PAPER • OPEN ACCESS

Modeling Automation of Space Payload Experimental Cabinet Based on Secondary Development of Hypermesh

To cite this article: Sinian Huang and Anping Wang 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **538** 012033

IOP Conf. Series: Materials Science and Engineering **538** (2019) 012033 doi:10.1088/1757-899X/538/1/012033

Modeling Automation of Space Payload Experimental Cabinet Based on Secondary Development of Hypermesh

Sinian Huang[1,2,a] and Anping Wang[2]

[1] University of Chinese Academy of Science, Beijing 100049, China
[2] Technology and Engineering Center for Space Utilization, Chinese Academy of Science, Beijing 100094, China

[a] Corresponding author: huangsinian16@csu.ac.cn

Abstract. For the repeatability of the finite element analysis process of the electronic cabinets, thermal control units and other equipment inside the space payload experimental rack, this paper developed a modeling automation system for the cabinets based on the secondary development function of HyperMesh. The system uses Nastran as a solver, and comprehensively use Tcl/Tk language, HyperMesh API function, Process Studio process editor for development. Details the technology and processes in the development process of HyperMesh, providing a new method for the finite element modeling analysis of the space payload experimental cabinets and improving the simulation efficiency.

1. Introduction

The space scientific experimental rack is an important structure for installing payloads in the capsules of the spacecraft and an important part of the space application system [1]. The experiment cabinet contains modules such as electronic chassis, thermal control drawer, and electronic control unit, these modules are part of the space scientific experimental rack, a large number of finite element simulation analysis is needed in the design and manufacturing process to provide a basis for the test of the laboratory cabinet, shorten the development cycle and save costs [2]. In the finite element analysis, the main types of analysis includes static analysis, modal analysis, harmonic response analysis, transient response analysis, etc. The analysis process is basically the same. The pre-processing of finite element modeling analysis of space payload cabinets is usually carried out in HyperMesh. The main processes include geometric model import, geometric cleaning, meshing, material attribute assignment, unit attribute assignment and boundary load setting, etc. The model is established and the finite element solution file is output and imported into the corresponding solver for calculation.

The finite element analysis of the space payload cabinets has many repetitive tasks, large workload, the content is tedious and error prone. Based on the secondary development function of HyperMesh, the corresponding modeling automation system is developed. The system uses Nastran as the solver, which can greatly improve the finite element modeling efficiency of the cabinets model, reduce a lot of repetitive work, and avoid mistakes in manual modeling. HyperMesh is a finite element pre-processing software developed by Altair Corporation of the United States. It is widely used in the field of finite element analysis and has strong meshing ability [3], which can provide users with high quality and efficient meshing technology. HyperMesh allows users to use the Tcl (Tool Command Language) and API functions for secondary development, users can write scripts to perform specific functions. Nastran software is a series of products of MSC in the United States, it is widely used in the aerospace field for simulation analysis due to its high precision and rapid convergence in finite element analysis.

Based on the secondary development function of HyperMesh, and aiming at the finite element analysis process of the space payload cabinets, taking the Process Studio module in HyperWorks as the development environment and using Tcl/Tk and HyperMesh API function[4], a set of space payload cabinets modeling automation system using Nastran as the solver was developed. The key technologies and theories in the secondary development process of HyperMesh are introduced in detail, and the finite element analysis process of the cabinet is automated and streamlined.

2. Development of Space Payload Cabinet Modeling Automation System

2.1. The Secondary Development Technology of HyperMesh

HyperMesh secondary development uses Tcl/Tk as its standard development language, similar to C++ and Java, Tcl is a scripting language [5], with fixed syntax, good readability and no need to compile. It is executed at run time through the interpreter for sequential interpretation of each statement. Tk is Tcl's graphical interface toolkit, it is a set of tools for developing graphical interface applications, where users can create interface components that trigger corresponding command functions and implement corresponding functions [6]. The TcL/Tk can be used as a stand-alone application, and the corresponding functions can be easily implemented and called by writing scripts[7].

