# RESEARCH TO OPTIMIZE PRODUCT SHAPE USING AUTODESK INVENTOR PRO SOFTWARE

Nguyen Nhat Duy*

*Dong Nai Technology University*

*Corresponding author*: Nguyen Nhat Duy, nguyennhatduy@dntu.edu.vn*

**GENERAL INFORMATION**

Received date: 16/03/2024

Revised date: 11/05/2024

Accepted date: 26/07/2024

**KEYWORD**

*FEM;*

*Stress analysis;*

*inventor software;*

*Shape optimization;*

**ABSTRACT**

Reducing the weight of aesthetically pleasing machine parts is one of the biggest trends today. Applications and variations of the part are determined using the *Finite Element Method* (FEM). Shape optimization is a method by which lighter details can be obtained while preserving the safety and durability of components. This article deals with optimizing the shape of a bearing plate commonly found in mechanical engineering. The verification of the suitability of the newly optimized model will be done on the basis of a static simulation of the shape using Inventor Pro 2023 software. Optimum results in lighter weight detail, higher aesthetics while ensuring durability. This optimization ensures novelty because it has only been integrated into inventors recently and continues to be developed.

## 1. INTRODUCTION

Shape optimization is one of the design methods that has evolved rapidly in recent decades. The main goal of shape optimization is to find the optimal material layout on a specific design domain with given edge conditions.

This method is often used to improve the overall features such as rigidity, deformation and external force resistance of the machine part (Lin D. , 2013). Shape optimization methods are divided into the following three categories: size optimization, shape optimization, and shape optimization. In typical size optimization, the aim is to find the optimal thickness of the shape. On the other hand, the goal of shape optimization is to find the optimal shape of the texture (Kang, 2014). Shape optimization involves identifying features such as shape, position, size, and hollow positions in the shape (X-duan, 2002).

The main goal of shape optimization is to find the optimal layout of a shape in a defined area (RONG, 2022). The input parameters when designing are the applied load, the structural mass, and the design constraints such as the location and size of some zones that are not allowed to change when optimized (Po, 2016).

## 2. RESEARCH METHODOLOGY

In this paper, the optimization will focus on the shape of a bearing plate that has been designed using conventional methods. Using Inventor Pro 2023 simulation software, the new bearing plate shape design based on edge conditions aims to reduce bearing plate weight.

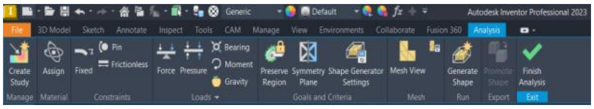


**Figure 1.** Shape optimization analysis tool table

Autodesk Inventor provides tools to determine structural design performance (shape engine) directly on the Autodesk Inventor Shape Generator model. Stress analysis includes tools for placing loads and constraints on a part or assembly and calculating stress, strain, safety coefficient, and resonant frequency mode.

With stress analysis tools, we can:

– Perform structural, static, or modal analysis of a tool part or tool assembly.

– Apply force, pressure, bearing load, torque, or body load to the top, face, or edge of the tool model, or import the motion load from the dynamic simulation.

– Apply fixed or non-zero displacement constraints to the tool model.

– Model the different mechanical contact conditions between adjacent tool parts.

– Evaluate the impact of multiple parametric design changes.

– View analysis results for equivalent stress, minimum and maximum principal stress, strain, safety factor, or method frequency.

– Add or block features such as building parts, design re-evaluations, and solution updates.

– Animate the model through different stages of distortion, stress, safety factor, and frequency.

– Generate complete and automated technical design reports of the visualization tool in HTML formatting.

The stress analysis environment allows for the analysis of assembly or part design and the evaluation of different options quickly under different conditions, e.g. the use of different materials, loads and constraints (boundary conditions), etc.

## 3. RESULTS AND DISCUSSION

### 3.1. The product needs to be optimal, materials and border conditions

#### *3.1.1. Introduce*

**Original Product**

In the study, the Inventor Pro 2023 software was used to simulate a bearing frame (Figure 2) that is stressed by two forces in different directions. The weight and volume of the bearing plate are calculated to be 5.79e-3 kg and 0.739 cm before optimization.

