Numerical Simulation of 3-D Temperature Distribution of the Flame Tube of the Combustion Chamber with Air Film Cooling

Chang Haiping   Huang Taiping*   Chen Wanbing
Department of Power Engineering, Nanjing University of Aeronautics & Astronautics, China

The wall temperature distribution of the flame tube of the combustion chamber is strongly affected by the combustion, radiation and flow. The interaction of these influential factors forms a coupling system. In this paper, a new method, which is different from the previous methods, has been developed for calculating the temperature distribution of the flame tube wall together with the flow field inside and outside the flame tube. In the calculation, the combustion, heat radiation, cooling air film and injection stream mixing inside the flame tube as well as the secondary air flowing outside the flame tube have been simulated. The calculation, in this paper, uses the SIMPLE algorithm, the k-e turbulence model and the auto-adjustable damping method. By using this method, the 3-D temperature distribution of the flame tube wall of the combustion chamber of an aeroengine has been simulated successfully. The calculation results are compared to the experimental data. The error of wall temperature is less than 10%.

Keywords: numerical simulation, temperature distribution, flame tube, turbulent model, auto-adjustable damping method, coupling system.

INTRODUCTION

The high temperature combustion chamber is widely used for the high performance gas turbine aero-engines nowadays and will also be in the future. The durability of the combustion chamber hot section is one of the important problems because of the high temperature surrounding and erosion of the gas. A significant aspect, which influences the durability of the hot section considerably, is predicting the thermal state of the flame tube of the combustion chamber accurately, and further, offering the scientific basis for adjusting and improving the thermal state of the flame tube. This has been directing the researcher's attention.

In the past, the investigation of the combustion chamber is undertaken by elementary analysis, experimentation and a great quantity of tests on components and full scale test-bed. It is almost impossible to measure the combustion and flow field in the combustion chamber directly, because the flow is very complicated and operates in high temperature surroundings. The numerical simulation method, which has been fast developed, offers an effective supplementary tool for the design process of the combustion chamber. It will play a significant and active role to the design capability in the future, and has been arousing the attention of the engineers nowadays. NASA presented the Turbine Engine Hot Section Technology (HOST) in 1980's and systematically investigated the durability problem of the hot section, including the evaluation of the thermodynamic model in the combustion chamber and the numerical method for the complex flow field in the combustion chamber etc. It has played an important role in the design of the hot section and the assurance of its durability. In the recent ten years, a great progression has been made in the field of numerical calculation of the combustion chamber[1].

One of the key points, which considerably influences the durability of the combustion chamber hot section, is the wall temperature of the flame tube.
The wall temperature distribution of the flame tube is strongly influenced by the combustion, radiation and flow as well as the cooling and mixing jets (secondary air flow). The interaction among these influential factors forms a coupling system. In case the distribution of the heat generation rate of the combustion chamber, the flow field and the jet mixing can not be correctly described, the wall temperature of the flame tube will be unable to be predicted exactly. Previously, the following methods were used for predicting the wall temperature of the flame tube of the combustion chamber:

(1) Calculate the convection behavior and heat radiation at the inner and outer walls of the flame tube by using empirical formulae. This method is not precise enough, and can not fulfill the modern design requirements;

(2) Numerical calculation by using a model with simple geometrical shapes. This method obliterates the character of the flame tubes, so it is not practical for engineering. Generally, it is just suitable for the verification of theoretical study.

(3) Numerical simulation by component regions and segments. This method is widely used in the numerical calculation with high precision such as in the general program NASTRAN etc. Although it is a good idea that the coupling problem is treated by component regions and segments, not all of the complicated coupling problems can be treated by this way. One exception is the calculation of the wall temperature of the flame tube of the combustion chamber. If the numerical simulation by component regions and segments is used, the influence of the complex flow field and radiation on the wall temperature of the flame tube of the combustion chamber will be difficult to be described. So that, the temperature distribution can not be predicted precisely.

In this paper, a new method has been developed for calculating the flame tube wall together with the flow field inside and outside the flame tube. By using this method, the 3-D temperature distribution of the flame tube of the combustion chamber of an aeroengine has been simulated successfully.

**METHODOLOGY**

The numerical simulation method presented in this paper is the whole flow field coupling calculation for the wall temperature of the flame tube, including the simulation of the combustion, heat radiation and the flow fields of the primary air flow, the cooling air film and the mixing jet (secondary air flow) inside the flame tube as well as the secondary air flow outside the flame tube as shown in Fig.1, where the thick solid line represents the solid wall. The calculation zone is the whole combustion chamber. The unique governing equation has adopted in the calculation.

![Fig.1 The calculation zone of combustion chamber and computational grid](image-url)