



Technical research on optimization design of contacts of electrical connector

LV Bin[†], ZHOU Sheng-jun, ZHAO Lan-yan

(Hangzhou Aerospace Electronic Technology Co., Ltd, Hangzhou 310015, China)

[†]E-mail: apporol1106@yahoo.com.cn

Received Dec. 19, 2006; revision accepted Jan. 12, 2007

Abstract: Pin and socket contacts are the key parts of electrical connector as aerospace electronic components. The contacts are also the direct carriers for signal transmitting of electronic equipments, passing the signal from the input end to the output end of electrical connector. The reliability of pin and socket contacts directly influences signal transmission. The goal of this study is to enhance the contact reliability of aerospace electrical connector. Computer simulation analysis was made on contacts performance data by utilizing the simulation system developed by PCL Language of MSC software. Furthermore, the results were experimentally validated so as to realize the objective of optimizing contacts design.

Key words: Electrical connector, Contacts, Simulation, Optimization design

doi:10.1631/jzus.2007.A0506

Document code: A

CLC number: TM5

INTRODUCTION

Contacts, the critical parts of aerospace electrical connector, are direct carriers for signal transmitting of electronic equipments, passing the signal from the input end to the output end of electrical connector. The reliability of contact performance directly impacts the signal transmission. Due to the special and important role the contacts play in the connector, they may result in particularly severe failure in the parent product. Therefore, to study the pin and socket contacts and to find ways to optimize their design is of great significance in improving the reliability of electrical connectors.

DEVELOPMENT OF CONTACTS SIMULATION SYSTEM

Based on MSC/Patran interface, contacts simulation system (Ilxy, 2006) applies MSC.Marc to analyze contact force and socket stress. MSC.Marc (MSC & SOFTWARE, 2003) is a full-functioned analysis module and evaluator by MSC Software

Company for high nonlinear finite elements, with perfect combination of theoretical method and software practice. It has great function in analyzing various linear and nonlinear structures, with the features of high value stability, high precision and rapid constringency. To further improve calculation precision and analysis efficiency, MSC.Marc provides a number of great self-reacting techniques for element step extension, e.g. automatically analyzing the appropriate step extension of inflection, squirm, thermal elasticity and drive (Jia and Dai, 2006). What is more, it has excellent grid self-reacting technique to automatically adjust grid density by many rules of error, which cannot only enhance the analysis precision of large linear structure, but also provide optimized grid density for the analysis of local nonlinear stress concentration (Du, 2006), moving boundary or contact. Thus it has dramatically improved the calculation efficiency of nonlinear analysis at the same time ensuring the calculation precision. In addition, MSC.Marc supports the automatic repartition of 2D and 3D grid to correct the grid aberrance caused by excessive deformation in order to ensure the continuity of large deformation analysis. Meanwhile, MSC.Marc

features a convenient user interface by which user can develop it anew if needed and can access or modify the defaults of programs from geometry modeling, grid plotting, boundary definition, material selection to evaluation and results output. On the basis of the original functions of MSC.Marc, the user may greatly develop its capability of finite element analysis. Besides supporting single CPU analysis, MSC.Marc can make mass parallel processing (Guo *et al.*, 2006) with multi CPU or multi network nodes on NT or UNIX platform. The unique calculation method of parallel finite elements featured by MSC.Marc is based on the area partition method and can realize the parallelism to the greatest extent during the process of finite element analysis, whose efficiency is up to potential linearity, even linearity or super linearity. Thanks to its powerful calculation capability of parallel processing, MSC.Marc can give more prompt, more detailed and more accurate results of simulation for virtual products running and machining.

Contacts emulation system mainly includes two parts of pin and socket. In the use of electrical connector, pins and sockets need to engage each other many times and be bonded firmly to ensure the circuit activity. So, the mating and unmating force of the contacts is required to be in the permissible range with each insertion and separation. But in practice, it is always hard to absolutely reach the intended mating and unmating force value, which may result in such failures as relaxed stress of socket and hence degrade the contact performance and the reliability of electrical connector. According to our tests, this problem is mostly due to the improper dimension of socket slot, or socket excessive deformation after being closed up causes it to be beyond the elastic limit of the material. In our former practice, we made a series of tests to look for a solution in this case. But now we hope we can actually see the contacts stress distribution when inserting and releasing by contacts simulation system, and consequently achieve higher contact reliability of product in design and development.