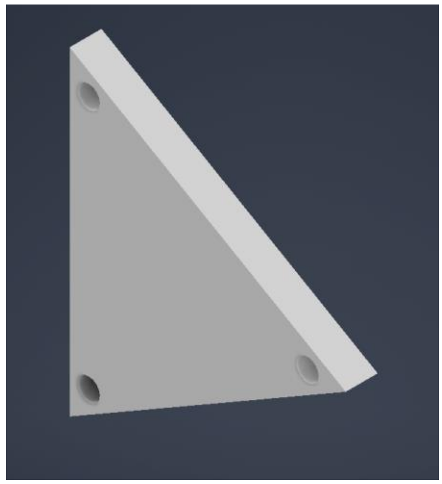

**Figure 2.** Original Bearing Frame Products

#### *3.1.2. Product Materials and Edge Conditions*

The bracket is made of steel material. The mechanical properties of the material are presented in Table 1. The optimization of the shape of the bearing plate will be carried out on the basis of simulating a static problem with the use of Inventor Pro 2023 simulation software. The applied load is two forces with magnitude Fx = 50N and Fy = 100N shown in (Figure 3). 6 degrees of freedom binding is applied at holes A and B to fix the bearing plate during the simulation.

**Table 1.** Mechanical properties of the material

| Specific Volume [kg/cm$^3$] | Flow Stress [MPa] | Elastic Module [GPa] | Coefficient Poisson |
|---|---|---|---|
| **7800** | 235 | 207 | 0.3 |

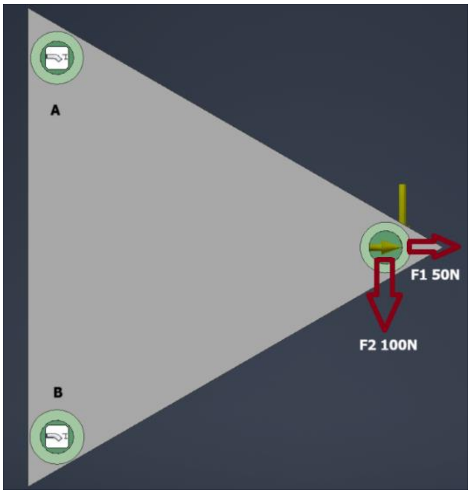


**Figure 3.** Model border conditions

### 3.2. Simulate and optimize product shapes

The finite element method (FEM) is often used as part of a two-stage design process for size, shape, and shape optimization problems. In the simulations, the tetrahedral element is used with a size of 0.1 mm in the model meshing (Figure 4).

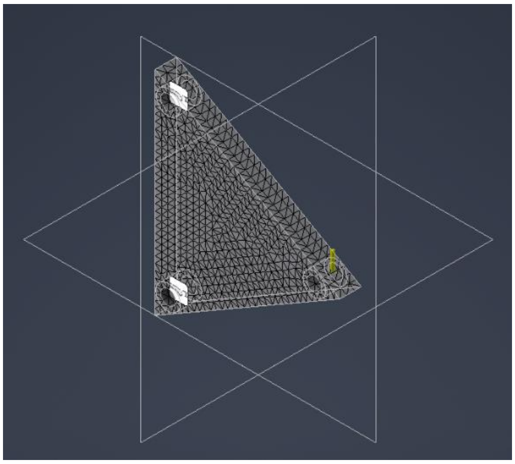

**Figure 4.** Detailed gridding

**Table 2.** Product Meshing Parameters

| | |
|---|---|
| **Avg. Element Size (fraction of model diameter)** | **0.1** |
| **Min. Element Size (fraction of avg. size)** | 0.2 |
| **Grading Factor** | 1.5 |
| **Max. Turn Angle** | 60 deg |
| **Create Curved Mesh Elements** | Yes |

The process of shape optimization is carried out in several cycles, starting with the meshing of the shape of the object to be optimized, conducting an analysis of the object's workability based on the finite element method, changing the density of the elements based on the defined optimization algorithm, and repeating the process with a new mesh based on the newly formed shape. The design space during the TO process is divided into two main areas. The first zone represents the "design space", namely the location of elements whose density is allowed to be modified (added or subtracted) when optimized, while the second zone represents the "non-design space", namely the zone consisting of elements that cannot be modified. (Figure 5) shows that the parts that can be optimized are highlighted in red, and those that are not applicable to the optimization process are highlighted in light green.

