Design of Machines and Structures, Vol. 11, No. 1 (2021), pp. 34–38.
https://doi.org/10.32972/dms.2021.004

CASE STUDY: DESIGN OF A VACUUM GRIPPER

DÁNIEL KISS[1] – ATTILA SZILÁGYI[2]

[1, 2]*University of Miskolc, Department of Machine Tools*
3515 Miskolc-Egyetemváros
[1]*kiss.daniel@uni-miskolc.hu*
[2]*szilagyi.attila@uni-miskolc.hu*

Abstract: In this paper we present the design process of a vacuum gripper based on an existing design, which have to be modified. During the design process each step was analysed by finite element methods, to see that the change in the model was made into the right direction. During these steps an appropriate solution was selected to be used later. In the paper we present the base design and the results of its analysis. Later we discuss the modifications made on the model and the final result of simulations of the modified geometries.

Keywords: *FEM, mechanical, numerical*

1. INTRODUCTION

When designing machines, all factors related to the accuracy and in-service parameters of the machine must be taken into account. These factors include deformations and vibrations caused by certain thermodynamic and mechanical phenomena. The effect of these should be taken into account already in the planning phase, since in this case it is the most economical to detect errors. Since the device is not physically available in the design phase, these phenomena can be modelled using simulation software [1].

The part to be examined is a vacuum gripper which have to be modified based on an existing design. The modification is necessary because it have to hold in place bigger pieces, this also means that the structure have to be optimized for minimal deflection caused by the self-weight of the structure [2], [3].

This griper holds in place different types of foils, during other operations done. The existing design is made for 600 mm wide foils, while there is a demand for holding wider pieces up to around 1,200 mm in width. The paper does not discuss the effect of the vacuum, because that has more effect on the foil rather than on the structure of the gripper.

The material used for this device was selected to be EN-AW 2014-T6 aluminium alloy, which has good mechanical properties, and easy to machine. The design and analyses was done in Autodesk Inventor 2020 design software. This software has a FEM module which can be used for simpler engineering analyses.

2. DEFINING THE MODELS

The assembly of the gripper consists of 4 pieces:

- vacuum plate
- side supports (2pcs)
- middle support

In the beginning simple geometries were made for the support, and vacuum plates, which were modified during the design phase. The initial design is shown in *Figure 1*. In the figure the fixed constraints (white squares) and gravity load (yellow arrow) can be also seen. The middle support only attached to the lower side of the vacuum plate. The notch between the two plates have to be left free accessible. This causes the problem, that the upper side of the plate cannot be supported in the middle which will cause uneven deformation between the two holding plane.

During analysis the automatic mesh generator was used, which uses tetrahedral elements (359,236 elements and 235,247 nodes for the assembly). The element size is defined by a fraction of the bounding box of each part. Since the parts will be assembled by screws, the contact regions were modelled as rigid (bonded) contact. The material definition is the built in properties in the software of the mentioned alloy.