DEFINITION OF FINITE ELEMENT MODEL OF SIMULATION SYSTEM

Contacts simulation system is the newly-developed software on MSC.Nastran and MSC.Marc

platform and applies sequential linear programming (SLP) to make optimization calculation. The main idea of sequential linear programming is to expand the nonlinear (Diehl, 2005) target function and limitation at the original point by Taylor series, and then to select linear items whose result is subsequently obtained by SLP. If the result does not meet the precision requirement of design, the former nonlinear target function may be expanded again by Taylor series at the acquired approximate result to get new linear items. Thus repeatedly make iterative calculation until the acquired result meets the precision requirement of design (Chen *et al.*, 2006).

In contacts simulation system, the generation of mating and unmating force is due to the deformation caused by the contact between pins (rigid) and sockets (elastic) during their insertion and separation. To describe this process by finite element method, it is required to set the following definitions.

To define finite element model for pins and sockets

In order to make convenient the analysis of contacts design, a basic finite element model of contacts should firstly be established (Qiu and Lin, 2006). Thus only the appropriate parameters of contacts dimension need to be changed when analyzing contacts. To simplify and validate the model, combining the generation mechanism of the mating and unmating force of contacts system, we treat the pin part as a purely rigid body and describe it as surface, a simple geometric concept, while we treat the elastic socket part as a deformation body and describe it as a 3D block unit. Furthermore, for reducing the computer processing time, 3D unit blocks apply hexahedral grid plotting with even grids for surface and sparse grids for inner part which can achieve the high efficiency of finite element model analysis at the same time ensuring that the analytic data of contacts meet the required precision (Liang, 2006).

To define the material properties and geometric properties

Considering material nonlinearity, in many cases the parameters and boundary conditions of material may change because of one or more factors (Ge, 2006). In addition, some data of material may not be constants but functions of some variables such as time or temperature, in which case a curve should be given

in advance. In the model, we have established a database of contacts material properties such as elastic modulus, Poisson's ratio, yield stress, etc.

In the definition of geometric characteristics, pin is treated as a simple shaft unit while socket is a cantilever beam unit. As a 3D entity unit, the inner free degree of the socket is appropriately reduced.

To define the contact boundary conditions

Boundary conditions have different definitions for different problems, not only for use on unit element but also on geometric element relating to the unit. In this finite element model, the deformation and stress of the socket (deformation body) during contact is a group of regular units set and is defined as 3D block unit in this paper. And deformation body boundary is defined as the whole contact of socket (deformation body) and pin (rigid body).

The deformation of the pin (rigid body) during contact may be neglected due to its small impact on analytic results. So it is enough to describe the contacts set only using the outer profile of the rigid body that is called surface in this paper. With a given speed, the movement of pins is defined as rigid movement with equal speed.

SIMULATION AND OPTIMIZATION

To establish finite element model

To establish finite element model, you only need to open the MSC software, select the menu of contacts simulation and then click the <Establish finite element model> button in the main interface. In the window for establishing finite element model, please input appropriate parameters of the contacts model such as outer and inner diameters of socket, slot width, slot depth, bore depth, pin diameter, etc. After that, click the <OK> button and the finite element model of contacts simulation analysis appears (Fig.1).

To add boundary conditions

On the window shown in Fig.1, click the <Add boundary conditions> button and you can access the interface for adding boundary conditions. According to the contents of optimization, it is divided into mating and unmating force optimization and residual stress optimization.