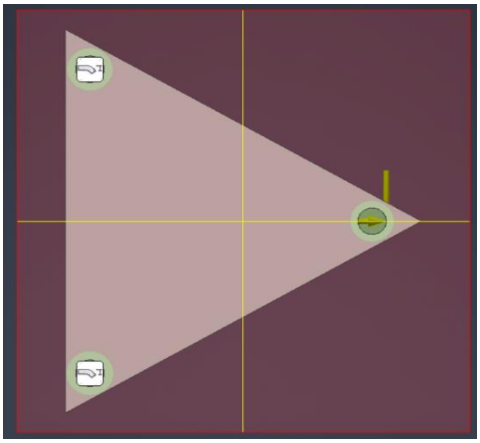

**Figure 5.** Optimal surface (covered with red block), non-optimal (covered with light green block).

The goal is to optimize the shape and weight of the bearing plate by adding or removing materials to elements in the optimized surface to achieve the desired design. In which, the volume is retained at least 50% of the volume compared to the original design of the model (Figure 6).

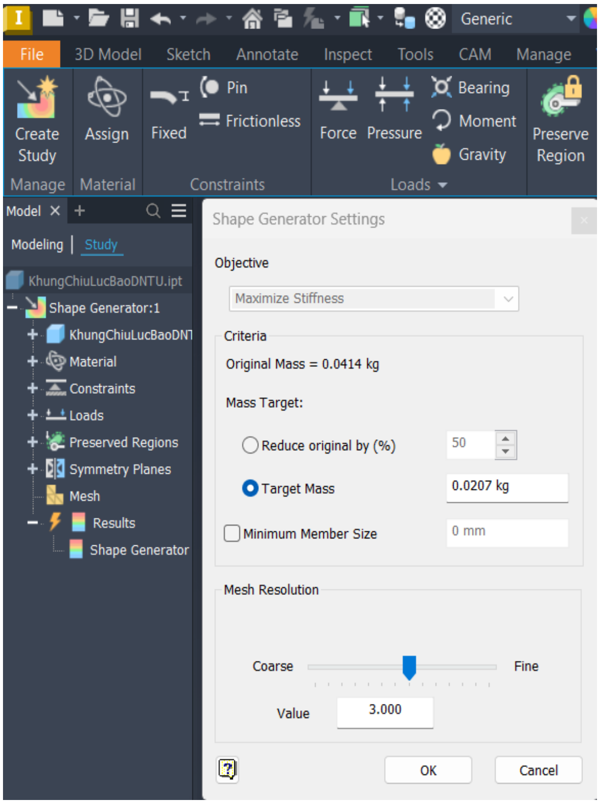


**Figure 6.** Optimal parameter setting

Shape optimization is finding the optimal material distribution function that satisfies constraints throughout the design domain. The goal of shape optimization is to minimize the deformation energy of the design shape, and the binding function is the density of the material (volume). A variety of combinatorial textures can be considered to obtain the optimal texture shape that satisfies the objectives and constraints with the help of developing multiple shape optimization algorithms. Evolutionary optimization diagrams are another method being studied to achieve shape optimization results (Figure 7).

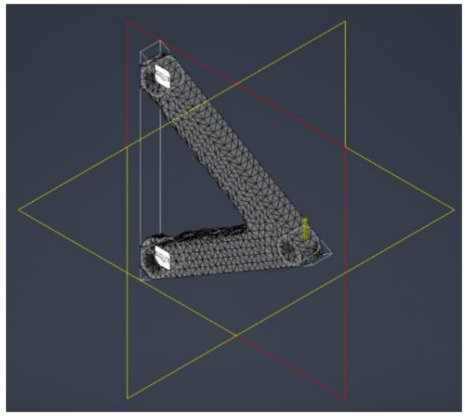

**Figure 7.** Optimal shape analyzed

For each calculation cycle, the optimization process includes: creating new material properties and elements during the shape optimization process; modify the coordinates of the nodes during shape optimization in order to create a new model; redesigning the product shape based on the optimal Inventor software shape (Figure 8); analyze the newly created model and record and evaluate the analysis results.

**Figure 8.** New product redesigned for optimal shape

### 3.3. Testing the new product adaptation analysis

The output of a math solver is generally a significant amount of raw data. This amount of raw data would often be difficult and tedious to explain without the traditional data arrangement and graphical representation known as post-processing. Post-processing is used to create a graphical display that shows the stress distribution, deformation, and other characteristics of the tool model.

The outline color displayed in the results corresponds to the range of values displayed in the commentary. In most cases, the results displayed in red are of the most interest, as they exhibit high stress or high deformation or low safety factor.

