Journal of Physics: Conference Series

PAPER • OPEN ACCESS

Structure Analysis and Optimization Design of the Base of Four-pillar Hydraulic Testing Machine

To cite this article: Yurong Zhu *et al* 2021 *J. Phys.: Conf. Ser.* **2002** 012036

Structure Analysis and Optimization Design of the Base of Four-pillar Hydraulic Testing Machine

Yurong Zhu[1], Yan He[1*] and Jialong Ren[2]

[1] Shanghai Normal University Tianhua College, Institute of Artificial Intelligence, shanghai, 201815, P.R.China.
[2] Jiangsu University of Science and Technology, School of Mechanical and Power Engineer, Zhenjiang, Jiangsu, 212003, P. R. China
Email: jeury@sina.com

Abstract. Hydraulic testing machine is a kind of testing machine that transfers energy to implement mechanical test by liquid as medium. With the emergency requirement of energy saving and emission reducing, it becomes significant to kick off the infrastructure research to optimize the hydraulic testing machine. At present, some new products always designed with experience is without any improvement. Consider the current situation in testing machine industry,and combined with the desire of design lightweight , some job about optimization and parametric design of product was done from the angle of finite element in this paper.Following content is included: getting a more accurate finite element analysis method about the analysis of the base based on Solidworks, Hypermesh ,*ANSYS*, and optimize model of the base by Optistruct. A more reasonable model of the base with full strength is been got. Puts forward a kind of finite elements analysis for the structure of the hydraulic testing machine. Meanwhile,share shape optimization techniques through which the structure is designed more reasonable.
Keywords: Hydraulic testing machine; base; Finite elements analysis; Optimization

1. Introduction

Hydraulic testing machine is a kind of instrument which transfers energy by liquid as medium, testing and verifying whether the product meets the design requirement under pressure. Hydraulic testing machine is an important mechanical testing equipment, and it is also one of the important symbols to show the machinery manufacturing industry capability of a country. According to the design requirements of hydraulic testing machine and the increasing demand of energy saving and consumption reduction, it is an important research topic to the infrastructure research and optimization for hydraulic testing machine, as there is great practical significance to save resources and energy, improving the economic and social benefits of hydraulic testing machine.

The design of a new hydraulic testing machine and the structural improvement of an existing one are often implemented through empirical comparison. It has not good effect as expected, and is easy to bring various unexpected problems. As the sample with four column hydraulic testing machine,because the structure of base and stress of the hydraulic testing machine are very complex, it is difficult to calculate the stress and strain. Structure of the base in hydraulic testing machine is also complex that it is impossible to use the mathematical model of constraint function and objective function to optimize the iterative function. Therefore, these conventional optimization methods are useless to solve such problems.

The paper focus on the research of Hydraulic testing machine, according to the features of high precision and low speed [1-2].

(1) Stress analysis of the base, bring out the stress distribution under the maximum loading calculated by the finite element analysis.

(2) Setup the optimization plan according to the stress distribution nephogram, then optimize by using Optistruct software. On the precondition of ensuring strength of the base, there will be a more reasonable structure, saves materials and reduces production cost.

2. Stress Analysis for the Frame of four Column Hydraulic Testing Machine

2.1. Structure of four Column Hydraulic Testing Machine

In this paper, the four column hydraulic testing machine is composed of base, hydraulic cylinder, moving parts and its guiding device, crossbeam, worktable and other auxiliary devices. Its structure diagram and three-dimensional model are shown in figure 1 and figure 2 respectively. This structure can realize two main movements: first, the motor drives the crossbeam to move up and down along the column (lead screw); Secondly, the hydraulic cylinder installed in the base pushes the worktable to move up and down. Put the workpiece to be tested on the worktable before testing. After the crossbeam drops to a certain height, the hydraulic cylinder pushes the worktable upward to realize the extrusion of the device [3]. At this time, sensor collects the relevant datas of device deformation for subsequent research and analysis of the test process.

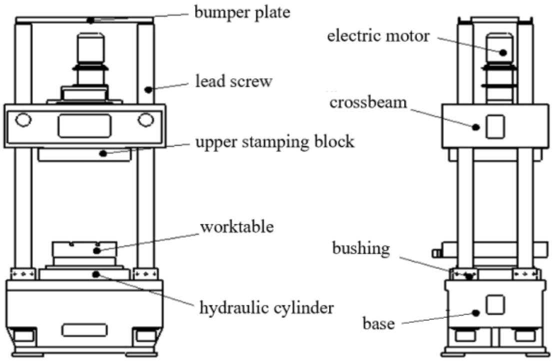

Figure 1. Diagram of hydraulic testing machine.

