

$$\sigma_r = \sigma_{max} - \sigma_{min} \quad (2)$$

Mean stress

$$\sigma_m = \frac{\sigma_{max}+\sigma_{min}}{2} \quad (3)$$

Alternating stress

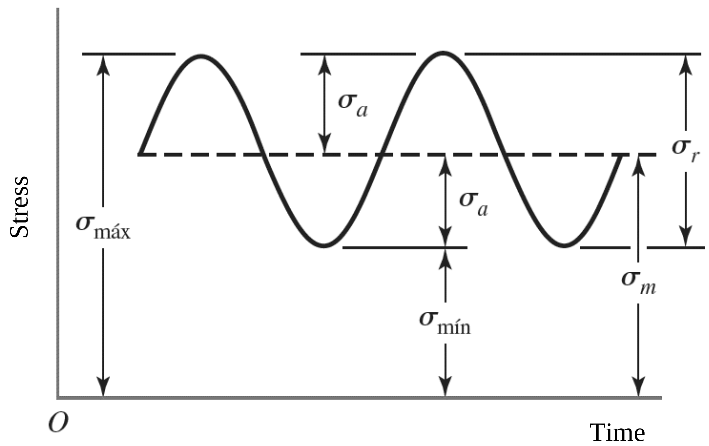
$$\sigma_a = \left|\frac{\sigma_{max}-\sigma_{min}}{2}\right| \quad 4)$$

Investment ratio (stress ratio)

$$R = \frac{\sigma_{min}}{\sigma_{max}} \quad (5)$$

Figure 6.3 illustrates these parameters. The horizontal axis represents the time and the vertical axis the applied stress. The illustration shows a sinusoidal behaviour, however, it does not necessarily have to be that way. The purpose of the graph is to illustrate the terminology: Maximum stress, minimum stress, alternating stress and stress range.

Figure 6.3 Fatigue terminology



Source: (Budynas & Nisbett, 2019)

6.4.3.2 Mean stress different from zero

In practical cases, you have different stress ratios, i.e. the mean stress is different from zero. In general, laboratory tests typically start from a mean stress equal to zero. The effect of the mean stress is that, if it increases, the amplitude of the equivalent fatigue stresses decreases. There are several theories of fatigue failure for non-zero mean stress including: Sodeberg, Goodman, Gerber, Langer, etc. Goodman's criterion is perhaps the most widely used for the design of machinery components subjected to cyclic loading (Mott et al., 2018). In this study we will look for the component to have infinite life and the Goodman criterion (Equation 6) will be used. It is important to emphasise that Dowling's research indicates that this criterion is not accurate when finite life is estimated and other criteria are suggested (Dowling et al., 2009).

$$\frac{S_a}{S_e} + \frac{S_m}{S_{ut}} = 1 \tag{6}$$

Where:

Sa = Alternating strength.

Se = Fatigue strength limit

Sm = Mean stress

Sut = Ultimate stress

6.4.4 Modal analysis

In order to determine the behaviour of structures subjected to loads that change over time, structural-dynamic analyses are carried out. In these, the inertia and damping of the structure play an essential role. There are several types of such analyses such as: Random, Transient, Modal, among others. In the present work a modal analysis is carried out in order to obtain the natural frequencies and the modes of vibration of the system.

Equation 7 is used to obtain the natural frequency and modes of vibration of the system. It is assumed that the vibration is free, the mass and stiffness matrices are constant (Howard & Cazzolato, 2015).

$$[M]\{\ddot{u}\} + [K]\{u\} = \{0\} \tag{7}$$

Where:

$[M]\{\ddot{u}\}$ = Inertial force

[K]{u} = Elastic force

[M] = Mass matrix

[C] = Damping matrix

[K] = Stiffness matrix

$\{\ddot{u}\}$ = Acceleration vector

$\{\ddot{u}\}$ = Velocity Vector

{u} = Displacement vector

(t) = Time

The oscillation is assumed to be harmonic of the form:

$$\{u\} = \{\emptyset\}_n \cos \omega_n t \tag{8}$$

Substituting the value into the above equation converts it to:

$$(-\omega_n{}^2[M] + [K])\{\emptyset\}_n = \{0\} \tag{9}$$

$\{\emptyset\}_n$ = The eigenvector representing the modes of vibration of the natural frequencies.

ω_n = It is the natural circular frequency

A trivial solution is $\{\emptyset\}_n = 0$, l he is following series of solutions corresponds to Equation 10.

$$|[K] - \omega_n{}^2[M]| = 0 \tag{10}$$

With the natural circular frequencies (eigenvalues) the natural frequency f_n can be obtained (Equation 11).

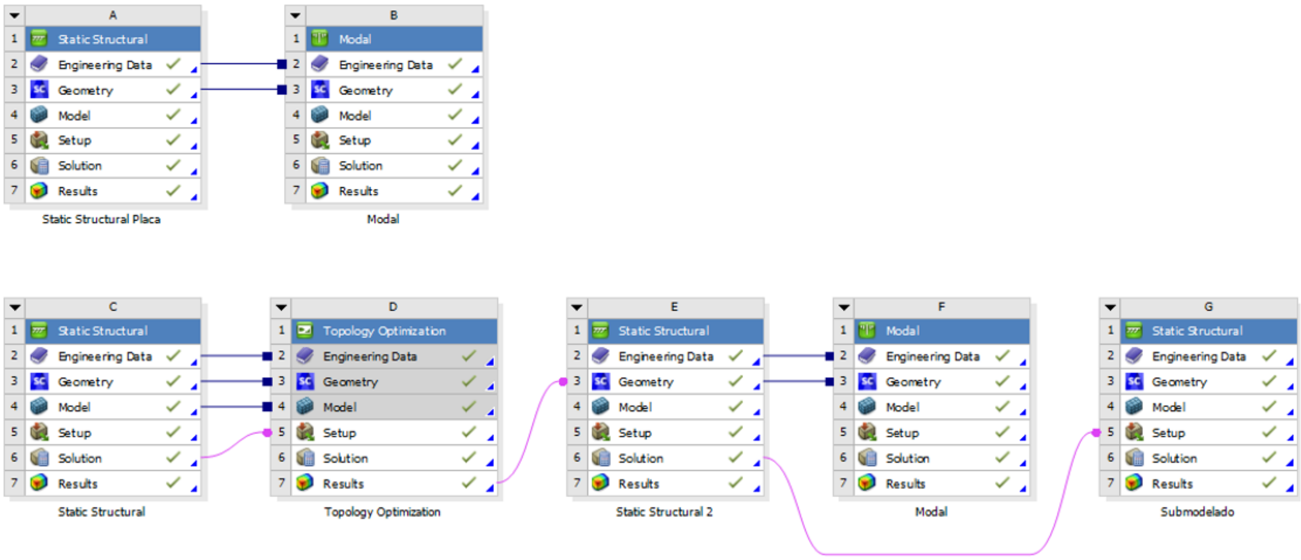
$$f_n = \frac{\omega_n}{2\pi} \tag{11}$$

