Simulation Exploration of Assembly Process and Key Parameters of TWT

Xiaofang Zhu [1], Jingyuan Che [2], Yulu Hu [3], Quan Hu [4], Bin Li [5], Tao Huang [6], Xiaolin Jin [7], Li Xu [8]

National Key Laboratory of Science and Technology on Vacuum Electronics
Academy of Computer Simulation Technology, University of Electronic Science and Technology of China
NO.4, Section 2, North Jianshe Road, Chengdu, P. R. China 610054
xf@uestc.edu.cn [1]; cjy1995@126.com [2]; yuluhu@uestc.edu.cn[3]; huquan1981@uestc.edu.cn[4]; libin@uestc.edu.cn[5]; huangtao@uestc.edu.cn[6]; jinxiaolin@uestc.edu.cn[7]; lixu@uestc.edu.cn[8].

Abstract*: Assembly process and key parameters have important effects on performance of traveling wave tubes. In this paper, simulation exploration on Graphite heat extrusion process and the cathode-heater assembly distance of electron gun are introduced, which have been carried out during the last two years and is to be used to guide the assembly process in TWT. More simulation exploration in processing are still in progress.*

Keywords: thermal deformation, assembly processing, simulation exploration, graphite heat extrusion process

Introduction

The machining and assembly technique of Travelling Wave Tubes (TWTs) have great influence on performance. Machining details such as processing duration and control temperature will affect geometrical parameters. The assembly error caused by thermal deformation will lead to performance degradation of travelling wave tubes.

There are many kinds of processing technology involved during manufacturing process of TWTs, such as the mechanical elastic method, thermal stress force method and welding method. To establish an appropriate processing and assembly flow for certain TWT, a large number of experiments are required to determine processing details. Using computer aided simulation technology to accurately simulate the assembly process and determine the key parameters will help to shorten the development cycle of the device, greatly reduce the experimental cost and ensure the device performance. Take the thermal stress force method as an example, by simulation of the processing flow, processing details as control temperature and duration can be proposed. With only a few test, the processing flow can be established and used for the manufacturing of certain TWT.

For the processing flow of a certain method is very complex and hard to simulate, we need to find a way to duplicate the main process and observe the corresponding results so that we can determine the processing details. Considering the importance of computer aided simulation in TWTs, a lot of simulations have been done in the whole process of design, processing and assembly. Here, we are focusing on simulation of processing methods widely used in manufacturing of TWTs and trying to get key assembly parameters by simulation.

Simulation of Graphite heat extrusion process

The Graphite Heat Extrusion Process is a kind of high-frequency structure assembly technique which is widely used in space helix TWTs. In this method, the graphite shell is used to fix the outer shell of the structure during the processing for its low expansion coefficient.

The pre-assembled helix, support rod and shell are inserted into graphite shell and are put in the processing device as a whole. The temperature in the processing device is increased to a designed value and maintained for a period of time. Then the processing device is gradually cooled to room temperature. In the heating process, the high frequency structure will deform due to thermal expansion. For the low expansion coefficient of graphite, the shell of the high frequency structure will expand inwardly and squeeze the rods and the helix to make them contact closely. In the cooling process, the shell shrinks and the internal components are further compressed.

To simulate the Graphite Heat Extrusion Process, which includes both heating and cooling process and thus cannot be simulated in a simulation, the whole processing flow are separated into heating process and cooling process and are simulated respectively in ANSYS Workbench. For the model after heating process is somewhat damaged, the thermal damaged model is repaired in SpaceClaim before the cooling process simulation.

Once the simulation is ready, the transformed model can be observed and the changes of geometrical parameters can be extracted. With the information, the effects of the processing on the electrical performance can be further studied. As well, processing details such as temperature and duration of heating and cooling process can be optimized according to the specifications.

To demonstrate the simulation method proposed here, the Graphite Heat Extrusion Process of a helical high-frequency structure for 5-8GHz helix TWT are simulated. The details of the simulation are introduced and the consequence of the processing is analyzed. The helical high-frequency structure is illustrated in Fig.1.