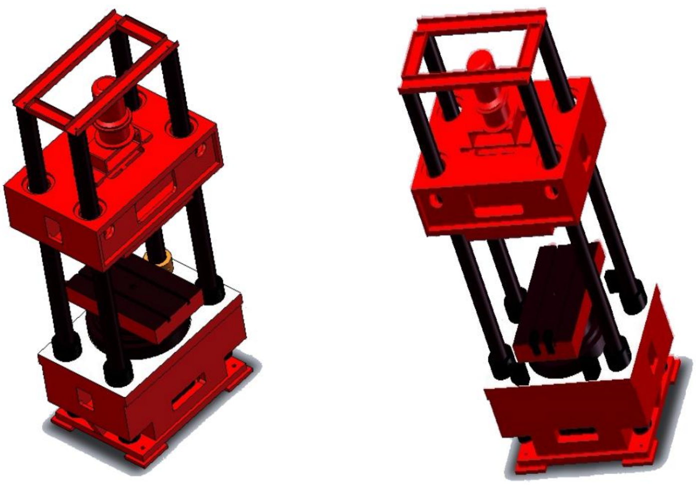

Figure 2. 3D diagram of hydraulic testing machine.

2.2. Working Conditions of four Column Hydraulic Testing Machine and Calculation the Stress of Base

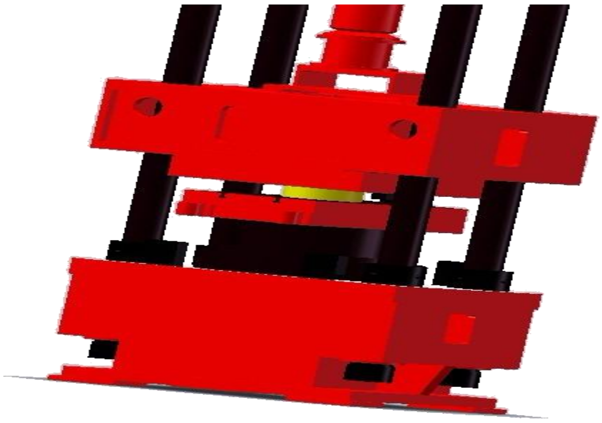

Figure 3. Working condition diagram of device under pressure test.

As shown in figure 3, during test, the workpiece placed on the worktable receives the pressure from the worktable which is come from the hydraulic cylinder on the base, and transmits the pressure to the upper beam. The height of the working position is adjusted by moving up and down the crossbeam which is sheathed on the four lead screws installed on the four corners of the frame through the snap ring.The stress of the footprint base is shown in figure 4.

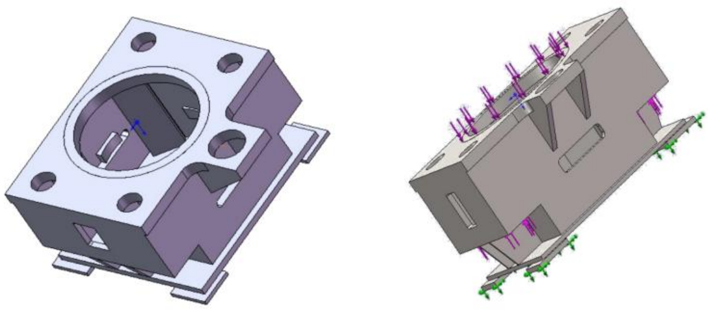

Figure 4. Stress diagram of footprint base (pink arrow for stress, green arrow for support).

If the frame is regarded as a space frame, it can be solved according to forces of the space frame without its symmetry. In order to eliminate the false and retain the true, the following assumptions should be made in the engineering calculation.

(1) 2D frame is used to replace the space frame as the front and back side of the frame are basically symmetrical.

(2) Rigid connection between the lead screw and the footprint base.

(3) Temperature stress is ignored in the calculation.

(4) Maximum force is 20KN on the hydraulic testing machine.

(5) The loading point on the frame is on the center.