6.5 Numerical simulation

The numerical simulation was performed in ANSYS® Student, a free software, but it is limited to the number of elements or nodes it can solve. For structural analyses the number of nodes or elements, whichever is reached first, is 128 000 (Download Ansys Student | Workbench-Based Simulation Tools, 2021).

The general scheme that was realised on the Workbench platform is shown in Figure 6.4. Workbench is the platform where all the modules offered by ANSYS® are located. In addition, it allows coupling the modules.

Figure 6.4 Diagram of the analyses in ANSYS® Workbench

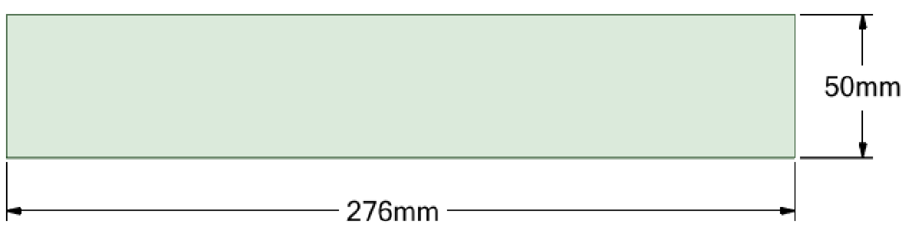


Following the methodology, the following steps are taken:

6.5.1 Creating a crude model

To start the simulation, the geometrical model is required. Geometries can be lines (in this case a cross section is indicated), surfaces or volumes. It should always be simplified in order to use less computational resources. In the case of the study we start with a surface as we intend to build the model with plates. Figure 6.5 shows this surface with its dimensions. The thickness considered is 4.76 mm (3/16"). In addition, the SpaceClaim program was used to prepare the geometry for the simulation by splitting the edges. This allows to select where the boundary conditions are to be applied.

Figure 6.5 Initial geometry



6.5.2 Determination of the operating conditions of the component

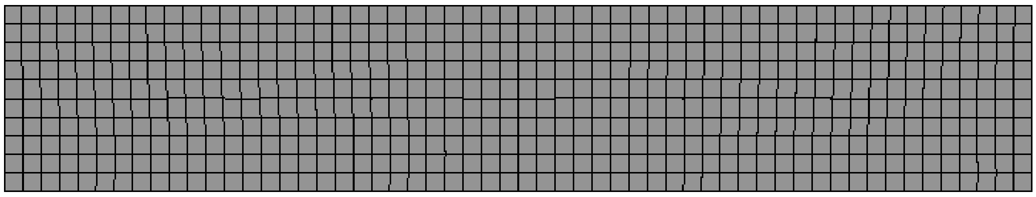
The component is attached to two bearing housings fixed with bolts (see Figure 6.1). A maximum vertical force of 490.5 N is applied to induce a bending moment in the specimen. The component is to be constructed from a structural steel with the following properties:

- Young's modulus = 200 GPa
- Poison Ratio = 0.3
- Yield stress = 250 MPa
- Ultimate stress = 460 MPa

6.5.3 Structural simulation

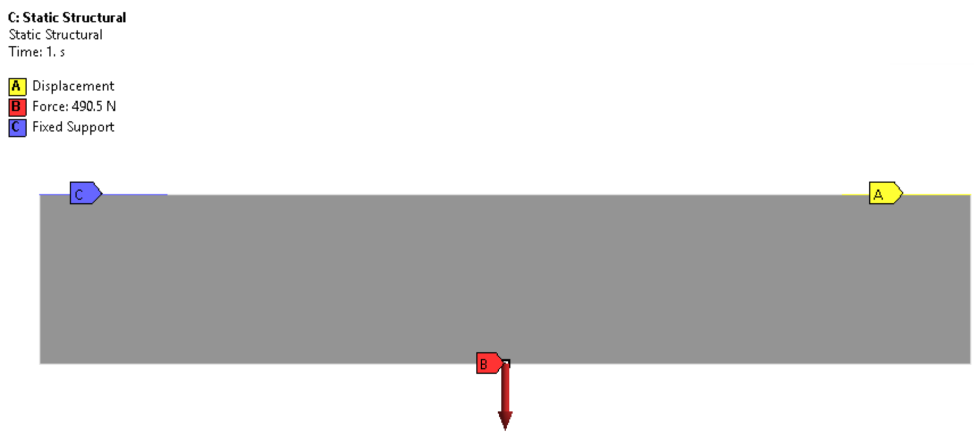
Once the material has been assigned to the model and the boundary conditions are ready to be assigned, the mesh is generated (see Figure 6.6). For structural analysis it is recommended that the mesh quality "Element Quality" is not less than 0.2. A mesh size control of 5 mm was used. The number of nodes is 627 and the number of elements is 560. Its minimum mesh quality is 0.96 and the average is 0.98. The closer to one the better. This first analysis is coupled with the topology optimization analysis.

