

PAPER • OPEN ACCESS

3-D modeling and finite elements analysis of KM6 motion simulator

To cite this article: Q S Xiao *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **504** 012072

View the [article online](#) for updates and enhancements.

3-D modeling and finite elements analysis of KM6 motion simulator

Q S Xiao¹, J W Zhang², G F Wang², Z H Du² and H Fang²

¹ Beijing Institute of Spacecraft Environment Engineering. 100094, Beijing, China

² National Key Laboratory for Precision Hot Processing of Metals, Harbin Institute of Technology. 150001, Harbin, China

Corresponding author and e-mail: Q S Xiao, xqs.hit@163.com

Abstract. As an important part of the satellite space sport simulation system of the KM6 manned spacecraft space environmental test equipment, the satellite motion simulator has the function of simulating and testing the satellites on the ground. In this paper, the 3-D model of the motion simulator is established, and its performance is analysed using finite elements simulation software under different conditions. Under the different conditions of static force and dynamics, the simulation results of the vertical state of the spin axis and tilted 30° state of the spin axis are compared and the optimal working conditions is designed. Modal analysis and buckling analysis are performed to prove that the design of the structure is feasible.

1. Introduction

When using a solar simulator for vacuum testing in a floor-heating vacuum simulation room, motion simulator is used to hold the spacecraft in different postures facing the sun while maintaining real-time energy and information contact. The motion simulator consists of mechanical structure system, source communication system, measurement and control system, cooling system, heat sink and load [1-3]. This article uses the 3-D drawing software Solidworks to draw the movement simulator, then carries on the interference check after the assembly, the revision improves the original design to have no interference [4]. Hypermesh and Marc 2012 are used to investigate the state of the motion simulator with different spin axis angles and analyze the state of different working conditions [5,6].

2. 3-D model of the motion simulator

The motion simulator consists of mechanical structure system, source communication system, measurement and control system, cooling system, heat sink and load. Among them, the mechanical structure system consists of a spin axis, an attitude axis, a bracket, and a torque balancer. The source communication system is composed of sink rings, water hinges, and microwave loads; the measurement and control system is equipped with three temperature measuring instruments and cables installed on the motion simulator; and the cooling system is installed on the part of the motion simulator. Cooling water pipes; heat sinks are supported on the outer wall of the spin shaft cavity through mounting legs; the load is mounted on the test piece flange at the top of the motion simulator for docking flower baskets. After components are drawn in the software, they will be assembled. The motion simulator is presented on Figure.1.