Assuming that the stiffness of the cross beam and the frame relative to the lead screw is huge, the bending stress caused by the deformation of the crossbeam and the frame can be ignored, then the lead screw only bears the axial tension and the uniform force. Therefore, the axial tension of each lead screw is

$$F = \frac{P}{n} = \frac{20\text{KN}}{4} = 5\text{KN}$$

P -- nominal force of hydraulic testing machine (KN);
N --- number of columns;

3. Finite Element Analysis of the Footprint Base

The basic flow and steps of the finite element calculation of the footprint base of the hydraulic testing machine (see figure 5) can be summarized as follows [4]:

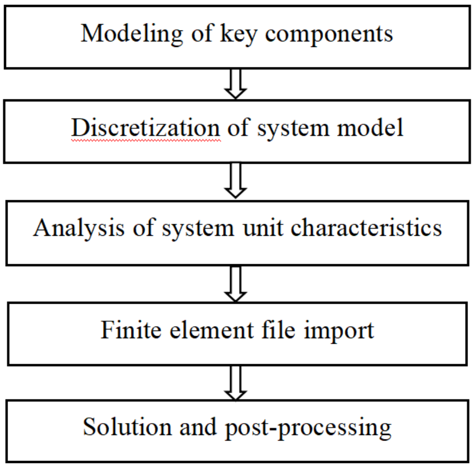

Figure 5. Calculation flow chart of hydraulic testing machine.

3.1. Four Column hyDraulic Test and Model Establishment of Base Structure

SolidWorks CAD system is used to create 3D geometric model, as shown in figure 6.

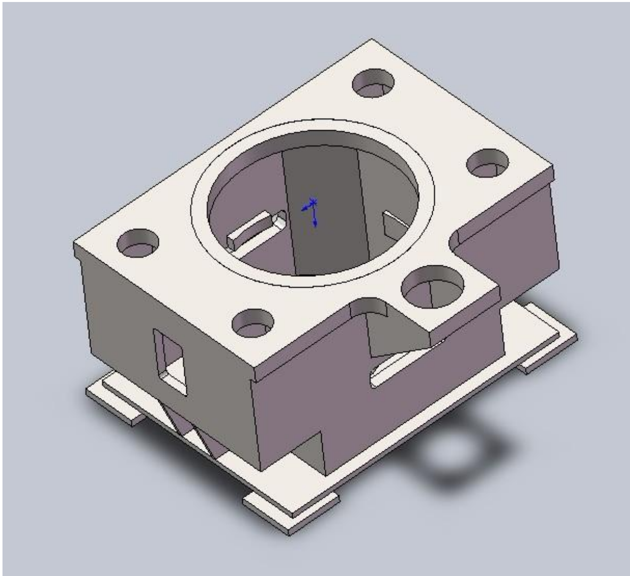

Figure 6. Geometric model of hydraulic presser base.

3.2. Discrete System Model

The base structure is relatively complex and difficult to divide it with the hexahedral element mesh which will kill long time on the job,so tetrahedral element mesh is used when the model of hydraulic press base built by SolidWorks is imported into HyperMesh.The system model is discretized into a computable model composed of many elements. Some elements with connection relationship need to establish contact relationship, and the other discrete element nodes are rigid connection [4-6]. The mesh generation is shown in figure 7.

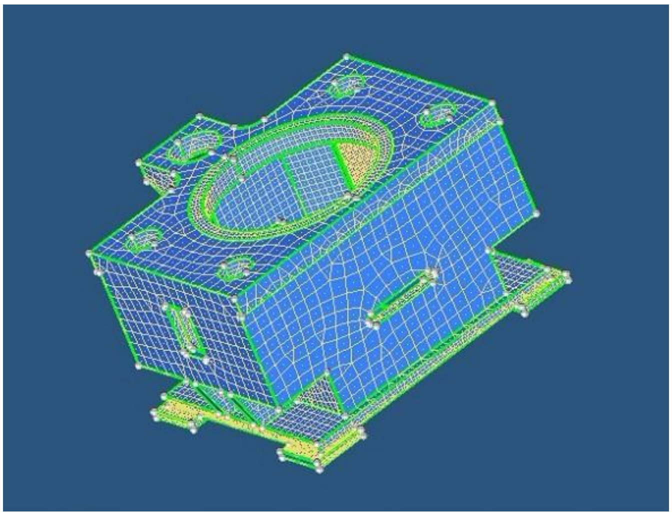

Figure 7. Meshing.