Figure 6.6 Mesh



The boundary conditions are: on the left side a fixed support "Fix support" and on the right side a displacement constraint "Displacement" in Y-direction. The load is a vertical force of 490.5 N in the vertical direction (see Figure 6.7).

Figure 6.7 Border conditions



6.5.4 Simulation from topology optimization perspective

The idea of this simulation is to obtain the material distribution in such a way that the component is as rigid as possible, i.e. that it deforms as little as possible under the load to which it is subjected. The process requires the following:

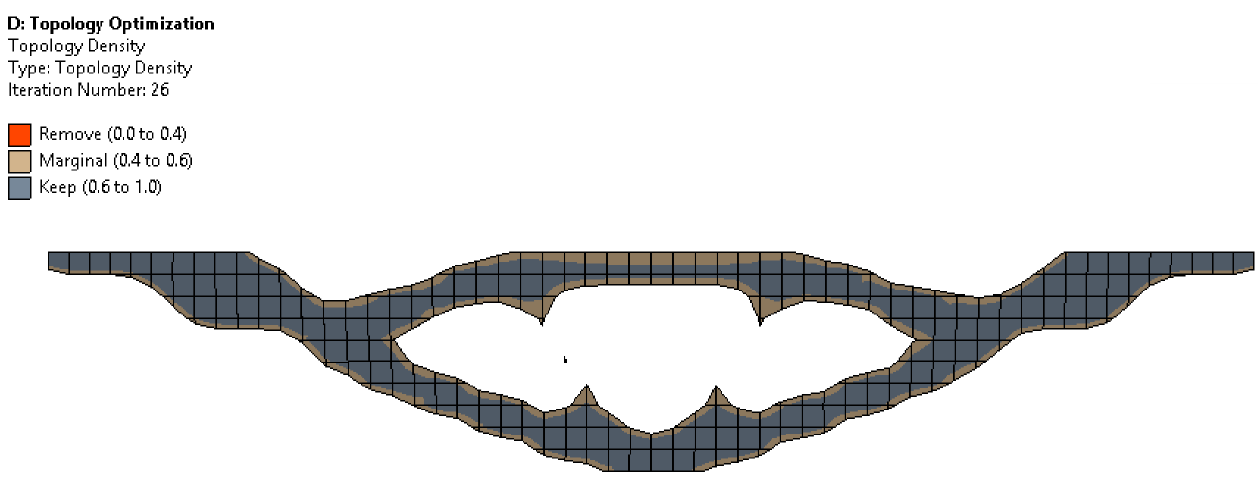
- Optimisation region. The entire component is given and boundary conditions are excluded.

- Objective. In this case, the model is to be as rigid as possible. This option is indicated as "Compliance".

- Restriction. It is indicated to reduce the mass to1 30% for this case, this parameter can be adjusted to see different distribution options.

The topology optimization analysis is coupled to the structural analysis (see Figure 6.4). The result of this simulation is shown in Figure 6.8.

Figure 6.8 topology optimization



6.5.5 Creation of the new model based on topology optimization

Based on the topology optimization, a new geometry is created (see Figure 6.9). The topology optimization model can be exported to STL format and worked with from any design program. In this work, the SpaceClaim module was used continuously and the optimisation result was coupled with the geometry of the following structural analysis (see Figure 6.4). The new design is shown in Figure 6.10.

Figure 6.9 Model based on topological optimisation

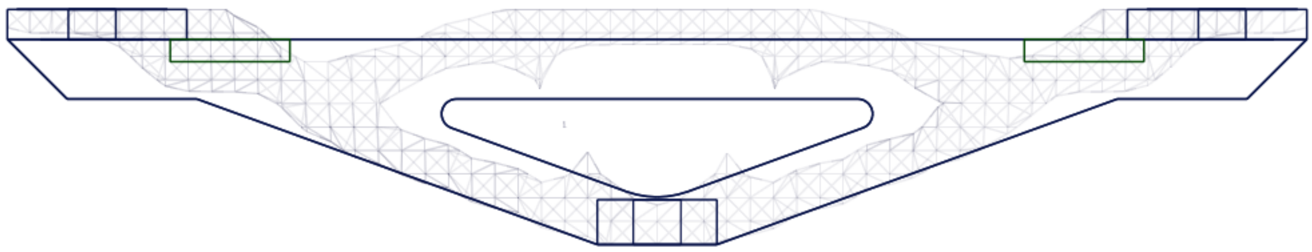
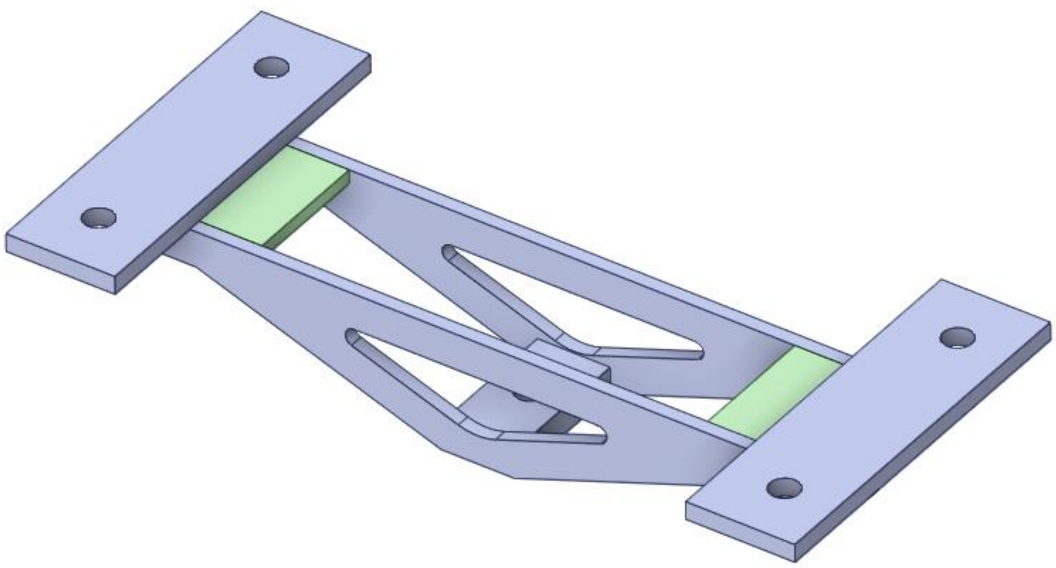