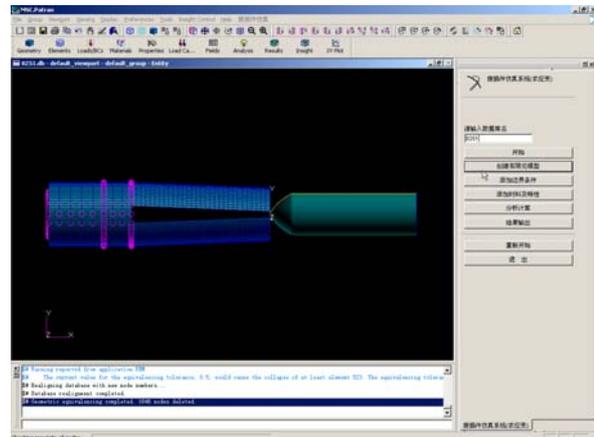


Fig.1 Finite element model of contacts simulation analysis

1. Mating and unmating force optimization

The system provides two types of boundary conditions, i.e. displacement and contact. The planar movement and rotation in displacement boundary conditions are provided as defaults and do not need to be input. But user should input the friction coefficient of pin and socket in the boundary conditions of socket contact and select the whole socket as the action point. Select the whole pin as the action point in the boundary conditions of pin contact and the speed is given by system. After the parameters input, click the <OK> button and finish the establishment of the boundary conditions for mating and unmating force optimization.

2. Residual stress optimization

It is different for residual stress optimization. Comparing with mating and unmating force optimization, it should add the pressure boundary condition because it needs to load pressure on socket. Input an appropriate pressure value in the "Pressure" dialogue box and then select a certain action area.

To add material properties

Following the above steps, we should define the material properties of contacts (Zou, 2006). Input the material name and the parameters of yield stress, Poisson's ratio and Young's modulus respectively in the corresponding input boxes. Because aerospace electronic component has its own distinction, its materials are always purchased by fixed procurement and the material data are provided directly by the supplier, which can ensure the accuracy of material property data. After inputting the parameters, click

the <OK> button to confirm the property data. Subsequently, click the <Return> button to access the main menu to carry on the next step.

Analysis and calculation

After clicking the <Analysis and Calculation> button in the main menu, the model established in the prior processor MSC.Patran will automatically be passed to MSC.Marc for the iterative calculation (Wang, 2006).

Results import

After calculation, you can observe the analytic results. Click the <Result export> button to access the interface shown in Fig.2. Input the name of the result file desired to import the corresponding box for file name, or select the result file to import (The file extension is t16). After clicking the <Import file t16> button, the result file will be imported into the system and all the working states will be shown in their select field. Select a working state and see the relevant results on stress distribution, deformation, mating and unmating force, etc., for further analysis.

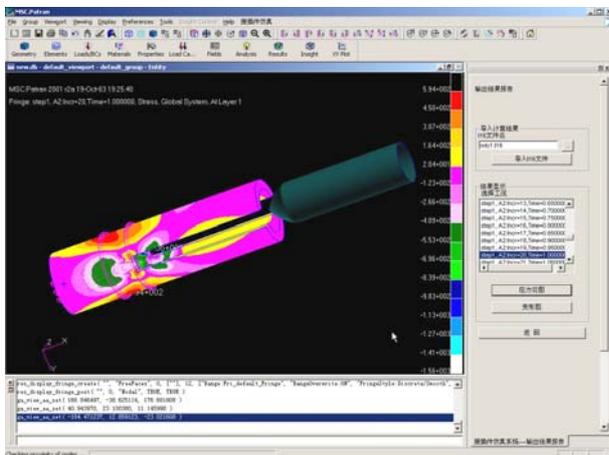


Fig.2 Result stress diagram

gold plating, and its finish is 1.6. In this simulation test, the slot width is selected as a medium value of 0.4. See Table 1 for the detailed parameters of finite element model, and Table 2 for the material properties. The boundary conditions: Friction coefficient=0.3.

See Fig.3 for the exported result.

Table 1 Finite element model (unit: mm)

Parameter	Value
Socket outer diameter	Ø2.3
Socket inner diameter	Ø1.6
Slot width	0.4
Slot depth	5.0
Bore depth	6.0
Pin diameter	Ø1.5
Contact length	8.0
Chamfer angle of bore entry	0.25
Deformation	0.4