3.3. Model Structure Simplification
Define the material of each part, then setup the components according to the cell properties of the system model. Base of hydraulic presser is so complex in shape that it is made of Q235 nodular cast iron. In order not to complicate the problem and ensure the calculation accuracy, the reasonable simplification of the structure will be conducive to the calculation efficiency. some assumes in this paper are setup as below [4][7].

(1) Shrinkage cavity, crack and other casting defects are not existing.
(2) Ignoring the supporting function of the lead screw.
(3) Ignoring the influence of gravity.
(4) Take the welded parts as a whole structure.

3.4. Establishment the Parameters and Boundary Conditions of the Base

3.4.1.Determine Parameters. The main technical parameters of the hydraulic testing machine involved in this paper are assumed as: the maximum force is 20KN, and the number of columns is 4.Material of the base is assumed as ductile iron with elastic modulus of 160Gpa, Poisson's ratio of 0.25, density of 7.3g/cm^3 and yield strength of 250Mpa.

3.4.2. Adding Boundary Conditions to the Model
(1) Force Constraints

After relatively simplified treatment, the torus on the upper surface of the base is subjected to an equivalent force of about 10kN, which is most consistent with the actual situation, that is, there are four reaction forces under the base in contact with the lead screw, each of them is 5KN.

(2) Displacement constraints

Under this condition, four support plates at the lower end of the base are used as fixed supports.

3.5. Analyze the Characteristics of System Unit
Import the established finite element file into the solver, set the boundary conditions and calculate the

control parameters, then calculate.

4. Solving and Post-processing

The stress and deformation distribution nephogram (figure 8) can be obtained after the finite element calculation, which can establish the condition of the hydraulic press footprint base parts, and then verify whether the design model is reasonable [8].

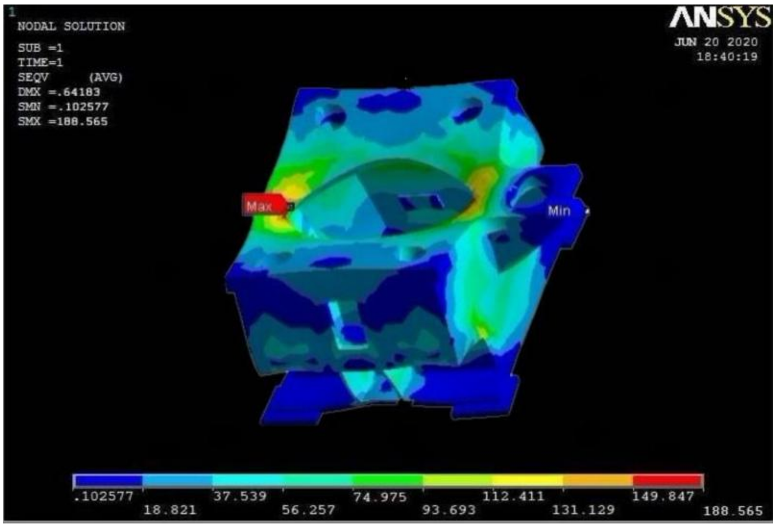

Figure 8. Equivalent stress nephogram of the base.

It can be seen from the equivalent stress nephogram (figure 8) that the maximum equivalent stress of the footprint base is 188Mpa, which mainly occurs in the upper end face of the footprint base contacting with the hydraulic cylinder and the four lower end face regions contacting with the lead screw. The former is due to the maximum force here, while the latter is due to the relatively strong constraint here. Under the maximum stress, the maximum equivalent force of the footprint base is less than the allowable stress of ductile iron, and the safety factor is 1.33, which means that the footprint base structure is safe. At the same time, it can be seen from the figure 8 that the maximum displacement of the footprint base is 0.6418mm, which occurs on the contact surface between the upper end of the footprint base and the hydraulic cylinder. The main reason is that the stress near here is the largest and the structure is relatively thin and is forming an analogous cantilever area here.

By observing the stress nephogram, we can also find that the structural layout of the footprint base is not the most reasonable. The whole footprint base can be regarded as composed of many ribs. Some ribs have small stress and large safety margin, that is, the ribs do not give full play to the supporting role, which means that the whole footprint base has a very large space for optimization, so we can try to optimize the structure of some parts.