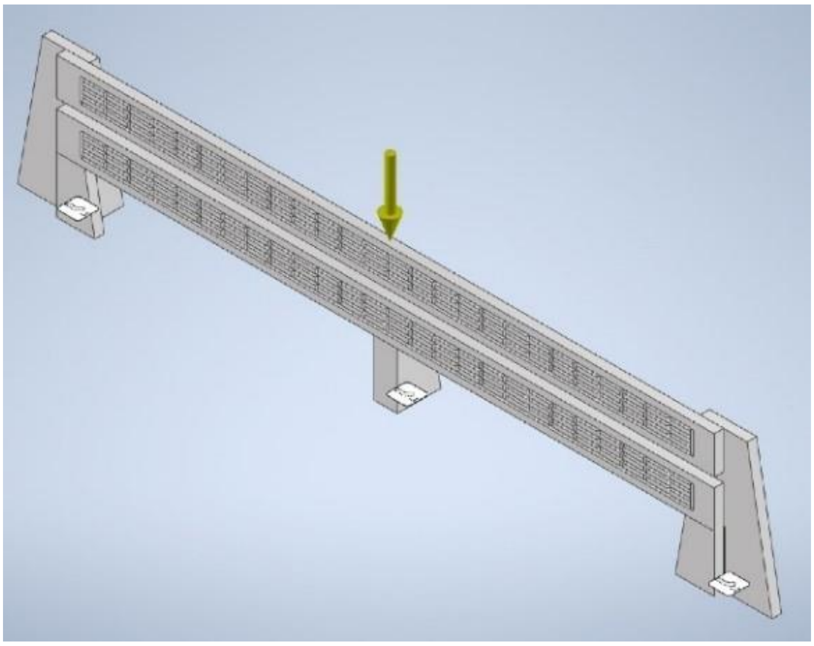

Figure 1. *Starting design showing 2 separate vacuum plates, which was later united*

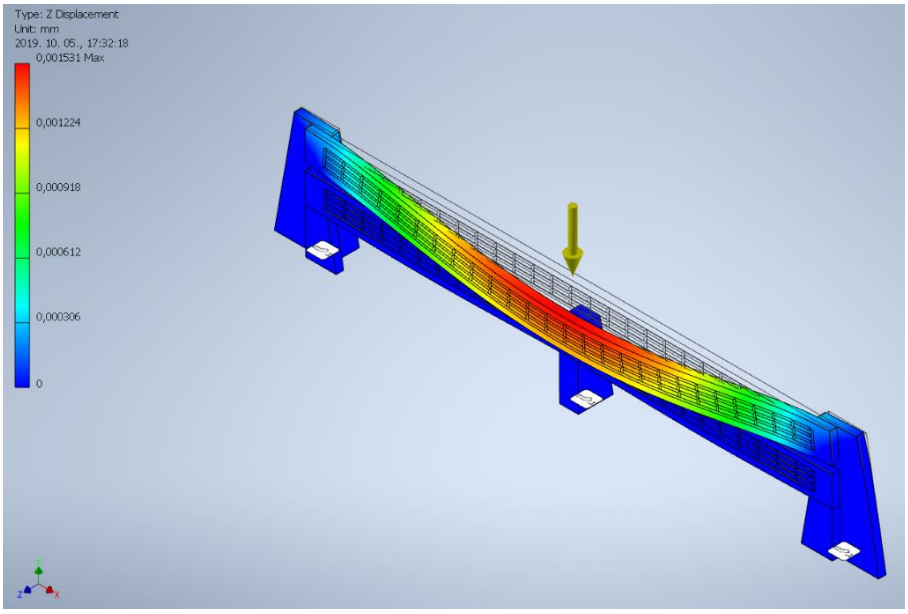

Figure 2. *Deflection in transverse direction (Z) (enlarged)*

The deflection of the initial model is shown in Figure 2. Buckling can be seen in Z direction, which is inadequate, because it causes the foil also to move off the desired plane. During the next steps these parts have to be optimised to get as low deflections as possible.

Figure 3 shows the cross section of the optimised vacuum plate.

Figure 3. *Cross section of the vacuum plate*

3. MODIFICATIONS IN THE GEOMETRIES

During the model optimization the support was redesigned first. Since the vacuum plates had to be redesigned also, these modelling steps were interacted to each other. The cross section of the vacuum plates were also redesigned in multiple steps until the optimal solution was found. In conclusion of the performed analyses we can state, that using thicker support, the deflection of the upper plate does not change significant, therefore an optimal solution was selected, with simple design and reduced weight, which is important for higher eigenvalues of the assembled device.

In *Figure 4* we can see the deflection of the optimized geometry in transversal direction. During several modification in the design, we succeeded to achieve a solution where the transversal deflection is under 1 μm. For precise operation of the device this value is adequate.