Table 2 Material properties

Yield stress	Young's modulus	Poisson's ratio
430	200000	0.35

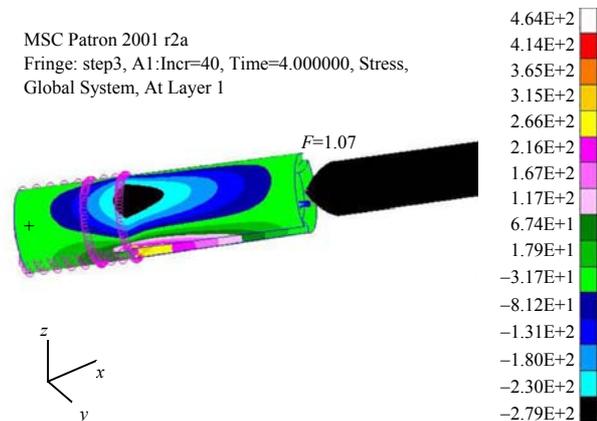


Fig.3 Analytic result export

CONTACTS SIMULATION TEST

For this contacts simulation system, we select the typical socket Y11-0000-42 of our company for the simulation test. 15 sockets are sampled at random. After closed up, the mating and unmating force is in the range of 0.8~1.2 N, and the measured slot width is between 0.35~0.45 mm. The machining inner bore is

CONCLUSION

Comparing the test result with the actual status, it is found that both basically conform, which indicates the contacts simulation system has certain value in application. Therefore, this simulation system can

give guidelines for the optimization design of contacts and has great significance in enhancing the reliability of electrical connectors.

References

- Chen, J., Deng, Q., Zhang, W.G., 2006. The Reasons and Correction for the Errors Occurred in Rigid Plasticity Finite Element Numerical Simulation. [Http://cad.newmaker.com/disp_art/1310002/17081.html](http://cad.newmaker.com/disp_art/1310002/17081.html)
- Diehl, T., 2005. How to address the hard issue of nonlinearity. *Design News China*, **2**(3).
- Du, P.A., 2006. Basic Principles for Finite Element Grid Plotting. [Http://cad.newmaker.com/disp_art/1310003/11921.html](http://cad.newmaker.com/disp_art/1310003/11921.html)
- Ge, S., 2006. The Units in Finite Element Analysis. [Http://cad.newmaker.com/disp_art/1310002/16069.html](http://cad.newmaker.com/disp_art/1310002/16069.html)
- Guo, J.N., Hu, J.X., Lu, Z.D., 2006. Parallel Area Partition Method for Auto-generated Finite Element Grids. [Http://cad.newmaker.com/disp_art/1310003/10006.html](http://cad.newmaker.com/disp_art/1310003/10006.html)
- Ilxy, 2006. Nonlinear finite element analysis of concrete filled rectangular steel stubs. *Simwe Electronics Periodical*, **8**:15-22.
- Jia, X.M., Dai, F., 2006. Three-dimensional static and dynamic stress intensity factor computations using ANSYS. *Simwe Electronics Periodical*, **1**:5-16.
- Liang, X.H., 2006. Mobile phone structure collision simulation applying the finite element technique. *Simwe Electronics Periodical*, **8**:23-27.
- MSC & SOFTWARE, 2003. Technical Report on MSC.Marc Software.
- Qiu, Z.D., Lin, C.Z., 2006. Finite Element Analysis Technology for Macro Structure in Engineering Mechanical Design. [Http://cad.newmaker.com/disp_art/1310003/8534.html](http://cad.newmaker.com/disp_art/1310003/8534.html)
- Wang, H., 2006. Elasticity and plasticity computations for discretional polygon configuration based on parametric variation principle. *Simwe Electronics Periodical*, **8**:36-46.
- Zou, Z.G., 2006. Material performance unit in finite element analysis. *Simwe Electronics Periodical*, **1**:1-4.



Editor-in-Chief: Wei YANG
ISSN 1009-3095 (Print); ISSN 1862-1775 (Online), monthly

Journal of Zhejiang University

SCIENCE A

www.zju.edu.cn/jzus; www.springerlink.com
jzus@zju.edu.cn

JZUS-A focuses on "Applied Physics & Engineering"

► Welcome Your Contributions to JZUS-A

Journal of Zhejiang University SCIENCE A warmly and sincerely welcomes scientists all over the world to contribute Reviews, Articles and Science Letters focused on **Applied Physics & Engineering**. Especially, Science Letters (3–4 pages) would be published as soon as about 30 days (Note: detailed research articles can still be published in the professional journals in the future after Science Letters is published by *JZUS-A*).