Topology Optimization Design of Launcher Bracket Based on Multi-body Dynamics

To cite this article: Min Li et al 2018 J. Phys.: Conf. Ser. 1060 012083

View the article online for updates and enhancements.

You may also like

- Multi-body Dynamic Analysis of the Braking Process for Megawatt Wind Turbine Disc Brake
  S Zhang, C Xue, J Yin et al.

- Automation of wheeled vehicles load bearing frames finite-element models loading procedure by using inertia relief method and vehicle multi-body dynamics model
  D S Vdovin, I V Chichekin, Y Y Levenkov et al.

- Dynamic design of grinding and polishing machine tool for blisk finishing
  Xing Tian Ou, Hao Zhang, Yong Bing Zhao et al.
Topology Optimization Design of Launcher Bracket Based on Multi-body Dynamics

Min Li, Xuezhi Wang, Ming Liu and Chao Li
Air Force Engineering University, Xi’an, Shaanxi, China
Min Li: 17691037413m@sina.cn, Xuezhi Wang: xuezhi@126.com

Abstract: According to the topology optimization theory and engineering application, a launcher bracket topology optimization finite element model was built, which was based on the multi-body dynamics simulation. The optimal material distribution and the force transmission path of the bracket under the constraint conditions were obtained. The stiffness evaluation standard for bracket structure is established by introducing the specific stiffness structural effectiveness. After comparative and analysis, the simulation results show that the topology optimization design of launcher bracket based on multi-body dynamics can achieve the lightweight design of launcher bracket, strengthen structure stiffness, increase the bracket modal frequency, which are helpful to reduce the initial disturbance of missile launching. The method provides reference for the design of similar products.

1. Introduction
Traditionally, the parameters of bracket design is gotten by adopting trial and error approach which depends on relevant experience or direct judgement. This design process requires enough raw materials and lengthen the product development cycle as well [1]. Topology optimization technology, as a promising and innovative technology, has been widely used in the development of aviation and automobile [2]. For the optimization of continuum variable structure, topology optimization mainly includes homogenization method, SIMP method, variable thickness method and topological function description method [3].

2. The principle of topology optimization
The topology optimization method of continuum structure based on SIMP method is mainly to establish the function relation between the material parameters and the structure density, through the change of the design variable of the discrete element to control the change of the stiffness of the element. Thus the change of the overall stiffness matrix of the structure is adjusted. Such iterative optimization will enable structural stiffness to be optimized and the material distribution to be ideal. [4]

The mathematical model of topology optimization is based on elastic mechanics, variational principle and energy quasi rigid establishment [5]. In the view of the mathematical, it is the problem that refer to solving the boundary value under the effect of face force, body force and concentrated force [6]. The interpolation function of the SIMP model is an exponential function, and its mathematical model is [7]

$$
\begin{align}
E_{vol}(x) &= \rho(x)^p E_0, \quad p > 1 \\
\int_\Omega \rho(x) d\Omega &\leq V \\
0 &\leq \rho(x) \leq 1, \quad x \in \Omega
\end{align}
$$
Where $E_{ijkl}$ is the elasticity modulus of the selected material, $\Omega$ is the design area, $V$ is the volume of design area, and $\rho$ is penalty factor.

The density value is between the material characteristic values:

$$E_{ijkl}(\rho = 0) = 0, E_{ijkl}(\rho = 1) = E_{ijkl}^0$$  \hspace{1cm} (2)

For the three dimensional model penalty factor $\rho$, the rule of value is as follows:

$$\rho \geq \max \left\{ 15 \frac{1 - \nu^2_0}{7 - 5\nu^2_0} \cdot 1.5 \frac{1 - \nu^2_0}{1 - 2\nu^2_0} \right\}$$  \hspace{1cm} (3)

It is assumed that the pseudo elastic modulus are all of the same character and Poisson's ratio is a constant. The establishment of mathematical equation of parameter by SIMP method is:

$$\begin{align*}
E(\lambda)/E_0 &= \rho(\lambda)^p \\
\int\rho(\lambda)d\Omega &\leq V
\end{align*}$$  \hspace{1cm} (4)

Where $E(\lambda)$ is Pseudo elastic modulus, $E_0$ is the real elastic modulus of material, $\rho(\lambda)$ is pseudo density.

Under volume constraint, and on the basis of SIMP method, the mathematical model of topology optimization for continuous structures is described as

$$\begin{align*}
\text{Min} : l(u) : \text{s.t. :} & \\
& a_E(u,v) = l(v), v \in U \\
& E_{ijkl}(x) = \rho(x)^p E_{ijkl}^0, p > 1 \\
& \int\rho(\lambda)d\Omega \leq V \\
& 0 \leq \rho_{min} \leq 1, x \in \Omega
\end{align*}$$  \hspace{1cm} (5)

This model can be applied to many occasions, such as the minimization of flexibility, and the problem of quality minimization.