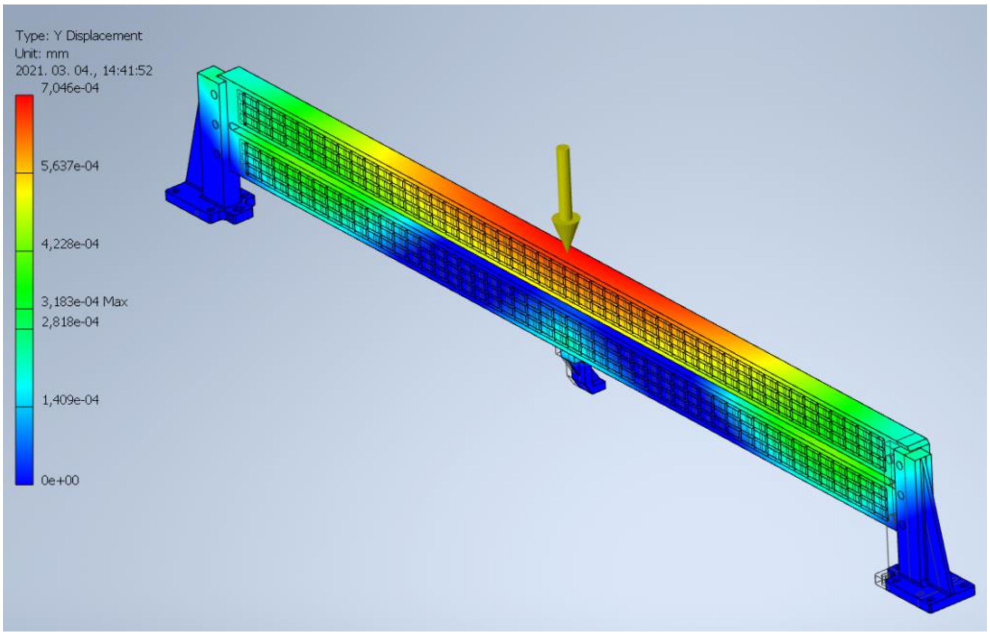

Figure 4. *The meshed model with contact definitions*

4. DEFINING EIGENVALUES

During operations it is advised to design the device in that way, that the self-frequencies of the device should be over the vibrations that are present from the environment. These frequencies are typically in the range of 1–150 Hz. In addition to the static stiffness, the smallest eigenfrequency value of the modified and already statically optimized structure have to be determined. We therefore considered it reasonable to design the structure so that its minimum eigenfrequency falls upwards from the referenced frequency range.

Table 1

The result of the modal analysis

Mode	Frequency (Hz)
1	88.42
2	188.38
3	217.84
4	274.32

Based on the data obtained as the result of the calculation *(Table 1)*, we can state that the smallest natural frequency of the examined structure is only in the frequency range mentioned. Since these parts will be mounted on an optical table, with dampening pads, this natural frequency will not cause deflections and unwanted vibration in the device, because one property of these optical tables and its supports is that these can eliminate unwanted frequencies from the environment with good efficiency.

5. SUMMARY

In the present article, we have depicted the modification of an existing geometry using numerical methods to verify the effects of each modification. We draw the conclusion from the results of the simulations for what have to be modified to get optimised geometry in accordance with the deflection caused by self-weight. The eigenvalues of the structure was also defined, despite the lowest value is in the range of the environmental frequencies, by the usage of other dampening devices, it will not cause any undesired phenomenon.

ACKNOWLEDGEMENT

The described study was carried out as part of the EFOP-3.6.1-16-00011 *Younger and Renewing University – Innovative Knowledge City – institutional development of the University of Miskolc aiming at intelligent specialization* project implemented in the framework of the Szechenyi 2020 program. The realization of this project is supported by the European Union, co-financed by the European Social Fund.

REFERENCES

[1] Baráti, A.: *Szerszámgép-vizsgálatok*. Műszaki Könyvkiadó, Budapest, 1988.

[2] Páczelt, I.: *Végeselemmódszer a mérnöki gyakorlatban*. I. kötet, Miskolci Egyetemi Kiadó, pp. 36–37.

[3] Kiss, R., Szilágyi, A.: Analysis of the dynamic behaviour of the CNC machine centre by FEM. *Design of Machines and Structures*, Vol. 9, No. 1, pp. 24–28, 2019.