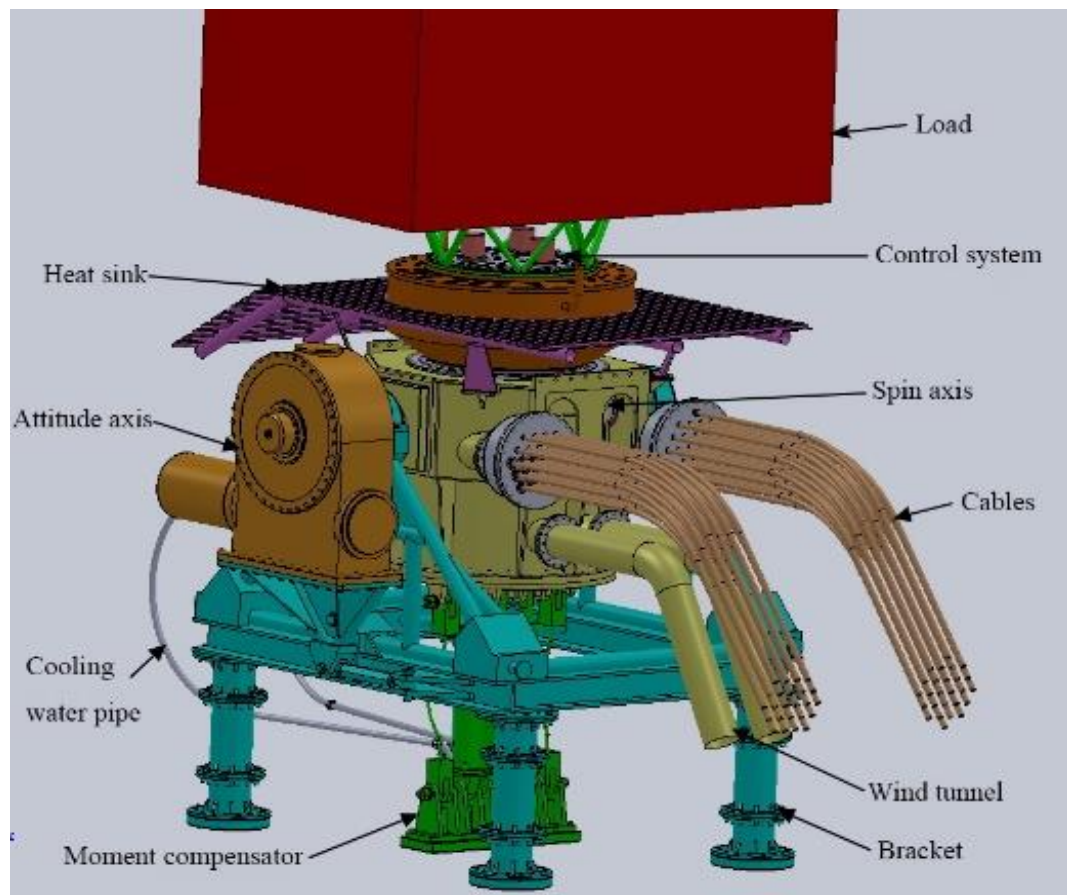


Figure 1. Motion simulators overall structure.

3. Static finite element simulation of KM6 motion simulator

The purpose of using the Hypermesh to mesh the 3-D model is to ensure the accuracy of the finite element calculation and improve the computational efficiency. The thin structure of the motion simulator is processed by shelling, and the 2D mesh is used to draw the mesh. The three-dimensional solid meshing of the components involved in the motion in the motion simulator ensures that the calculation accuracy is ensured, the number of solid meshes is effectively reduced, and the computational efficiency is effectively improved. Then it will be imported to the Marc to perform a finite element simulation.

The static conditions are simulated under two different conditions, and the overall stress of the motion simulator is examined. The results of the finite element simulation are shown in the figure below. Since the conditions investigated are static conditions, the shaft movement is not considered. Therefore, the stress and strain of the shaft are not meaningful in the results, and the results are not shown.

3.1. Spin axis vertical state

3.1.1. Strength analysis

The maximum stress is 100 MPa, which appears in the bracket part near the attitude axis and the lower hinge mounting connection part of the torque balancer, as shown in Figure.2-1(a). The maximum equivalent stress value is 37.5 MPa, which appears on the stent near the attitude axis as showed in Figure 2(b).