5. Optimization of Footprint Base of Four Column Hydraulic Testing Machine

According to the result of finite element analysis, it is found that the structure of the footprint base has the potential chance to optimize. The premise of optimization is that the footprint base structure bears all the forces of the whole machine, and its strength must be sufficient. The shape of the rib plate is optimized, and the optimized results are input into the CAD software for secondary design with the data generated by the tool of OSSmooth.The process of Optistruct shape optimization is shown in figure 9.

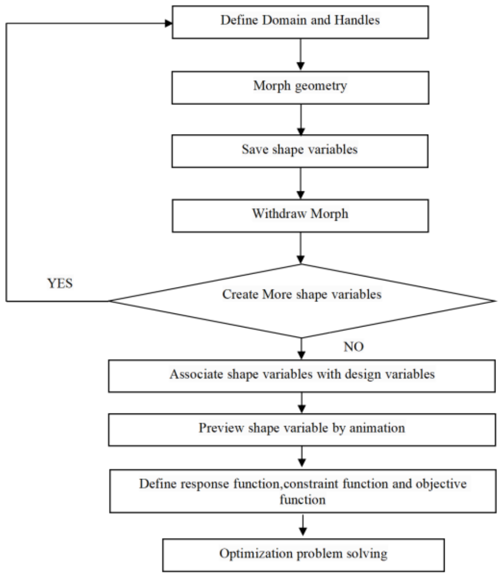

Figure 9. Flow chart of shape optimization with Optistruct.

5.1. Optimization of the Base Shape of Hydraulic Testing Machine

According to the above analysis of the footprint base, the stress distribution of the footprint base is obtained. In order to improve the performance of the footprint base with full stiffness, the shape optimization analysis is worked out. According to the results of finite element analysis, it can be found that the maximum stress is near the torus which is in contact with the hydraulic cylinder and the four small torus which are in contact with the lead screws at the upper end of the footprint base, and the lower structure of the plane where the latter is located is almost free of stress. It is obvious from the cloud picture that the stress here is very small.After careful observation of the cloud image and profile,we find several positions with great potential optimizal margin as shown in figure 8, and optimize size and shape of them.

5.2. Establish the Optimization Model

The prototype results are established and meshed to obtain the finite element model (as shown in figure 10). According to the finite element analysis results shown in figure 8, several dimensions shown in figure 11 are selected as optimization with shape.

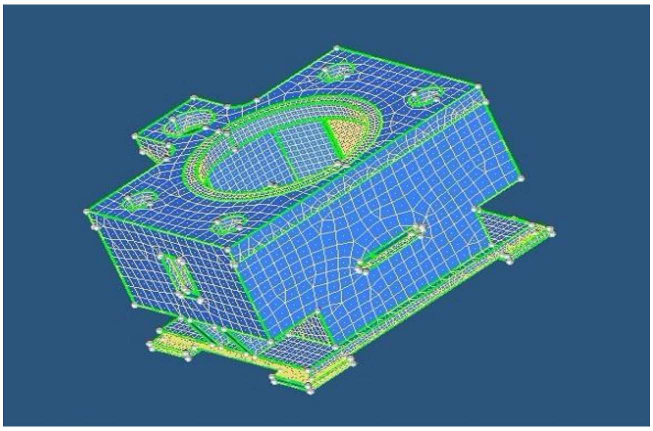

Figure 10. Footprint base initialization model.