Stresses (Von Mises), to represent these multidimensional stresses is to summarize them into an equivalent stress, also known as von Mises stress $\sigma_v$.

$$\sigma_v = \sqrt{\frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2}} \qquad (1)$$

In which:

$\sigma_v$ is the von Mieses stress in MPa,

$\sigma_1$, $\sigma_2$, $\sigma_3$ is the main stress in MPa.

Von Mises stress results use colored outlines to show us the stresses calculated in the solution for our drawing tool model.

Re-analyze the stress according to the new shape to check the required guaranteed detail (Figure 9).

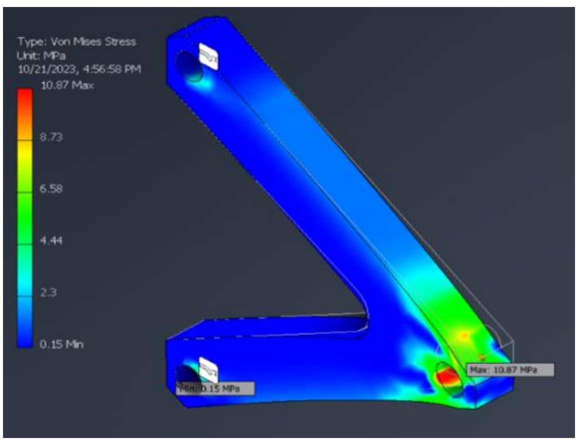


**Figure 9.** New Product Stress Analysis

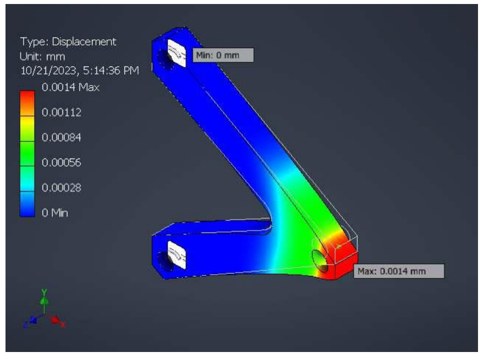


**Figure 10.** New Product Displacement

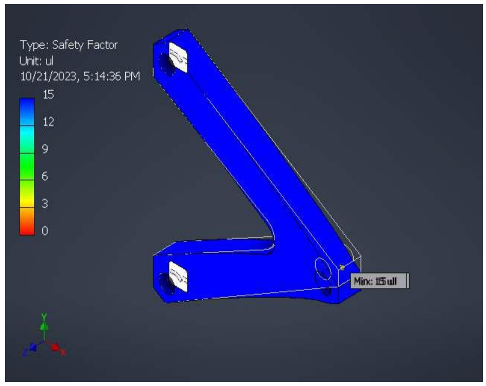


**Figure 11.** New Product Safety Factor

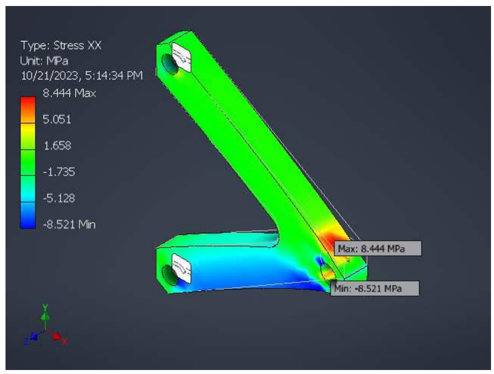


**Figure 12.** New Product Safety Stress

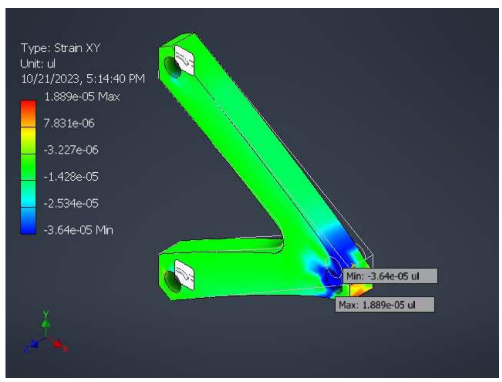


**Figure 13.** New Product Strain

**Table 3**. Analysis Results

| Name | Minimum | Maximum |
|---|---|---|
| **Volume** | 2820.4 mm^3 | |
| **Mass** | 0.022 kg | |
| **Von Mises Stress** | 0.15 MPa | 10.87 MPa |
| **1st Principal Stress** | -3.22 MPa | 10.75 MPa |
| **3rd Principal Stress** | -10.10 MPa | 2.61 MPa |
| **Displacement** | 0 mm | 0.0014mm |
| **Safety Factor** | 15 ul | 15 ul |
| **Stress XX** | -8.52 MPa | 8.45 MPa |
| **Stress XY** | -5.88 MPa | 3.05 MPa |
| **Stress XZ** | -1.07 MPa | 1.49 MPa |
| **Stress YY** | -7.45 MPa | 8.26 MPa |
| **Stress YZ** | -0.92 MPa | 0.97 MPa |
| **Stress ZZ** | -3.23 MPa | 3.11 MPa |
| **X Displacement** | -0.00022 mm | 0.00023 mm |
| **Y Displacement** | -0.00140 mm | 0.00001 mm |
| **Z Displacement** | -0.00002 mm | 0.00003 mm |
| **Equivalent Strain** | 0.00001 ul | 0.00004 ul |
| **1st Principal Strain** | 0.00001 ul | 0.00005 ul |
| **3rd Principal Strain** | -0.00005 ul | -0.00001 ul |
| **Strain XX** | -0.00003 ul | 0.00003 ul |
| **Strain XY** | -0.00003 ul | 0.00002 ul |
| **Strain XZ** | -0.000006 ul | 0.000009 ul |
| **Strain YY** | -0.00002 ul | 0.00003 ul |
| **Strain YZ** | -0.000005 ul | 0.000006 ul |
| **Strain ZZ** | -0.000010 ul | 0.000009ul |

The stress analysis results show that the maximum stress result is 10.87 MPa as shown in (Figure 9). The blue stress zone has a fairly small stress of 0.15MPa, which is negligible compared to the flow limit of the material used. Displacement parameters comply with the conditions of (Figure 10). Safety factor complies with the conditions of (Figure 11). And many images show the stress satisfying the conditions as in (Figure 12) and (Figure 13).