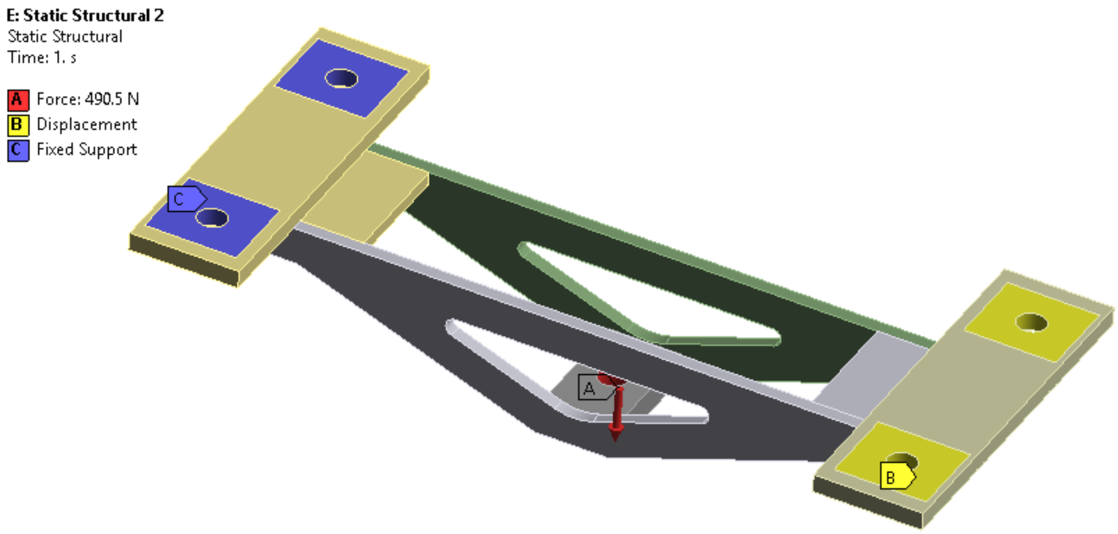
Figure 6.10 New model



6.5.6 Structural analysis of the new design

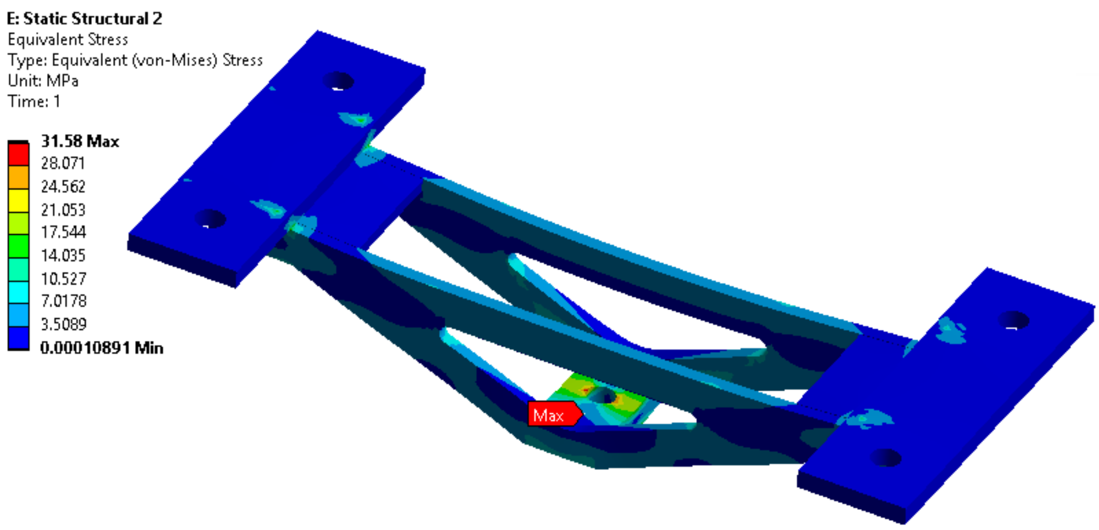
The same process as in section 6.5.3 for the structural analysis is followed again. Only now surfaces are selected instead of edges and lines for the boundary conditions (see Figure 6.11).

Figure 6.11 Model boundary conditions



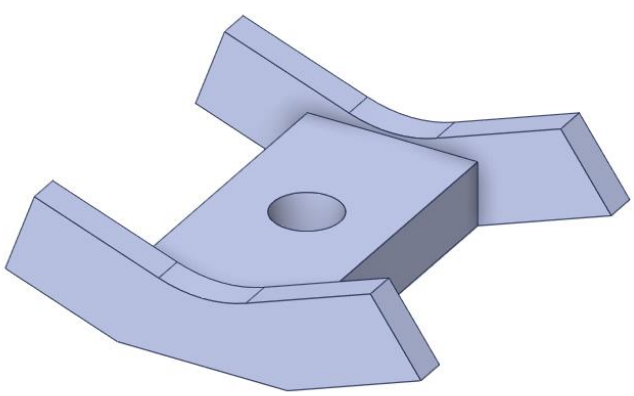
The static failure theory used is that of von Mises, as this is a ductile material. The highest indicated stress is at the bottom near the borehole (see Figure 6.12). However, this may change with the mesh size. So, a mesh sensitivity analysis is performed. The drawback that can occur is the limit of nodes that can be solved by the student licence. There are several techniques that can be used to reduce the computational cost such as symmetry and sub-modelling.