HyperMesh provides users with a large number of functions, through which the corresponding functions in HyperMesh can be achieved. In the secondary development process of HyperMesh, two commom functional functions are mainly used, one is Tcl Modify Commands, such as *defaultremeshsurf command function can be used for 2D mesh division; the other is Tcl Query Commands, such as hm_entityinfo can perform attribute query. With the help of the HyperMesh API function, the corresponding secondary development function can be completed.

2.2. Modeling Automation System Development Method

When performing finite element analysis of equipment such as space payload cabinets, it is necessary to clearly analyze the requirements and arrange the analysis process reasonably so that the analysis process is as standardized and efficient as possible. First, select a simple model for finite element modeling analysis, and fully consider the various problems encountered in the modeling process, and ensure the correctness and standardization of manual modeling. During the HyperMesh operation, each step of the operation commands is recorded in the Command.cmf file. The Command file is a standard ASCII file that records a series of operational commands during operation, the command stream strictly follows the HyperMesh function format. Through the finite element modeling analysis of the simple model, for each operation, the valid commands are extracted, and the command function is modified to conform to the Tcl/Tk syntax.

The process automation template is established through the Process Studio, the finite element modeling analysis process and the system operation interface are designed, and the corresponding Tcl code and function are written for the function corresponding to each node in the process tree. Process Studio is a process development tool in HyperWorks that provides users with a visual process editing environment, by writing a process tree to create a corresponding finite element modeling analysis process, the secondary development process is organized. Each node in the process tree has a specific process corresponding to it, by writing a corresponding operation interface, it can form an interaction with the user, which greatly facilitates the user's operation. The well-written modeling automation system is called by Process Manager in HyperMesh, the user can input the corresponding data according to the process prompt to complete the finite element analysis process, which greatly improves the efficiency of finite element analysis. As shown in Figure 1, it is the secondary development method flow of the modeling automation system.

IOP Conf. Series: Materials Science and Engineering **538** (2019) 012033 doi:10.1088/1757-899X/538/1/012033

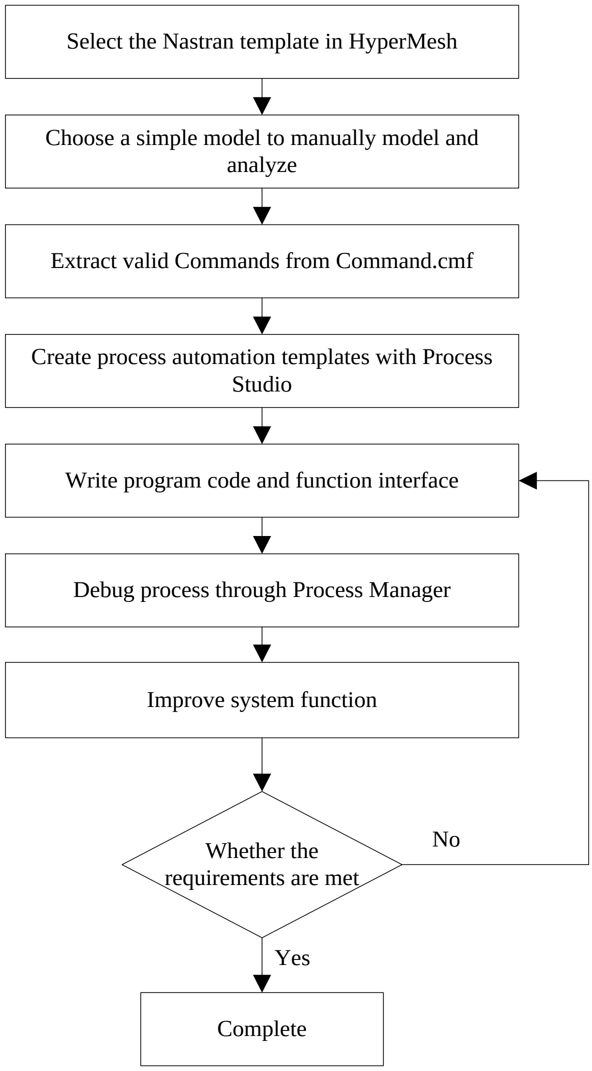

Figure 1. Modeling automation system secondary development method

2.3. Create Process Automation Templates with Process Studio

Manually complete the establishment of the finite element model and extract the valid command functions in the Command.cmf file. Create a process automation template through the Process Studio and add a Task to each function node in the modeling analysis in Process Studio, each node has a corresponding GUI interface. Each of the individual Tasks is written separately to performs a specific function.