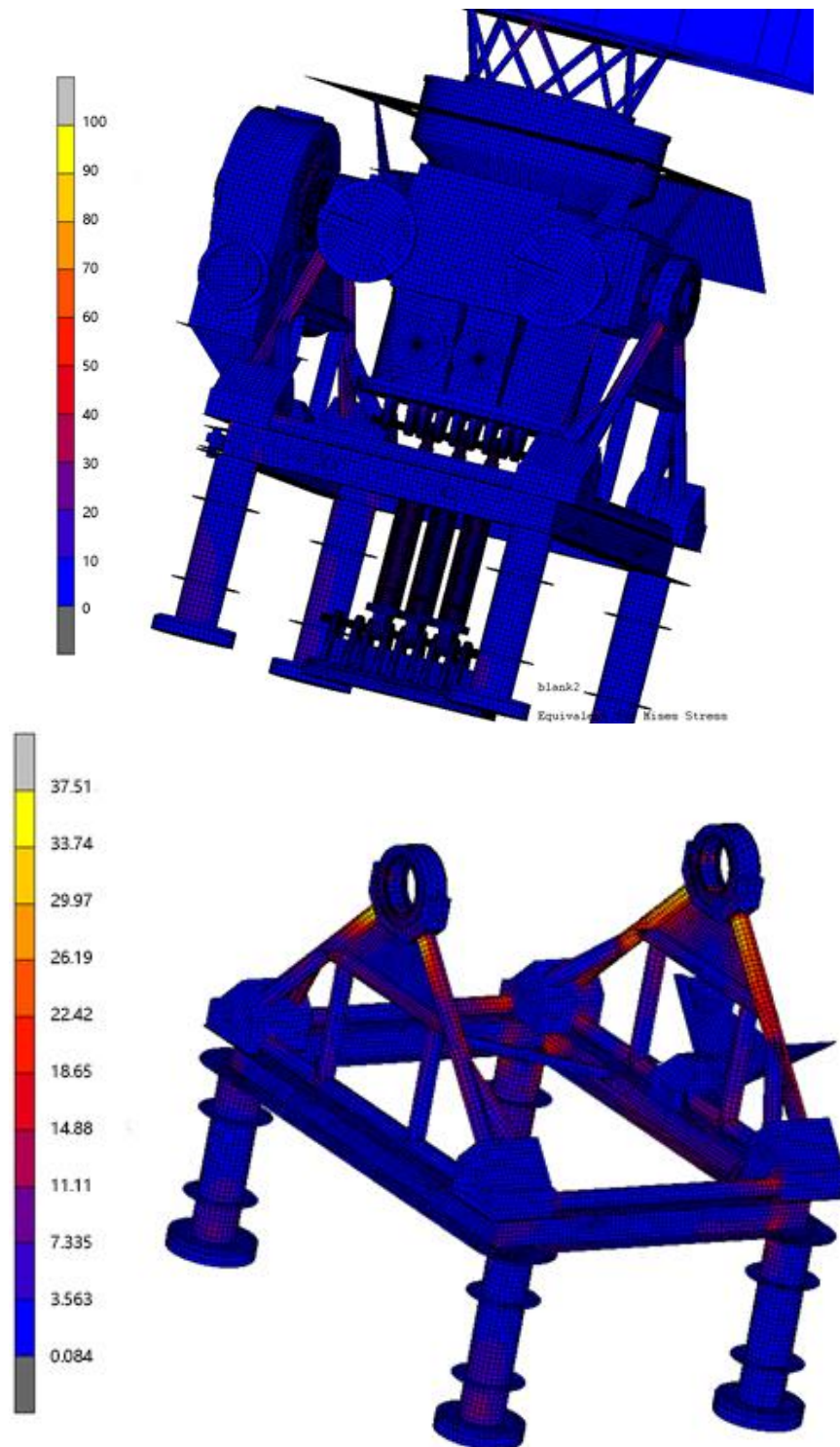


Figure 2. (a) under condition 1; (b) Stress distribution clouds diagram of the ridge support.

3.1.2. Rigidity analysis

As showed in Figure.3, the maximum displacement value is 1.5mm, which appears in the worm gear box. This is mainly due to the large weight, and the turbine shaft is a bearing with a certain gap. When adjusting the fit of the turbine shaft and the worm gear box, the actual situation will not be so serious.

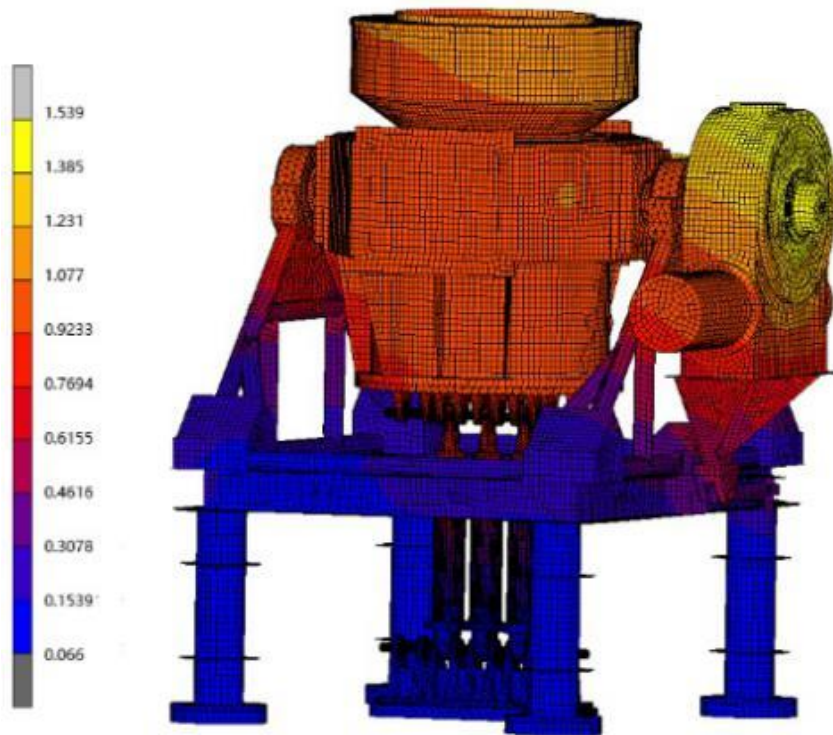


Figure 3. Motion Simulator Displacement Distribution Cloud.

As showed in Figure.4, maximum displacement values of 0.952mm occur in the area where the torque balancer is located near the spin bearing force frame.