Figure 6.12 von Mises Stress



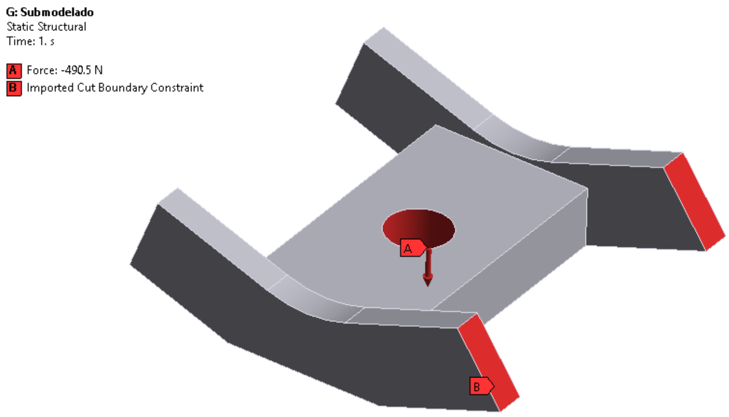
Sub-modelling consists of selecting the region of the geometry where further refinement is required. It must take into account the area where it cuts. It must not have stresses that change drastically. The solution of the static study has to be coupled with the setup of a new structural study (see Figure 6.4). The geometry of the new study corresponds to the refinement region (see Figure 6.13).

Figure 6.13 Geometry for sub-modelling



For the boundary conditions, the constraints in the shear zone are imported and the vertical load is applied to the borehole (see Figure 6.14).

Figure 6.14 Sub-model boundary conditions



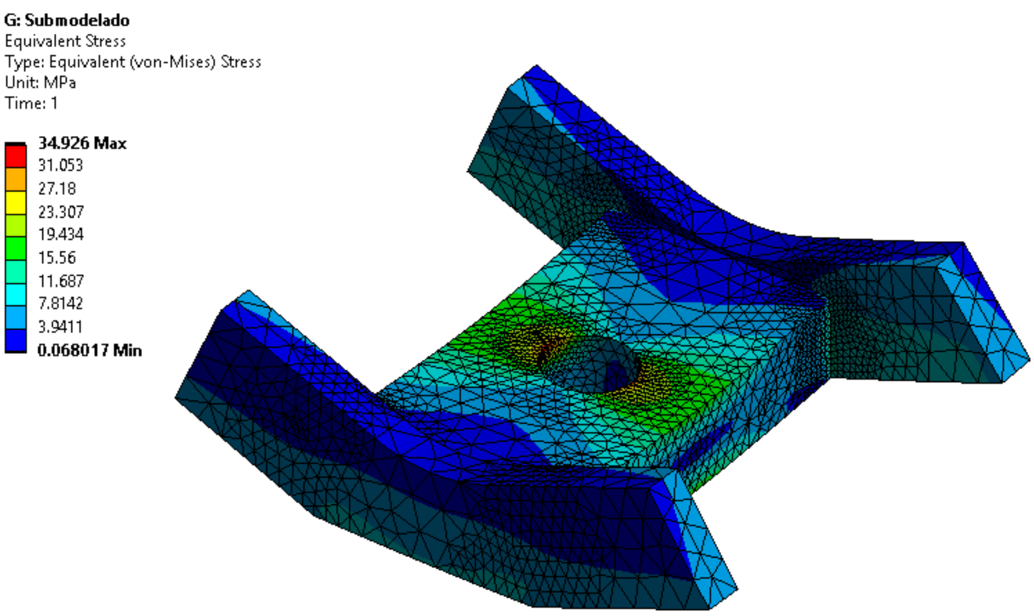
The simulation run is continued and the von Mises forces are obtained. Then, the convergence condition is appended to the von Mises results. It is indicated that the results should not vary by more than 5 %. You can adjust this value according to your requirements. This value indicates that if the stress results of each run with mesh refinement are within the range the simulation stops. The results are said to converge. Table 6.1 shows this process with the number of nodes required. A considerable increase in the number of nodes is observed which implies a longer calculation time.

Table 6.1 Mesh sensitivity analysis

Corrida	von Mises (MPa)	Change (%)	Nodes	Elements
1	31.822		8816	5206
2	34.35	7.6407	48744	32369
3	34.926	1.6616	126821	87808

The result indicates that the maximum von Mises stress is 34.9 MPa (see Figure 6.15). This is below the yield stress of the material. Therefore, it will not fail under static loading. The next study is the fatigue analysis. Since the component is subjected to cyclic loading.

Figure 6.15 von Mises Stress in the sub-model

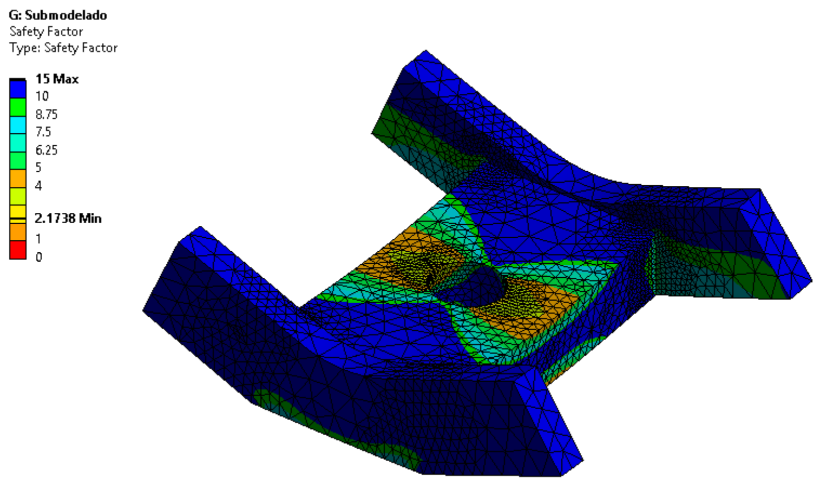


6.35.7 Fatigue study

The fatigue failure theory used is Goodman's (Equation 6). The stress ratio is 0 "Zero based". A correction factor for surface finish, size and reliability of 0.48 is considered. Infinite life is sought. Under these conditions, the fatigue safety factor is obtained (see Figure 6.16).

