FEA and Optimization of a Door Used in Large Vacuum Chamber

Yi Zhang *, Shizeng Lv, Xiao Han and Jie Gong
Beijing Institute of Spacecraft Environment Engineering, 100094 Beijing, China

*E-mail: spcomyth@139.com

Abstract. The door is the access for the test piece and personnel, and it is one of the important parts of the vacuum chamber. This paper introduces a kind of door of a large vacuum chamber, which moves radially along the ground guide rail. Because of the risk of overturning, we use the finite element software to create a finite element model, define the material properties, and apply the load. On this basis, we carry out stress analysis, displacement analysis and overturning analysis. According to the analysis results, we give two optimization methods for anti-overturning. Finally, we analyse the overturning resistance of the optimized model.

1. Foreword
In recent years, the demand for comfort testing in cockpit has gradually increased in aviation field. The large vacuum chamber with diameter of 10m is the principal part of the environmental simulation equipment used in the experiment, and the door is the key part of the vacuum chamber [1].

The door moves radially along the ground rail [2,3]. At present, there are many analyses on its airtightness, stress and displacement, but there is no analysis on its overturning [4,5]. However, there is a risk of overturning for this door, especially for the large vacuum chamber, the impact of the door’s collapse is unimaginable. Therefore, we use the finite element software not only to analyse the stress and displacement of the door structure, but also to analyse its overturning risk [6]. According to the analysis results, the optimization method of anti-overturning is given to ensure the safety and reliability of the door.

2. Structure of the Door
The structure of the door is shown in Figure 1, which is composed of head, flange and supporting frame. It is the passageway for the specimen and personnel, and is one of the important parts of the vacuum chamber [7]. The support frame is equipped with wheels and moves radially along the ground guide rails. The head is spherical crown shape, the inner diameter is φ10m, and the size of support frame is 12000mm (L) × 3000mm (W) × 5300mm (H).

3. Finite element analysis
The structure of the door is complex, so the traditional mechanical analysis method can't be used to analyse it. However, the finite element method has its unique advantages in stress, displacement and overturning analysis [7].

We create the finite element model of the door, and carry out stress analysis, displacement analysis and overturning analysis.

3.1. Pre-processing

3.1.1. Create a finite element model. Because there are some fine structures in the model, in order to obtain the simulation results more quickly and accurately, we use CATIA software to simplify the model, the simplified model is shown in Figure 2. We use HyperMesh software to mesh the model, as shown in Figure 3, the mesh size is 80mm, the total number is 43775. The mesh is evenly distributed, appropriately dense and of good quality.

3.1.2. Material properties. The door material is Q345R. See table 1 for the properties of materials [8].
### Table 1. Material properties

| Material | Elastic Modulus (MPa) | Poisson ratio | Density (kg/m³) | Yield strength (MPa) | Factor of safety | Allowable stress (MPa) |
|----------|-----------------------|---------------|-----------------|----------------------|------------------|-----------------------|
| Q345R    | 210000                | 0.3           | 7850            | 345                  | 1.7              | 202                   |

3.1.3. **Applied load.** The geometric boundary condition of the finite element model is mainly gravity load. According to the actual working condition, all units bear gravity (g = 9.8m / S2), and a horizontal support surface is built at the bottom, as shown in Figure 4.

![Figure 4. Boundary conditions.](image)

3.2. **Stress analysis results**

The overall stress distribution of the door and the location of the maximum stress are shown in Figure 5. The area with high stress is mainly concentrated at the joint of inclined square pipe and vertical square pipe of frame. The maximum stress is 171.4 MPa, which is located at the bottom support point, which is lower than the allowable stress of material by 202MPa.

![Figure 5. Stress distribution of door.](image)

3.3. **Displacement analysis results**

The overall displacement distribution and the maximum displacement position of the door are shown in Figure 6. The maximum deformation displacement is 15.03mm, which is located on the top of the spherical crown head. The area with large displacement of the supporting frame is mainly distributed in the vertical square tube connected with the inclined support, as shown in Figure 7.
3.4. Overturn analysis results
As shown in Figure 8, the support point at the bottom far away from the door has an upward displacement of 0.4mm, which indicates that the support point far away from the door has been suspended, so we can see that the door has a tendency of toppling forward.