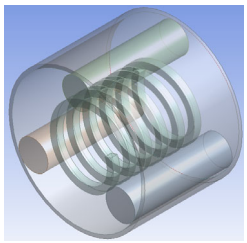
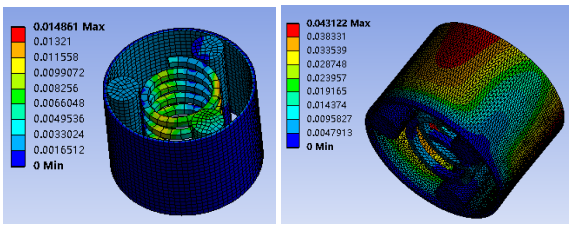


Fig. 1 Illustration of the helical high frequency structure

In the heating process simulation, we take the heating temperature as 700°C and the ambient temperature rises from 22°C to 700°C during 100 seconds. Keep 700°C for another 100 seconds. Once the simulation is ready, the

processed model saved as STL format is exported to SpaceClain of ANSYS where the mesh fragments are repaired. Saved as sat format, the repaired model is now ready for further processing.

When simulating the cooling process, the ambient temperature decreases from 700℃ to 22℃ during 100 seconds. Keep 22℃ for another 50 seconds.



(a) heating process (b) cooling process

Fig. 2 The high-frequency in heating and cooling process

To study the effects of the deformation of the structure, we exported the final deformed model into SpaceClaim and extract the key geometrical parameters for further analysis.

After simulation of the Heat Extrusion Process, the inner radius of the shell and the helix inner radius decreases. It implies that the Heat Extrusion Process can make the assembly of the high-frequency structure more compact. The dispersion and impedance performance before and after processing can also be analyzed. For more details, please refer to "A Simulation Method of Graphite Heat Extrusion Process for High-Frequency Structure of Helix TWTs" [3].

Simulaton to determine cathode - heater assembly distance

Electron gun is a most import part of TWT whose performance is closely related to TWT. As a precise and complicated part, the assembly error caused by thermal deformation will seriously affect the performance of electron gun. Among the assembly parameters, the cathode-heater assembly distance needs to be designed carefully. With improper design, contact damage may occur or heating efficiency may decrease due to the thermal deformation of the cathode and the heater in the working operation.

As a traditional way, a large number of experiments are required to get the optimal cathode-heater assembly distance, which is time-consuming and wasteful. With the development of computer simulation technology, computer aided design is suggested as a pioneer. However, for the thermal deformation of every part of electron gun is generally irregular, the cathode-heater assembly distance is approximately obtained by observing the deformed model and is not accurate enough. To some extent, this explains why it is hard to get the optimum cathode-heater assembly distance, ensuring high heating efficiency and no contact occurring in operation of electron gun.

A simulation method is put forward to optimize the cathode-heater assembly distance. In this method, thermal analysis of electron gun is performed in ANSYS and the deformed model of the cathode and the heater are repaired and transformed from STL to SAT format in SpaceClaim. The deformed models with SAT format are then exported into CST, where the cathode-heater distance is accurately measured by gradually moving the heater towards the fixed cathode until contact alarm rises. The accuracy of this distance measurement can be controlled within 1 μm. For detailed process and simulation results, please refer to the submitted article on IVEC 2020 entitled "A Simulation Method to Determine the Assembly Distance between Cathode and Heater of Electron Gun".

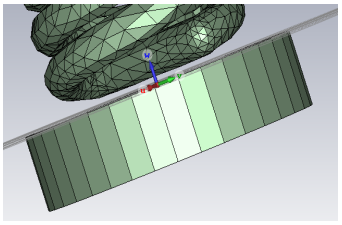


Fig. 3 Distance measurement in CST

Conclusion

This paper introduces application of simulation in the area of machining and assembly of TWTs. By separating the Graphite Heat Extrusion Process into heating and cooling process, simulation of Graphite Heat Extrusion Process is realized in ANSYS and the effect of the processing can be further analyzed with MTSS. Combing ANSYS and CST, the cathode-heater assembly distance can be accurately determined and optimized, leading to minor deformation and higher heating power efficiency.

Acknowledgment

This work is supported by National Natural Science Foundation of China (Grant No. 61771105 and 61921002) and Fundamental Research Funds for Central Universities (Grant No：2672018ZYGX2018J037).

References

[1] Jiufu Ruan. Thermal deformation of gridded electron gun in traveling wave tubes [J]. High Power Laser and Particle Beams, 2013, 25(2): 423-426

[2] L.X, Lu. Assembly performance simulation and optimization of helix slow wave structures based on manufacturing characteristics [D]. 2016.

[3] Jingyuan Che. A Simulation Method of Graphite Heat Extrusion Process for High-Frequency Structure of Helix TWTs [C]. IVEC, Busan, South Korean, April 29- May 1, 2019 .