The minimum safety factor considered for this component is 2 as it is subjected to dynamic loads. The factor of safety obtained from the fatigue study is 2.17. The results indicate that it will not fail due to fatigue.

Figure 6.16 Fatigue safety factor



6.5.8 Modal analysis

The component must not operate at its natural frequency as this would result in high stresses that would compromise the structural integrity of the component. For this reason, these values are determined. Figure 6.17 and 6.18 show the first two modes of vibration. The first natural frequency is 596 Hz and the second is 1350 Hz.

Figure 6.17 First vibration mode

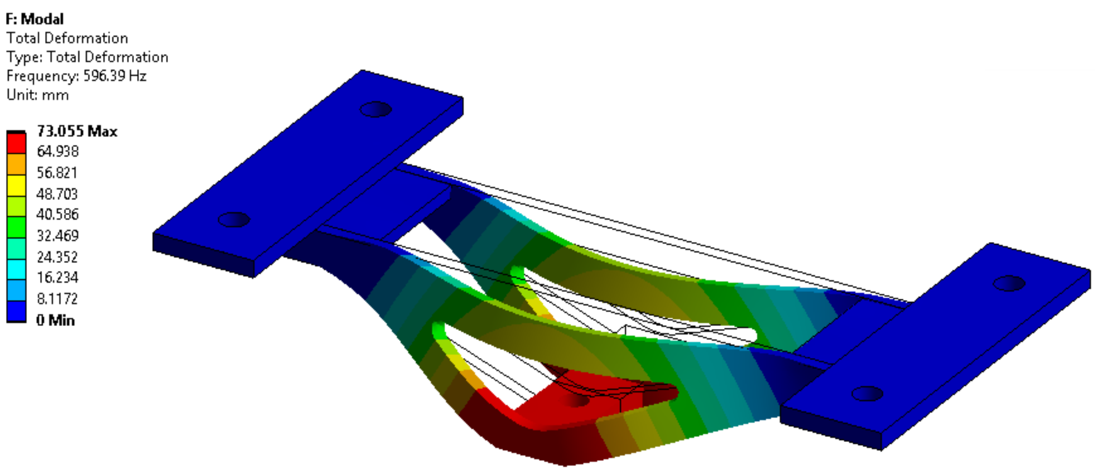
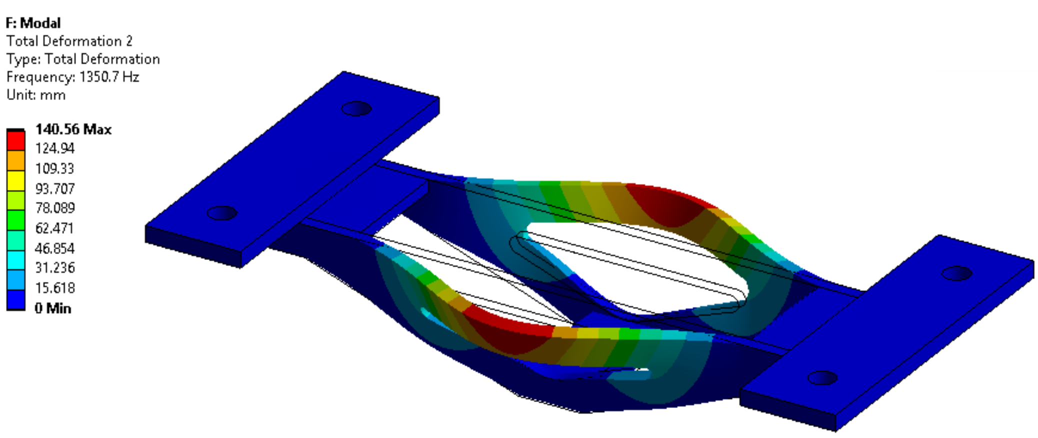


Figure 6.18 Second vibration mode



6.6 Analysis of results

6.6.1 Deformation

Structural and fatigue analyses are performed on the guide plate (original component). The boundary conditions used are the same as those used in the previous sections. Figure 6.19 shows the deformation of the plate which is at the centre and has a maximum value of 0.0166 mm. Figure 6.20 shows the maximum deformation of the new model which is 0.0153mm. Both results are very similar.

