# Numerical Analysis and Structure Design for the Airborne Pod Bracket Based on UG and ANSYS workbench

Dongfang Hu[1, a], Ruonan Lei[1, b] and Yan Zhao[2, c]

[1]School of Mechatronics Engineering, Henan University of Science and Technology, Luoyang 471003, China

[2]LandGlass Technology Co., Ltd, Luoyang 471003, China

[a]hdf@haust.edu.cn, [b]1030520781@qq.com, [c]hlxc-2009@163.com

**Keywords:** Photoelectric Pod; Bracket; Structure Design; Numerical Analysis

**Abstract.** Pod bracket is the connecting part of azimuth axis and pitch axis, due to the influence of the plane attitude, wind resistance and other factors on the plane, its rationality and reliability of design deserves a high degree of concern. First, the specific structure is determined according to the technical requirements of pod. Then, the bracket is modelied based on UG, and the "assignment variables method" is adopted to realize the whole parameters correlation. At the same time, the bidirectional refresh modification of UG and ANSYS Workbench is achieved by setting the relevant parameters of software. Finally, after determining the optimal range of mesh and comparing the result of Nastran module and ANSYS Workbench module analysis data, the stress and displacement of the bracket is used to do dynamic simulation to prove its safety and feasibility, and provide new ideas for the future optimization.

## Introduction

Pod, a kind of structure composed of an airborne turntable and a stable imaging system, which can help the helicopter to complete search as the core of maritime search and rescue technology.

The azimuth angle and pitch motion of pod is adjusted by the way that the motor drive gimbal mechanism. And UG were used to support for the geometric modeling, "variable assignment method is used to realize the parameterized. At the same time, through parameter settings of relevant software, and modification of the geometrical parameters under UG environment, two-way refresh is realized. Eventually, receiving the seamless connection of UG and ANSYS Workbench, which can export finite element model in a real time.

Based on the same boundary conditions, the results of the NASTRAN and Workbench ANSYS are compared, the causes and their respective advantages are analyzed. And through the static analysis technology, satisfying the structural requirements of the designed support, and determining an optimal design of specific boundary conditions.

## The Key Issues of Interface Setting and Collaborative Simulation of UG and ANSYS Workbench

**UG Interface Settings.** ANSYS Workbench usually has two kinds of modeling methods: the first is to establish the model directly in the environment; the second is to establish the geometric model in the three-dimensional modeling software such as UG, Pro/E, CATIA and so on [1].

**The Bracket Model Import and Collaborative Simulation Processing in UG.** Geometric model of airborne pod is created by UG. After setting, the original menu bar will increase the ANSYS Workbench. Through ANSYS Workbench, graphic information by loading can keep synchronization with models in UG, which means that the bidirectional refresh and collaborative modeling are implemented. If the finite element model of Workbench ANSYS needs to be modified, just do it in UG, then refresh, and vice versa.

The cooperation of NX, ANSYS and Workbench UG actually represents the interaction of data in the design process. First of all, the parametric model of pod support is established in UG software, or directly modified existing models. Then, the model should be imported to the AWB (ANSYS Workbench), and carried on the simulation analysis [2]. In AWB, the parts are divided into grid, the load is applied, and the result is analyzed, and an optimization model is worked out. Eventually, import the results to UG. The realization of the whole process is shown in Fig. 1