Process Studio provides users with different utils control types, users can choose different types of controls for GUI interface design. There are two ways to implement data connections between utils and Tcl scripts. One way is to implement specific function directly by Tcl Scripting, for example, to get a data named “Text field” text box, you can use program statements: set strValue [::hwpmgr::PmgrGetData 0 “Text field.value”]; Another way is to directly call the Tcl program file that has been written through the source command.

3. Application of Modeling Automation System

For the most common electronic cabinets in the space scientific experimental rack, different modeling automation systems were developed according to the different types of finite element analysis requirements of the cabinets. For the common electronic cabinets modal analysis process, based on

HyperMesh's secondary development function, the modeling automation system using Nastran as the solver was developed by Process Studio to conduct modal analysis.

3.1. Get Command Stream

The command stream in the Command.cmf file is obtained by manual modeling and modified to conform to the Tcl/Tk syntax. The command stream includes automatic import of geometric models, automatic creation of connections, automatic addition of washers to holes and automatic creation of materials, etc. Some of the command streams are shown below.

```
# Import .stp geometry model file
*feinputwithdata2 “#ct/step_reader”
# Add a washer to the selected hole
*trim_by_offset_edges 0 1 $r 0 0 0
# Meshing and the grid order is 1, the size is $ELE_SIZE
*defaultremeshsurf 1 $ELE_SIZE
# Set the material property to MAT1
*dictionaryload materials 1 "D:/Program Files/ templates/feoutput/nastran/general" "MAT1;
# Define unit property
*dictionaryload properties 2 "D:/Program Files/ templates/feoutput/nastran/general"
# Set the modal analysis card and output the first 10th order frequency
*dictionaryload loadcols 2 " D:/Program Files/ templates/feoutput/nastran/general" "EIGRL";
*createmark loadcols 1 " EIGRL"
*attributeupdateint loadcols 2 804 1 1 0 10
……
# Output bdf and hm files
*feoutputwithdata
*writefile
```

3.2. Create Analysis Process and GUI Interface

According to the modal analysis process, Process Studio is used to establish the process automation template of the cabinets modal analysis to develop the modeling automation system. Figure 2 shows the electronic cabinets analysis system process by Process Studio.