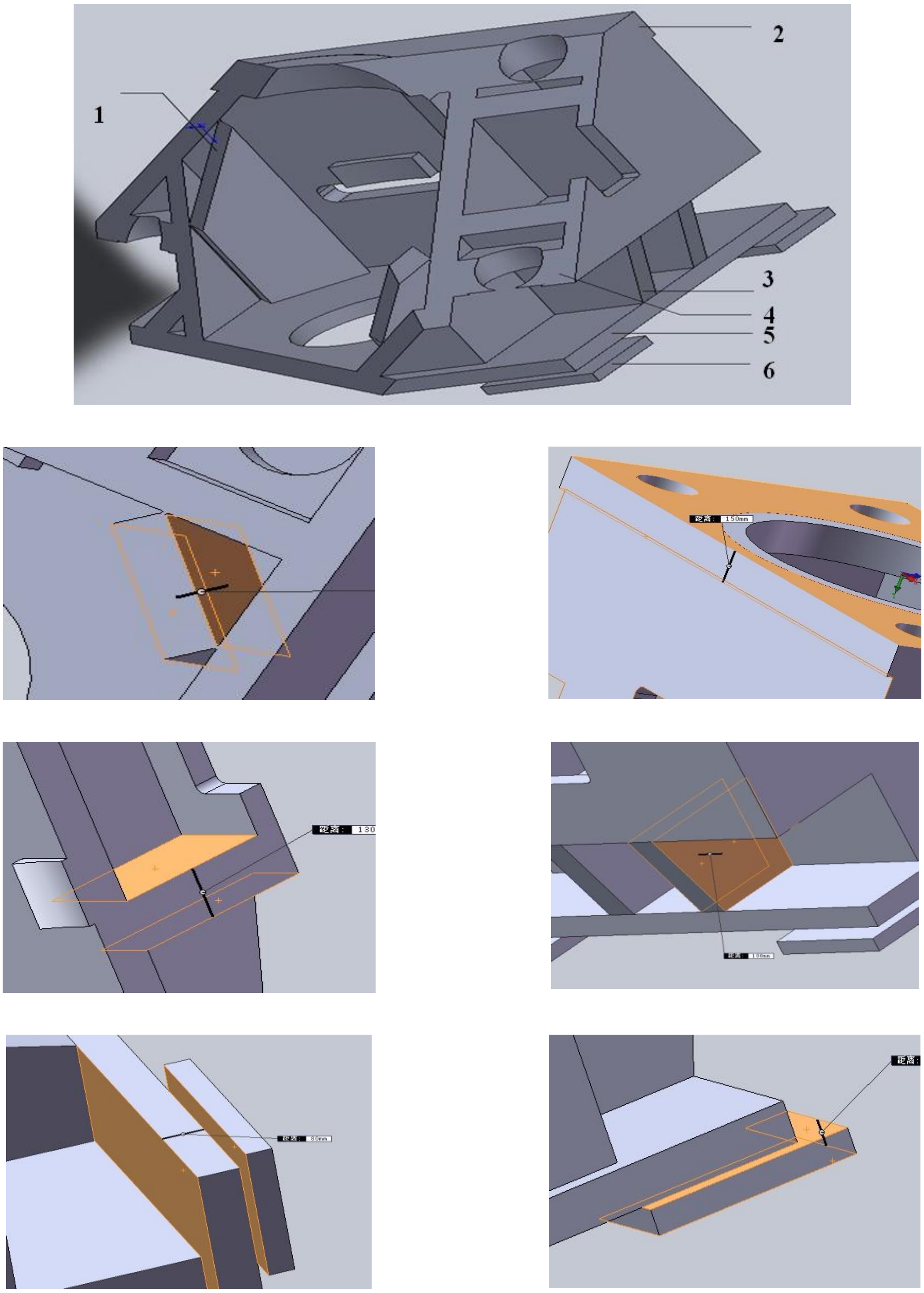

Figure 11. Establish the optimized shape variable (the last six figures correspond to the serial number of the first figure respectively).

5.3. Footprint Base Optimization Results

After solving, the optimized model is obtained (figure 12). Before optimization, the minimum safety factor of the base of hydraulic testing machine is 1.33, and its mass is 15.04 tons; After structural optimization, the safety factor is 1.30, but the mass is reduced to 12.56 tons, and the mass is reduced by nearly one fifth, as shown in table 2 (table 1 for the size optimization data of each components). Hence, It ensures the structural safety, also saves a lot of steel, reduces the production cost, and obtains relatively large economic benefits. The stress and displacement distributions of the optimized the base model are shown in figure 13 and figure 14.

Table1. Optimization results of the components of the base.

Series#	Option	Usual size(mm)	Optimized size(mm)
1	Four ribs in hydraulic cylinder cavity of the base	100	80
2	Upper wall of the base	150	100
3	Four rib plates at the lower end of the lead screw	100	40
4	lower wall of the base	130	100
5	Baseboard of the base	80	40
6	Footplate of the base	50	30

Table 2. Comparison of optimized parameters with these of the usual.