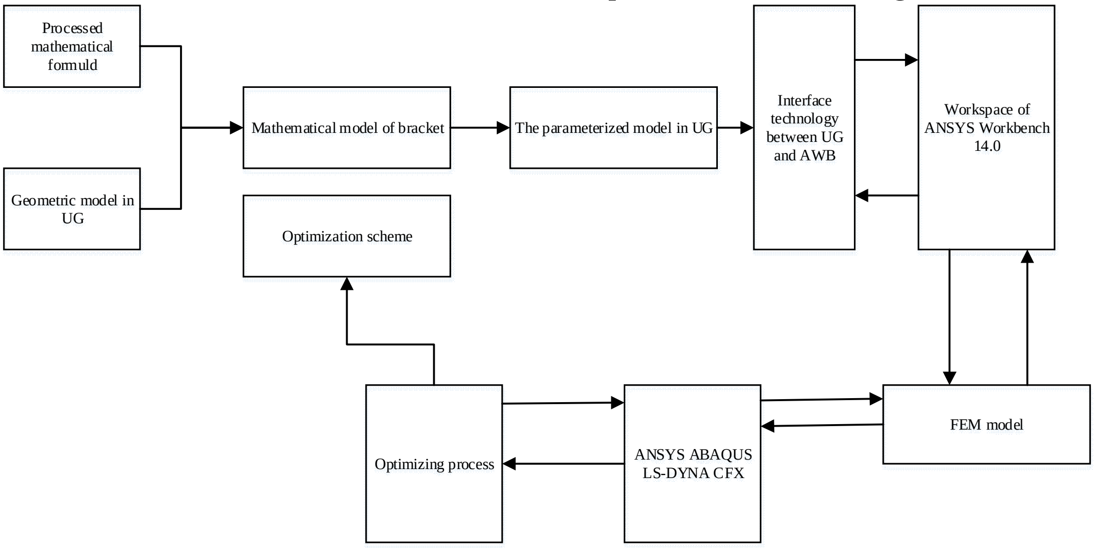



Fig. 1 Collaborative simulation data flow based on UG and ANSYS workbench

## Analysis of Key Problems for Airborne Pod Bracket

**Geometry Modeling.** Geometric modeling in UG is a process of describing the shape of physical or mathematical objects using the concept of geometry [3]. Reference given by enterprises technical requirements of pod bracket parameterized solid modeling by UG solid modeling module.

**Mathematical Treatment of Geometric Entities**. In terms of the concept of parametric modeling, the realization of parametric modeling in UG is based on a series of parameters, constraints and relationships. However, the faint beam generated automatically will limit the parameters of the relevant changes in the modeling process, and easy to cause the failure of the model regeneration, so it is necessary to preview the established geometric model.

Although UG provides a number of secondary development tools such as GRIP UG/Open, API UG/Open and UG/Open, but these tools need to be used by designers with high technical ability. However, the table driven technology provided by UG can also be used to create 3D model library of product series design. First, building a three-dimensional geometric model of the airborne nacelle support based on the two-dimensional map; secondly, setting design variables and assigning them to the geometrical features of the model; finally, create an external spreadsheet containing these variables and link it to the current model. Variables in the electronic watch is referenced by the size of current graphic file, so the designer can modify parts' size by controlling the external electronic table. Avoiding the loss caused by changes in the design, and just one model can be expressed a series of similar structure parts.

**FEM Analysis of NASTRAN Under UG Environment**. In order to reduce the Inertia generated by the process of the bracket motion, reduce physical quality, and ensure the strength and toughness of the bracket, after a comprehensive analysis of mechanical property for the material, and ultimately preferred ZL205A aluminum copper alloy treated by T6, and grade is ZAlCu5MnCdVA, copper content range from 4.5% to 5.3%, its manganese and titanium elements can significantly increase the high temperature strength and casting performance. Furthermore, After T6 treatment, the density becomes 2.82g/cm3, and Young's modulus: E=68GPa, Poisson's ratio: u=0.32.

**Mesh Quality Control and Optimal Interval Selection**. The degree of mesh subdivision has a certain influence on the reliability of the final results. In order to determine the scope of the optimal grid size, the numerical value of the mesh size is

$$y = 2^{\frac{1}{x}}\left(x = 0.2\times n; n = 1\cdots+\infty\right) \tag{1}$$

In this equations, X is a set of numbers, and mesh in order.

Tetrahedral four node grid technology can be used to fit the curve. When the unit size is less than or equal to 1 mm, each reduction of 0.2mm size, the system processing time grid will increase nearly 10 times, the mesh quality can enhance the overall, but excessive waste of computing makes the work efficiency is greatly reduced, so the grid division is not more detailed more better; on the contrary, when the element size is equal to or larger than 3mm, mesh processing time basically in equilibrium.

Comprehensive consideration of the airborne nacelle stent design requirements, grid quality and the extent of division, establishing the optimal range change from1 to 3mm, the cell size is 2mm, and the number of nodes is 69690. Therefore, there are 320034 unit.

**FEM Model FEM Analysis Under the Environment of ANSYS Workbench.** Similarly, select ZL205A alloy treated by T6 in the material library, the density is 2.82g/cm3, Young's modulus: E=68GPa, Poisson's ratio: u=0.32.

**Meshing Parameters and Optimal Interval Selection.** Tetrahedron divided into two kind algorithms: Patch Conforming and Patch Independent, the biggest difference between the two algorithms is the model boundary processing, therefore, Conforming Patch is better choice, which is more similar with the NASTRAN solution. And sizing option parameters are defined in Mesh control option which under the Mesh project, after a number of test results, Element sizing is determined to 2mm, and Behavior is set to soft [4].