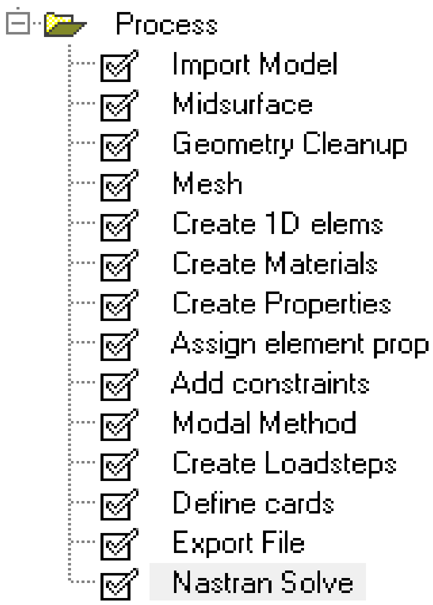

Figure 2. Electronic cabinets analysis system process

In the process structure, each node corresponds to a specific function, after the user completes the operation of a function node, the system automatically jumps to the next function node to guide the user to complete the analysis process. Reducing the number of user switching interfaces and improving the analysis efficiency. Through this modal analysis analysis process, users can perform operations such as geometric model input, automatic creation of connections and meshing.
In the process structure, each function node has a corresponding function operation interface, the user can complete the data input and other operations through the designed function interface. As shown in Figure 3, it is the meshing interface of the modeling automation system. After selecting the specified area and inputting the grid size, two-dimensional grid can be automated, and the grid quality can be checked. It avoids the repeated switch of interface and improves the efficiency of meshing.
After the meshing is completed, material creation and attribute assignment are performed, as shown in Figure 4, which is a material creation interface in the automatic modeling system. In the material creation process, the user only needs to input the material, name, elastic modulus, Poisson’s ratio and density, the materials can be created automatically, which greatly simplifies the process.

Figure 3. Meshing interface

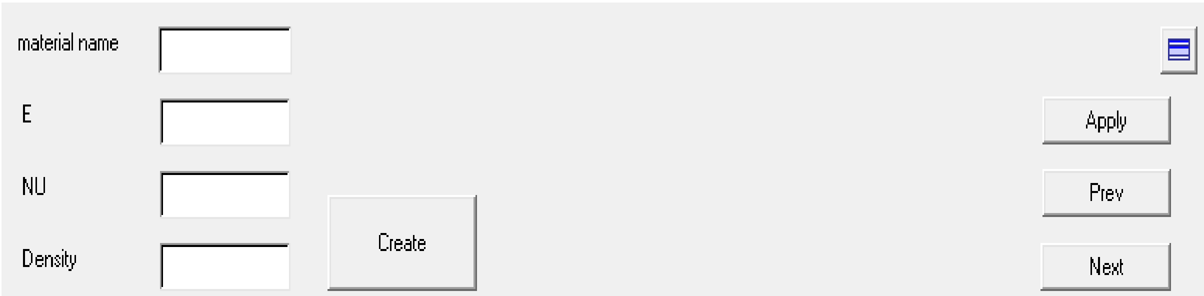

Figure 4. Material creation interface

In the development process of the modeling automation system, for some functional interfaces that need to be repeatedly switched, it is inconvenient to write the function of the program, and the original function panel can be directly called. For example, if call the command to add washer in HyperMesh, it can use the hm_setpanelproc {hm_callpanel “quick edit”} to call the original function panel directly.

3.3. Application of Modeling Automation System

The modal analysis of the space payload cabinet is carried out by modeling automation system, setting the User Profiles to Nastran in HyperMesh, and calling the system through the Process Manager, the payload modal analysis process can be performed.
Taking the payload cabinet shown in Figure 5 as an example, the 3D geometric model of the cabinet is imported into HyperMesh. The user can be complete to operation of each function node according to the guidance of the modeling automation system, and the entire element analysis process can be successfully completed. According to the system’s prompt, the finite element model file and the bdf file with Nastran as the solver can be exported. As shown in Figure 6, the finite element model is established by the cabinet modal analysis system.