4. Anti-overturning optimization
4.1. Optimization method 1
In order to avoid the risk of overturning, it is necessary to optimize the structure of the door. The first method of optimization is to widen the base, extend the square pipe with the mark of 1-5 by 100mm along the direction of red arrow. The comparison between the pre optimization and post optimization is shown in Figure 9.
5

Figure 9. Comparison before and after optimization.

The finite element simulation results after widening the base are shown in Figure 10. The four support points are all in contact with the support surface, and the trend of the optimized model dumping disappears.

Figure 10. Displacement distribution of optimized bottom support point.

4.2. Optimization method 2
The second optimization method is to place a rectangular iron bar with a mass of 10t on the beam of the supporting frame to play the role of counterweight, as shown in Figure 11.
After adding 10t counterweight to the supporting frame, the results of the finite element simulation are shown in Figure 12. All four supporting points are in contact with the supporting surface, and the trend of the optimized model dumping disappears.

4.3. Analysis of door overturning resistance

The first optimization method is adopted in this paper. In order to inspect its overturning resistance after optimization, the door is tilted forward $1^\circ$ during pre-processing, as shown in Figure 13. It can be seen that the front support point of the door is in contact with the ground, while the back support point is far away from the ground.

The simulation result is shown in Figure 14, the back support point of the door contacts the ground. This shows that the door can return to its normal position without overturning under the condition of $1^\circ$ inclination.
5. Conclusion

In this paper, a set of door used in large vacuum chamber is introduced, and its structure is analysed by finite element method. The analysis contents include: stress analysis, displacement analysis, overturn analysis. The analysis results are as follows:

- The maximum stress of the door is 171.4MPa, which is located at the bottom wheel support point, lower than the allowable stress of the material;
- The maximum displacement of the door is 15.03mm, which is located on the top of the head;
- The support point at the bottom away from the door has an upward displacement of 0.4mm, indicating that the support point away from the door has been suspended, and the door has a tendency of forward overturning.

In order to avoid the risk of door overturning, two kinds of optimization methods for anti-overturning and the analysis results after optimization are given:

- The base of the supporting frame is widened by 100 mm. The four support points of the optimized model are all in contact with the support surface, the tendency of dumping disappears, and can resist at least 1° of overturning;
- Place a weight of 10t above the beam under the supporting frame. The four support points of the optimized model are all in contact with the support surface, and the tendency of dumping disappears.

At present, the optimized door is in use with stable and reliable operation.

References

[1] Xiao Han, Yan Qi and Botao Liu 2010 Structural design and optimization for door flange of a large vacuum vessel (Spacecraft Environment Engineering vol 4) p 493-495
[2] Hengguo Mao, Guoming Xu, Hangsheng Zhou and et al 2012 Three new designs of the motion mechanism for the vacuum vessel gate (Spacecraft Engineering vol 6) p 671-673
[3] Shouquan Ke, Bencheng Huang, Chuanda He and et al 1993 Spacecraft environmental engineering and simulation test (Beijing: Astronautic Press) p 116-118
[4] Rongda Mao, Bo Li, YuanLiang Liu, and et al 2018 ANSYS-based Structure Analysis and Improvement of a Special-shaped Vacuum Vessel (Vacuum & Cryogenics vol 5) p 348-352
[5] Long Wen, Guilong Ling, Guobiao Cai 2011 Deformation Simulation of Single—Axis Double-Pin-Hinged Doors (Chinese Journal of Vacuum Science and Technology vol 1) p 50-56
[6] Shizeng Lv, Xiao Han and Yan Qi 2016 Reliability optimization design of vacuum chamber for extra-large space environmental simulation equipment (Chinese Journal of Vacuum Science and Technology vol 1) p 38-43
[7] Bencheng Huang and Jinming Chen 2005 Space Vacuum Environment & Vacuum Technology (Beijing: National Defense Industry Press) p 38-68
[8] GB 150.2-2011 2012 Pressure vessels-Part 2: Materials (Beijing: Standards Press of China)