**FEM model parameter setting and solution scheme.** Reference the boundary conditions and loads in UG, the specific conditions of load is as shown in Fig. 2

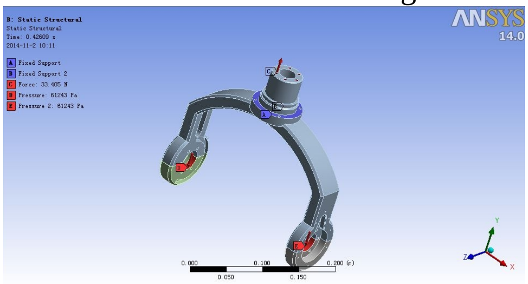
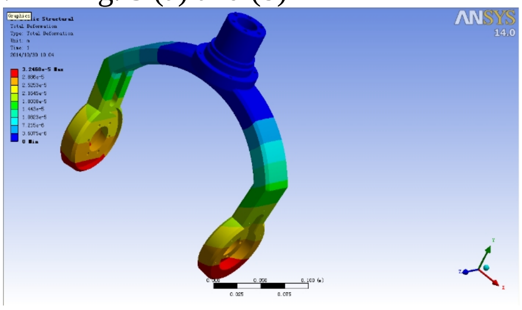
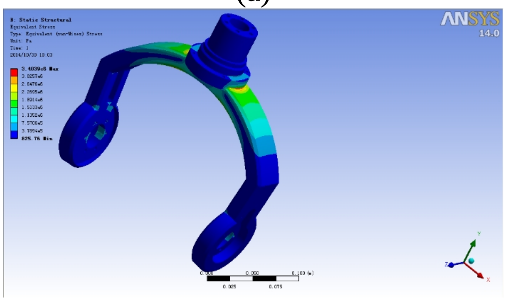



Fig. 2 Setting load parameters for airborne pod holder

**Pod solution schemes.** The equivalent stress and the total deformation of the node are processed by the ANSYS solver, as shown in Fig. 3 (a) and (b)




(a)




(b)

Fig. 3 The nephogram of node for ANSYS workbench

According to the two results of static analysis, Fig. 3 (a) shows that the maximum deformation of airborne optoelectronic pod reaches to 3.247×10-2mm. Similar with UG, the maximum deformation occurs in the lug of the pod bracket. Stress generated by motor torque concentrate on the inner side of the piston near the azimuth axis, the maximum value is 3.404MPa, and the position which most prone to fracture is located in the junction of the azimuth axis and the bracket arms. According to the ultimate tensile stress of ZL205A, the result meets the requirement.

## Conclusions

The parametric design based on UG for pod is more flexible and concise, and modification is extremely easy; compared to the ANSYS Workbench modeling system. The geometric features Interface of UG is friendly, parameters types are really rich, and comes with the CAD/CAE motion simulation module, making the geometric model for the inspection and assembly more targeted.

NASTRAN finite element analysis in the UG saves time of design and check, reduce production costs, providing a new way for the rationalization and reliability of parts design. Compared to the ANSYS Workbench, NASTRAN system has more advantages in terms of linear finite element analysis and dynamic calculation, according to the simulation data, the static structure analysis is basically consistent, but for the specific analysis of the data has bigger difference, further testing at the later stage is necessary.

ANSYS Workbench has spying function for the weak parts of finite element analysis, designers can quickly and effectively put forward improvement plan to ensure reliability, which provide a numerical basis for structure optimization and vibration safety inspection.

## Acknowledgements

This work was financially supported by the National Nature Science Foundation (No. 51475142), Key Technology R & D Program of Henan Province (No. 13A520232), and Major and Previous Pre-research Project of Henan University of Science and Technology (No. 2011CX016).

## References

[1] Xiuju Du, Changzhi Jia, and Zhaowei Dong: Chinese Journal of mechanical engineering, Vol. 44, (2008), No.8 p.123 (In Chinese).

[2] Yuejin Yuan, Yingying Xu, and Yanhua Zhang: *ANSYS Workbench 14.0 modeling and simulation technology and detailed examples* (Chemical Industry Press, China 2013) (In Chinese).

[3] Shuai Zhang, Jingyang Zhou, and Yansi Wang: ANSYS manufacturing automation, Vol. 34 (2012) No.3 p.107 (In Chinese).

[4] Paul Liu, Lianming Xia, and Yuan Tao: Mining & Processing Equipment, (2012), No.4 p.32 (In Chinese).