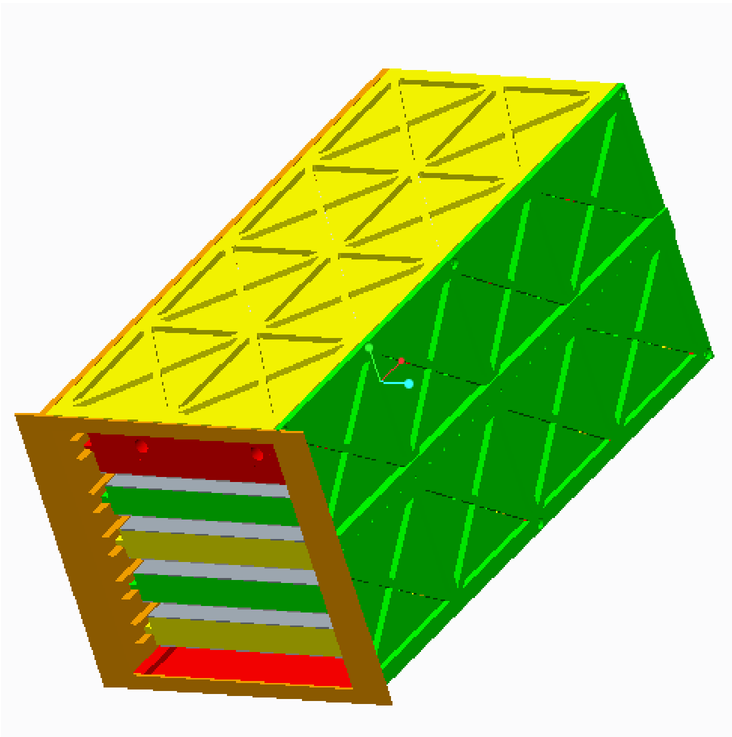

Figure 5. Geometric model of the space payload cabinet

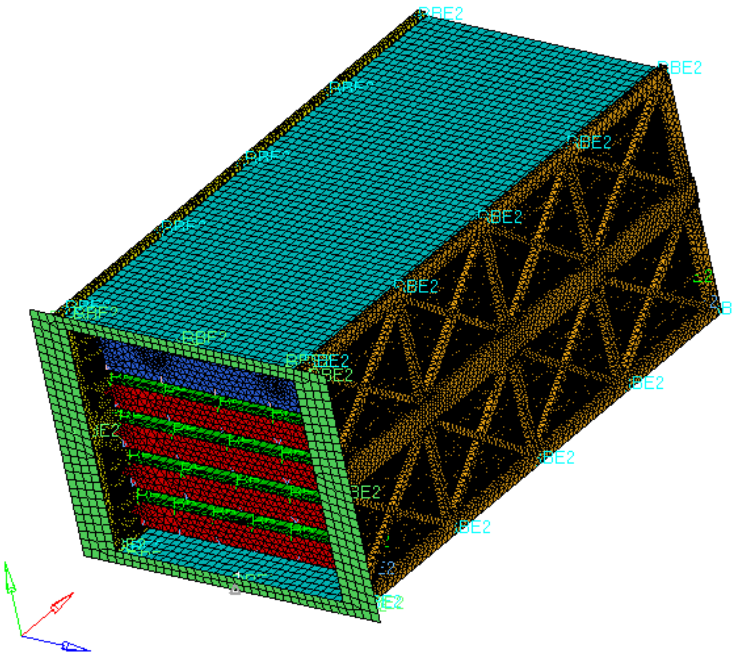

Figure 6. Finite element model of the space payload cabinet

By directly calling solving software Nastran, the finite element model can be automatically solved, and the solution results are saved in the corresponding folder. Next the payload cabinet is post-processed after the solution is completed, the frequency of modal analysis under fixed constraints can be obtained. As shown in Figure 7, it is the first four order modal shapes of the payload cabinet.