Figure 6.19 Plate deformation

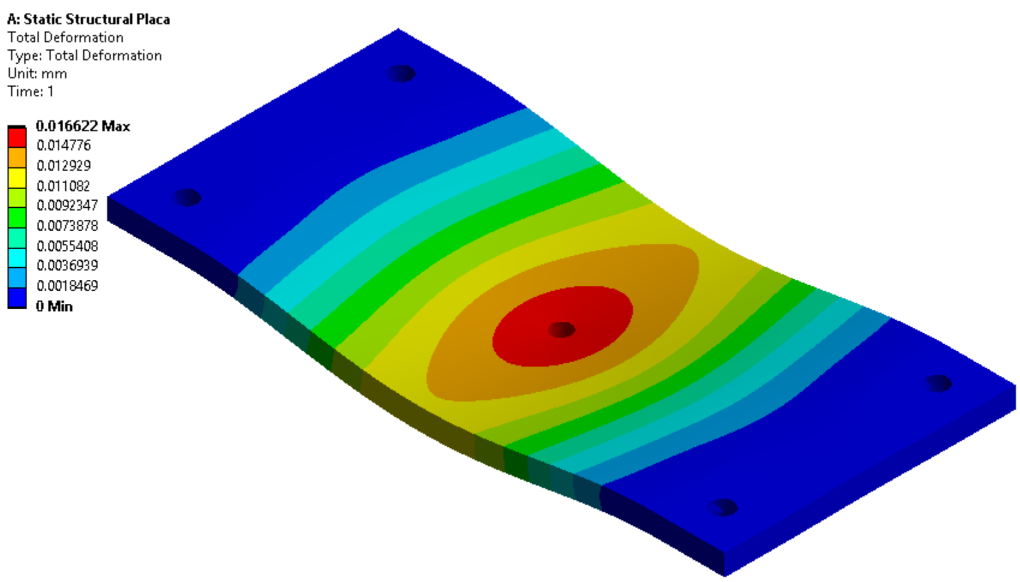
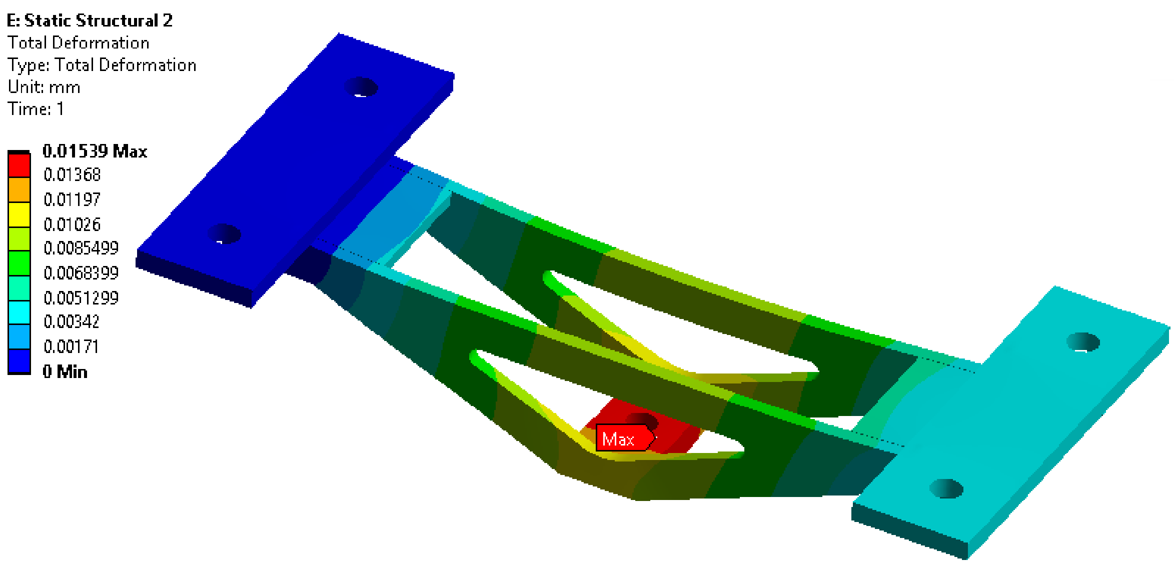


Figure 6.20 Deformation of the new model



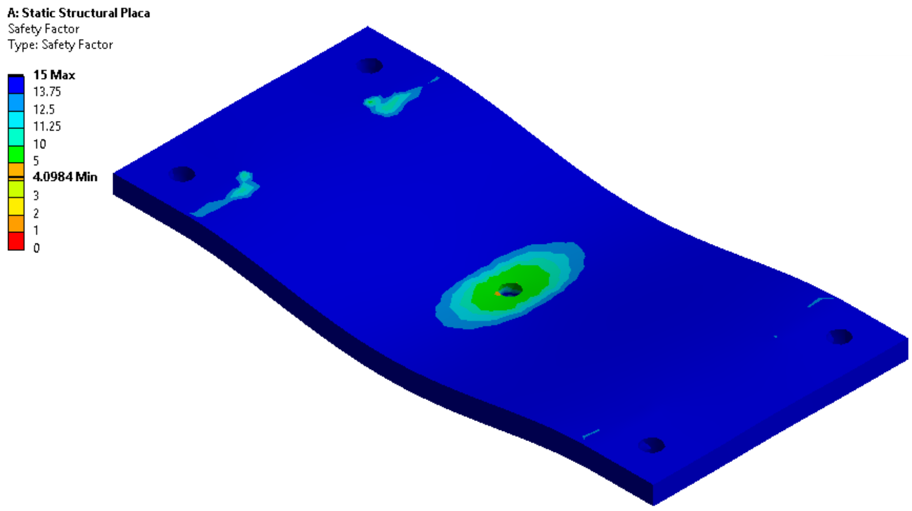
6.6.2 Mass

On the other hand, if we analyse the mass of each component we have for the plate a mass of 2.65 kg and for the new model a mass of 1.05 kg. As can be seen there is a considerable variation. This indicates that the new model is much lighter. The weight of the new design represents 39 % of the original model.

6.6.3 Fatigue

Figure 6.21 shows the minimum fatigue safety factor of the plate of 4.09 which is higher than that shown in Figure 6.16 of 2.17. Therefore, the fatigue strength of the plate is higher. However, the new design meets the required factor of safety.

Figure 6.21 Fatigue plate safety factor



6.6.4 Natural frequencies

Figure 6.22 and Figure 6.23 show the first vibration modes of the plate which are: 1007 Hz and 1556 Hz. Both are high. In the proposed model the first natural frequency is reached at 596 Hz. This is lower than that of the plate. Even so, it is still high as it corresponds to about 35760 rpm and the two-pole AC motor has an approximate speed of 3600 rpm.