3. The topology optimization of bracket

3.1. The establishment of the optimization model

In order to get the best optimization result of the bracket, the bracket was reasonably simplified. Using the outline entity of the bracket as the initial model and preserving the necessary feature structures. The necessary structures mainly are electric cabinet, support body, ear shaft and bearing race. The ear shaft is connected with the hydraulic rod and the landing gear to provide support and restrict the bracket. The bearing race is fixed on the frame frame by bolt, so that the bracket can realize gyrate azimuth. The hexahedral grids are used to mesh the brackets with each grid size of 50mm, and the division is shown in Figure 1. As the picture shows, the blue grid is the design area and the gray grid is the non-design area.

**Figure 1.** Bracket grid and schematic diagram.
The calculation precision of structural optimization is related to the constraint’s quantity and loads of the model. Rigid flexible coupling technology is adopted to dynamically analyze the launching process of missile in three typical working conditions: the march condition of launch vehicle in 3g gravitational acceleration caused by the road unevenness, the launching condition of launch vehicle at azimuth zero, pitch 60 degrees and at azimuth 45, pitch 60 degrees. From the dynamic analysis and calculation, the loading conditions of each sports deputy node and the amplitude-frequency characteristics of missile vibration were obtained [8]. Frequency response vibration diagram is shown in Figure 2.

![Figure 2. Frequency response vibration diagram.](image)

From Figure 2, we can see that the vibration of missile is the most intense during 10 to 20 HZ, and in the frequency of 20-90 HZ, pitching angular velocity has slowed down, but the maximum pitching angular velocity is up to 1.5deg/sec, which can’t be ignored. Therefore, in order to avoid the resonance phenomenon of the launcher, the first order frequency of the bracket structure should be more than 90 HZ, so that the impact of the bracket on the initial disturbance and vibration of the missile will be reduced to a minimum [9].

By the non-linear finite element analysis, the bracket has the maximum deformation under the third working condition, and the deformation is located at the end surface of the ear shaft, and the vertical displacement reaches 0.45mm. But under the limit load, the deformation displacement of the bracket are smaller than the design requirements, the maximum stress value is far less than the yield limit of the steel material, so there is a lot of structural optimization margin. When the key position of the bracket is topologically superior, the displacement and deformation of key parts should be within the design range, the modal natural frequency should reach the requirement of device vibration frequency, the volume of design area should be reduced as much as possible, and the manufacturing process conforms to the actual production conditions so that to achieve Optimization purposes successfully.

### 3.2. Optimization model parameter setting
- **Objective Function**: Volume minimization function of bracket design area
  - Constraints: the maximum vertical displacement of the joint at the ear axis is less than 1.5mm, and the natural frequency of the bracket is greater than 90HZ.
  - Design Variables: the relative unit density of the force region of the bracket body is the design variable.

### 3.3. Topology optimization results
The bracket volume optimization iterative curve and the first order modal inherent frequency iteration curve are shown in Figures 3 and Figure 4 and the result of the bracket topology optimization is shown in Figure 5. From Figure 4, the volume of the bracket in the first 6 times of iteration process is greatly reduced, indicating that the initial structure of the bracket has a large number of materials that don't bear load, and can be deleted. After that, the curve is gradually convergent after a small amplitude oscillation. The whole calculation process is calculated by 27 iterations.
After optimization, if the bracket element density is 1, then the material at that unit is very important and needs to be retained. If it is 0 or near 0, then the unit material is unimportant and can be removed. Setting the element density threshold as 0.3, so the element whose density threshold is more than 0.3 is retained. Finally, the preliminary optimization structure model is obtained. Compared with the original bracket, the new bracket design mainly adopts the box structure welded by steel plate and proximate matter. By adjusting the structure, the main load bearing material is concentrated on the bottom surface, which is in line with the actual manufacturing and processing requirements. The optimization results are exported through the OSSmooth tool, and the irregular parts of the optimized bracket are regularized, and the final geometry model is obtained considering the actual engineering structure of the bracket. The final geometry model is shown in figure 6.

4. Model verification analysis

4.1. Comparison and analysis of parameter performance
In order to analyze the overall stiffness change of the bracket structure, the specific stiffness effectiveness evaluation was adopted. The specific stiffness formula is shown in formula 4. The greater the value of specific stiffness, the higher the weight stiffness of the bracket unit. The finite element analysis is carried out for the optimized model, and the optimization results are compared with the initial model.

$$\beta = \frac{E / s_d}{m_s}$$  \hspace{1cm} (6)

Where $\beta$ is Specific stiffness efficiency, $E$ is Modulus of elasticity, $s_d$ is Structural displacement, $m_s$ is Structure weight.
Table 1. Comparison and analysis of performance parameters of optimal mode.