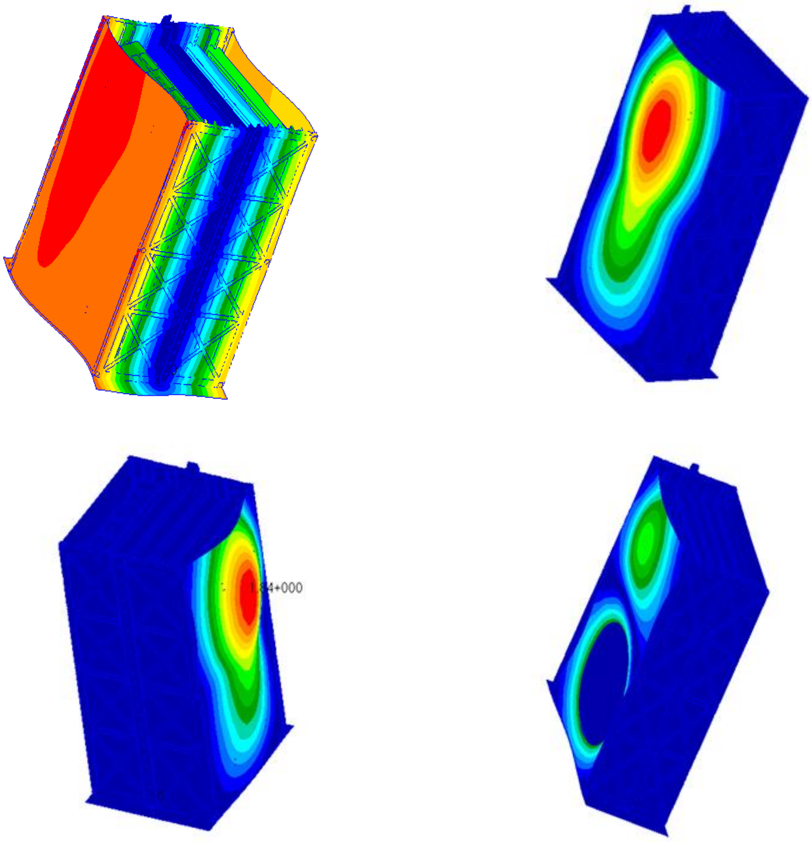

Figure 7. The first four order mode shapes

4. Concluding Remarks

This paper introduces the development method and key technology of space payload cabinets modeling automation system based on secondary development of HyperMesh. For the modules with similar geometry and consistent analysis of the space payload cabinets, comprehensively use Tcl/Tk language, HyperMesh API function, Process Studio to develop the modeling automation system, which provide a new method for finite element analysis of cabinets. The specific application examples are given to verify the feasibility and effectiveness of the method. The flow chart and program of the finite element analysis of the cabinets model are realized, and the modeling efficiency is improved.

References

[1] Wang Dapeng. Optimal Design of Payload Cabinet Structure[J]. Spacecraft Engineering, 2010, 19(2):34-39.
[2] Ding Tao, Peng Wang. Application of HyperMesh Secondary Development in Pre-processing of Bus Finite Element Analysis[J]. Bus Technology and Research, 2015,6:47-49.
[3] Zhu Feng, Li Shuxiao. Modal Analysis of Vehicle Exhaust System by Hypermesh and Workbench[J]. MECHANICAL ENGINEERING & AUTOMATION, 2014,1:62-64.
[4] Sun Jing, Huang Xuefei, et al. Develpment and application of Process Automation Systems based on HyperWorks[J]. RAILWAY COMPUTER APPLICATION, 2012,21(12):30-33.
[5] Yang Jindian. Automated Modeling Procedure of Semi-steel Tire in Finite Element Simulation[J]. Jinan : SHANDONG UNIVERSITY, 2015.
[6] Ousterhout J K, Jones K. Tcl/Tk Introduction to Classic[M]. 2nd Edition. Zhang Yuanzhang, Trans. Beijing : Tsinghua University Press, 2010 : 3-6.

[7] Lu Shanbin, Lv Jie, Chen Wei, et al. Finite Element Fast Modeling of Clinching Joints Based on HyperMesh Secondary Development[J]. JOURNAL OF GRAPHICS, 2014, 35(5):804-808.
[8] Liu Jianxiao, Li Bin, Du Chong. Modeling and optimization analysis on aircraft empennage composite based on secondary development of HyperMesh[J]. Computer Aided Engineering, 2016, 25(3):29-33.
[9] Narasimaha Rao, Vamsi Krishna. Automation for Material and Property Values in HyperMesh Using TCL[J]. Simulation Driven Innovation, 2012.
[10] JERZY KRUKOWSKI, Application of the Rigid Finite Element Method for Modeling an Offshore Pedestal Crane[J]. THE ARCHIVE OF MECHANICAL ENGINEERING, 2013.
[11] Faycal Benyahia, Abdulmohsen Albedha. Elliptical and Circular Bonded Composite Repair under Mechanical and Thermal Loading in Aircraft Structures[J]. Materials Research, 2014, 17(5): 1219-1225.
[12] Marc Alexander Schweitzer. Generalizations of the Finite Element Method. Central European Journal of Mathematics, 10(1), 2012, 3-24.