Figure 6.22 First plate vibration mode

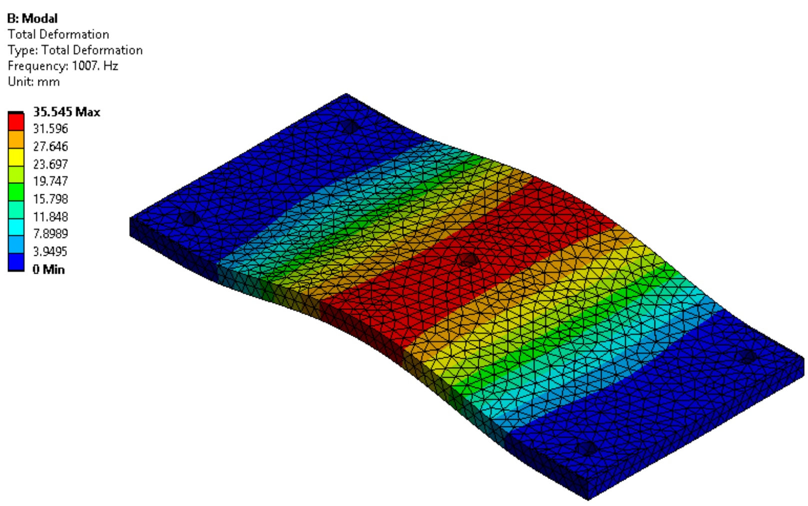
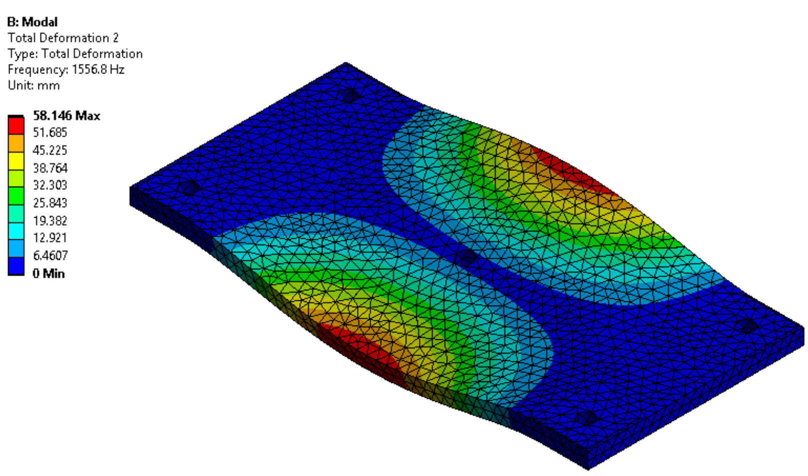


Figure 6.23 Second vibration mode of the plate



6.7 Conclusions

The same analyses were carried out under the same loading conditions for both designs. The differences in deformation are very similar and can be considered negligible (0.0166 - 0.0153 = 0.0013mm). As for the fatigue safety factor, it is noticeable. It can be said that the plate is in excess and the new design complies with the required safety factor. On the other hand, the modal analysis indicates that the new design reaches its first natural frequency (576 Hz) before the plate. This is still a high value compared to the revolutions of AC motors. Therefore, the redesign will not have any problems. The variable that was noticeable between the two models was the change in mass. The redesign contains a mass of 39 % with respect to the board (original model). Optimisation by topology is very useful if you need to find the least amount of material to create a component. However, it can give complex shapes. Manufacturing processes must be taken into account to adjust and build a new design. Manufacturing costs must be assessed as the weight of the material is not the only factor in this.

6.8 References

Budynas, R., & Nisbett, K. (2019). *Shigley's Mechanical Engineering Design*. McGraw-Hill Education.

Castagneto, L. A. (2020). *Escuela Superior Politécnica De Chimborazo*. Escuela Superior Politécnica De Chimborazo, Ecuador.

Chen, X., & Liu, Y. (2018). *Finite Element Modeling and Simulation with ANSYS Workbench* (2nd ed.). CRC Press. https://www.routledge.com/Finite-Element-Modeling-and-Simulation-with-ANSYS-Workbench-Second-Edition/Chen-Liu/p/book/9781138486294

Dowling, N., Calhoun, C., & ARCARI, A. (2009). Mean Stress Effects in Stress-life Fatigue and the Walker Equation. *Fatigue & Fracture of Engineering Materials & Structures*, *32*, 163–179. https://doi.org/10.1111/j.1460-2695.2008.01322.x

Download Ansys Student | Workbench-based Simulation Tools. (2021). https://www.ansys.com/academic/students/ansys-student

Fatigue Test. (2021). https://www.testresources.net/applications/test-types/fatigue-test/

Howard, C. Q., & Cazzolato, B. S. (2015). *Acoustic analyses using MATLAB® and ANSYS®*. CRC Press.

Londoño, B. S. (2019). *Diseño y construcción de banco de pruebas para estudios de fatiga en flexión rotativa*. Universidad de los Andes, Colombia.

Los accidentes aéreos que revolucionaron el diseño de los aviones - BBC News Mundo. (2014). https://www.bbc.com/mundo/noticias/2014/04/140421_accidentes_avion_seguridad_vert_fut_rg

Lukač, D. (2011). New dimensions in the CAE/CAD standardisation and certification process in the industrial and the educational sector. *International Journal of Knowledge and Learning*, *7*(1–2), 145–155. https://doi.org/10.1504/IJKL.2011.043897

Mott, R. L., Vavrek, E. M., & Wang, J. (2018). *Machine elements in mechanical design* (6th ed.). Pearson.

Olivera, M. F. (2019). *Diseño y construcción de máquina de ensayos flexo rotativa para el estudio de la fatiga mediante el método esfuerzo vida para la sede valle Jequetepeque de UNT*. Universidad Católica Santo Toribio de Mogrovejo, Chiclayo.

Simbaña, R. M., & Chango, D. M. (2012). *Diseño y construcción de una máquina para realizar ensayo de fatiga por flexión rotativa utilizando PC*. Escuela Politécnica del Ejército Extensión Latacunga.

Ugural, A. C. (2015). *Mechanical Design of Machine Components* (Taylor & Francis Group (Ed.); 2nd ed.). CRC Press.