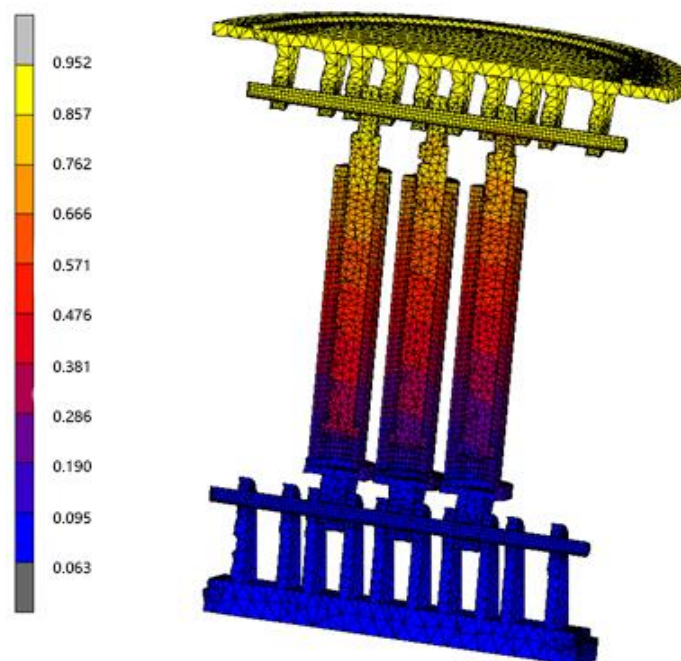


Figure 4. Displacement distribution clouds diagram of the guide rod and guides sleeve of moment balancer.

The method of Spindle tilts 30° is same as those in the vertical state.

3.2. Comparison of simulation results

The comparison results are shown in Table.1.

Table 1. Comparison of data under two conditions.

Data	Working condition 1	Working condition 2
Motion simulator maximum stress	100MPa	127.6MPa
The maximum stress value of the stent	37.5MPa	82.6MPa
Motion simulator overall maximum displacement	1.5mm	2.7mm

From the comparison of the results, it can be clearly seen when the axis of rotation is tilted by 30° . The performance examined is improved compared to condition 1. The maximum stress is within the material allowable stress range.

4. Dynamic finite element simulation of KM6 motion simulator

In order to study the stress and deformation of the entire mechanism under motion, different parts are set into different deformation bodies. And the contact relationship and friction boundary conditions between different deformation bodies are respectively set to realize the relative movement of different parts driven by the motor and Rotation to more truly reflect the stress and deformation of the mechanism under actual working conditions. The simulation method is the same as in Chapter 3.

The results show that the maximum value of the equivalent stress of the motion simulator is 276 MPa under the condition of condition 1, and the maximum value of the equivalent stress is 277.2 MPa at the condition of condition 2.

5. Buckling analysis

When the axis of rotation is vertical, the stress of the mechanism becomes larger as the external load increases. The maximum stress of the mechanism at 10 tons is 81 MPa; the maximum stress of the mechanism at 50 tons is 311 MPa; and the maximum stress at 100 tons is 718 MPa. The strain is only 0.003, indicating that the load resistance under condition 1 is very strong.

When the axis of rotation is tilted by 30° , the stress of the mechanism also increases as the external load increases. The maximum stress of the mechanism at 10 tons is 71.7 MPa; the maximum stress at 50 tons is 249 MPa; the maximum stress at 70 tons is 328 MPa, and the strain is only 0.027 at 70 tons. The beginning of plastic deformation indicates that the resistance to external load under condition 2 is weaker than condition 1, but it is still very strong.

6. Conclusions

- 1) The three-dimensional modeling of the motion simulator is completed. The simulation analysis shows that the overall structural design of the mechanism is feasible. After modifying the interference phenomenon at some positions, the required actions can be completed and the mechanical structure is free from interference.
- 2) The results of the static finite element simulation show that when the spin axis is tilted by 30° under working condition2, the performance obtained is higher than that of condition 1. The maximum stress is within the material allowable stress range.
- 3) The dynamic finite element simulation results show that when the spin axis is tilted by 30° and reversed by 30° , the maximum value of the equivalent stress at the local position of the flange and the load-bearing frame is 193 MPa, which appears in the ribs and the test piece. The contact part of the flange box body and the contact part of the reinforcement ribs and the load-bearing box are

recommended to be locally thickened to 15mm and 20mm, respectively. The maximum stresses and strains of the other indicators examined are within the allowable range.

- 4) The buckling analysis results show that even under condition 2, the strain is only 0.027 when the external load reaches 14 times the normal load, and the plastic deformation is just beginning to occur, indicating that the structure is very resistant to external loads.

Acknowledgement

Thanks to Professor G F Wang for giving me guidance. I also want to say thanks to the members of the laboratory for their help.

References

- [1] Larson W J and Wertz J R 1992 *Space Mission Analysis and Design* (California: Microcosm)
- [2] Wertz J R 1978 *Spacecraft Attitude Determination and Control* (Dordrecht: Kluwer)
- [3] Allotta B, Conti R, Meli E, Pugi L and Ridolfi A 2014 *Mechanica* **49** pp 615-644
- [4] Avanzini G and Giulietti F 2012 *Journal of Guidance, Control, and Dynamics* **35**(4) pp 1326-1334
- [5] Darvizeh A, Darvizeh M, Ansari R and Meshkinzar A 2013 *Thin-Walled Structures* **71** pp 81-90
- [6] Shetty S K 2006 Finite element study of energy absorption characteristics of a hybrid structure-composite wrapped on a square metal tube (Doctoral dissertation, Wichita State University)