	Max Stress	Max deformation	Min Safety Factor	Mass
Usual Base	188MPa	0.64mm	1.33	15.04t
Optimized Base	192MPa	0.72mm	1.30	12.56t

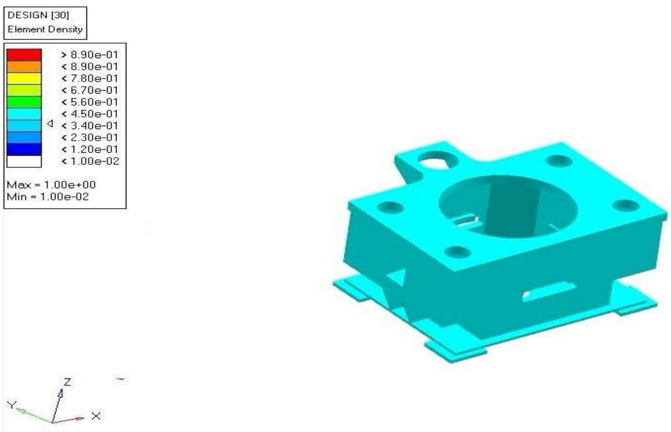

Figure 12. The base optimized by Optistruct.

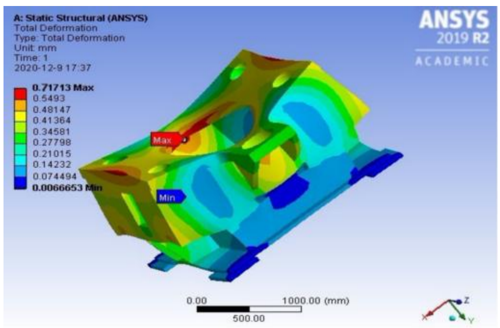

Figure 13. Displacement nephogram of finite element analysis after the base optimization.

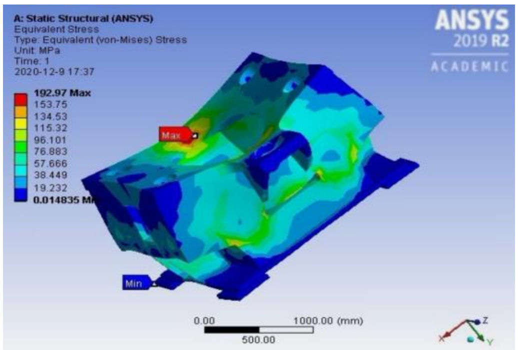

Figure 14. Stress nephogram of footprint base optimization finite element analysis.

6. Conclusion

The paper brings a way to get the deformation and stress nephogram of the footprint base under the maximum pressure. According to the structure of hydraulic press footprint base parts, the model is established by SolidWorks, meshing by HyperMesh, then perform the finite element analysis by ANSYS, combining the advantages of each software, it helps the analysis results more accurate. It not only tells us the footprint base structure can meet the strength requirements, but also a large space for optimization.

During the research, we define the shape optimization region of footprint base by 3D model, then setup the shape optimization variables and their range. After introducing the original boundary conditions and constraints, we setup the optimization objective function and the range of design parameters. Shape optimization calculation is carried out, and the results are obtained, which provides technical support for the footprint base improvement design in future.

Reference

[1] Mao Sh Y and Chen L 2017 Optimization design of CNC bending machine based on ANSYS workbench *Modular Machine Tool & Automatic Manufacturing Technique* (9): 132-135.

[2] Wu Sh, Yuan X and Han M 2021 Three-dimensional parametric modeling and finite element analysis of special coil steel platform container *Packaging Engineering* **42**(1): 129-135.

[3] Yu X L 2006 *The Design and Application of the Hydraulic Press* Beijin: Machinery Industry Press.

[4] Hu Q T 2009 *Finite Elements Analysis of Key Components of Vertical Mill* Wuhan: Wuhan University of Technology.

[5] Li Y C, Zhang L H, Liu Zh W, Wang D Y and Li S 2010 Lightweight and stiffness optimization of a hydraulic press's upper beam based on neural networks and genetic algorithms *Mechanical Science and Technology for Aerospace Engineering* **29**(2): 164-169.

[6] Liang J Zh and Chen L 2009 Finite element analysis and experimental study on the topbridge of the automatic hydraulic press *Machinery Design & Manufacture* (12): 20-22.

[7] Zhou H Q 2017 *Design and Performance Analysis of 120T Hydraulic Unit* Hebei:Yanshan University.

[8] Qin R F and Zha D P 2010 Nonlinear finite element analysis on landing skid of helicopter *ZKG International* **59**(10): 26-34.