|                  | s/mm | σ/MPa | m/t | f/Hz | β     |
|------------------|------|-------|-----|------|-------|
| Initial model    | 0.49 | 95.4  | 5.4 | 74   | 78.9  |
| Opt. model       | 0.55 | 143.8 | 3.8 | 110  | 100.5 |

The results of the analysis are shown in Table 1, where s and σ stand for maximum deformation and maximum displacement of the bracket, m and f stand for the quality of the bracket and the natural frequency of the first order mode. From table 1, the optimization model has a slight increase in deformation, but still within the range of 1.5 mm; the maximum stress value increased by 58 MPa, mainly due to the stress concentration at the connection of profiles. The rigidity of profile connection can be increased by welding mode, making it smooth, which can effectively reduce the maximum stress value. The specific stiffness effectiveness of the new bracket is increased by 27.4%, which indicates that the stiffness margin of the bracket unit is improved effectively by the optimization model. At the same time, the modal analysis shows that the first order natural frequency of the optimized model is raised by 36, which effectively avoids the resonance frequency range of the launcher.

4.2. Analysis of influence of initial disturbance

The new rigid flexible coupling model is established through the ADAMS, and the dynamic simulation calculation is carried out [10]. In this paper, the launching was analyzed while the launcher is full of missile. The simulation time is 1s, and the step size is 200. Through the dynamic response analysis, the effect of the optimization model on the initial disturbance of the missile is further verified [11]. Figure 7 and Figure 8 is the comparison curve of the missile pitch-angle velocity and the azimuth-angle velocity. Figure 9 is the vibration displacement curve of the launcher barrel.

Figure 7. Missile Pitch-angle velocity correlation curve.  Figure 8. Missile azimuthal-velocity correlation curve.  Figure 9. Vibration displacement curve of launch canister.
In Figure 7, Figure 8 and Figure 9, the blue dotted line is the initial model simulation curve, and the red real line is the optimization model simulation curve. The locking force of the missile is unlocked at 0.065 s, the front hanger is separated from the guideway at 0.175 s, and the rear hanger is separated from the guideway at 0.325 s. When the missile hangs on the guideway only by rear hanger, it forms a bending moment under the gravity and the vibration of launcher. As a result, it leads to the sinking of the missile. Before 0.325 S, as seen in Figure 7 and Figure 8, the red curve's peak is always lower than the optimal model. Although there is a moment of red peak is greater than the blue curve during the moving of rear hanger on guideway, the pitch-angular velocity and azimuth-velocity is still lower than the initial model after the rear hanger leaves. For the vibration of launcher's canister, the vibration amplitude of the initial model reaches 0.016 mm and the optimization model is only 0.007 mm. To sum up, compared with the initial model, the optimization model can effectively improve the performance of the launcher and reduce the influence of the resonance effect of bracket and launcher on the initial disturbance of the missile.

5. Summary
In this paper, the combination of multi-body dynamics and topology optimization theory is adopted to optimize the launcher bracket with heavy weight, low structural rigidity and low natural frequency, which can cause the resonance effect of the device and increase the initial disturbance of the missile.

1) Compared with the initial structure, the weight of the bracket is obviously reduced, the stiffness and natural frequency are improved, so that the resonant frequency range of the launcher is effectively avoided. It is verified by multi-body dynamics simulation that the optimized model plays a good role in reducing the initial disturbance of the missile and the vibration of canister.

2) After optimization, the displacement deformation of the model increases slightly under the limit load, but the deformation is within the range of design, which proves that the method is feasible.

3) The combination of multi-body dynamics and topology optimization is also instructive and referential for other structural optimization of launchers, and is also suitable for other similar products' initial design and manufacture.

References
[1] FAN Can 2016 Technology Innovation and Application vol 18 pp 137
[2] LIN Guowen, MA Dawei, ZHU Sunke 2012 Journal of Ballistics vol 24 pp 101-105
[3] ZHANG Guoxian 2012 Qinghuangdao: Yanshan University, pp 4-10
[4] HU Ya 2016 NanJing: Nanjing University of Science and Technology pp 25-30
[5] Park G J 2011 Structural & Multidisciplinary Optimization vol 43 pp 319-337
[6] Choi W S, Park G J. 2012 Computer Methods in Applied Mechanics and Engineering vol 191 pp 2105-2122
[7] CEN Haitang. 2004 Bionic theory of structure, bionic design of light part structure and verification of RP process Beijing: Beijing University of Aeronautics and Astronautics
[8] CUI Longfei, ZHANG Long. 2013 Journal of Gun Launch &Control Vol 4 pp 24-29
[9] YU Hailong, RUI Youting, HE Bin, et al. 2006 Journal of Ballistics vol 18 pp 62-64
[10] ZHANG Shiwei 2010 Science Technology and Engineering vol 10 pp 5456-5461
[11] CHEN Zhen, BI Shihua, LIU Guangpu, et al. 2010 Journal of Solid Rocket Technology vol 33 pp 112-114.