The analysis results show that the maximum stress value has not yet exceeded the load capacity for the material.

Some important things to consider when performing shape optimization:

(1) Meshing: Shape optimization results when analyzing finite elements can be affected by the mesh density of the shape. Therefore, it is worth considering how different types of meshes affect the analysis of the shape being designed.

(2) Minimum Element Size: Shape optimization can produce a very small shape thickness by finely dividing the mesh when designing which is difficult to produce in practice. Thus, it is recommended to consider the feasibility of production before making a decision on the smoothness of the mesh when optimal.

(3) Fabrication feasibility: Fabrication methods according to shape features are often predetermined such as extrusion, casting, so the optimal shape is considered feasible in terms of fabrication using existing methods.

## 4. CONCLUSION

In this paper, a load-bearing plate model is designed based on the results of a static problem simulation using shape optimization in Inventor Pro 2023 software. The results of the shape optimization analysis are highly dependent on the boundary conditions. When conducting shape optimization design, the edge conditions that affect the optimized shape need to be considered and applied correctly.

In the design process of shape optimization, the optimization helps to achieve the rigidity of the structure and reduce the reasonable weight, which meets the requirements of the reliability of the shape when the design is optimized. The main goal is to reduce the volume (weight) of the bearing plate while maintaining its safe operation. The new bearing plate model created by the shape optimization module shows a reduction in volume of about 50% while the greatest stress remains within the load-bearing capacity of the material.

**ACKNOWLEDGEMENT**

Thank you very much to AUTODESK Corporation for supporting the Autodesk Inventor Pro 2023 Software License Education so that I can complete this article.

**REFERENCES**

Kang, Y. (2014). Suspension Control Arm Multi-objective Topology Optimization. *PLA University of Technology*,, vol 12.

Lin, D. (2013). *Automotive Engine Bracket Topology Optimization And Finite Element Analysis*. Hangzhou: Zhejiang University of Technology.

Po, W. e. (2016). Topology Optimization Design of Automotive Engine Bracket. *Energy and Power Engineering*,, vol.8.

RONG, Y. e. (2022). Structural topology optimization with an adaptive design domain. *Computer Methods in Applied Mechanics and Engineering*, vol.389.

X-duan, S. T. (2002). Shape optimization using FEA software-A v-shaped anvil as an example. *Journals of Material Processing technology*, vol. 12.