Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 185 (2017) 105 – 109

Procedia Engineering

www.elsevier.com/locate/procedia

6th Russian-German Conference on Electric Propulsion and Their Application

Space factors influence on the size stability of small spacecraft structure elements

Yu.V. Skvortsov[a,*], S.V. Glushkov[a], Chernyakin S.A.[a]

[a] *Samara University, 34, Moskovskoye Shosse, Samara, 443086, Russia*

Abstract

This paper examines «Aist-2D» small spacecraft. Thermoelasticity problem is solved in two stages by finite element method using ANSYS® software. The first stage is thermal analysis. Space factors, devices and onboard equipment heat liberation are taken into account. External heat currents intensity and equipment work modes change depending on the spacecraft position on the Earth orbit. The second stage is stress-strain analysis of the structure, which is caused by the temperature field influence. As a result, the onboard equipment movement against structure elements and the acting forces are defined.

Peer-review under responsibility of the scientific committee of the 6th Russian-German Conference on Electric Propulsion and Their Application

Keywords: small spacecraft; space factors; thermal analysis; structural analysis; finite element method; onboard equipment; size stability.

1. Introduction

A small spacecraft is affected by the combined influence of many physical processes, which are causes by radiation fields of the Sun, the Earth and the spaces, as well as the residual Earth atmosphere.

The Sun radiation (which is usually treated as a point source of radiation with its light rays that are considered parallel) and the Earth radiation which leaves the Earth surface (the Sun radiation, reflected from the Earth atmosphere and surface, and the planet's self-radiation) are considered to be the most important for insuring the size stability of the small spacecraft structure.

In this work finite element method is applied through ANSYS® software. It provides unique opportunities for the interlinked multiphysics problems solving, which may combine, for example, strength and thermal physics not only

* Corresponding author. Tel.: +7-846-267-45-20.
E-mail address: proch@ssau.ru

Peer-review under responsibility of the scientific committee of the 6th Russian-German Conference on Electric Propulsion and Their Application
doi:10.1016/j.proeng.2017.03.326

within one program, but also within one model. The interaction between different physics fields in this case can be achieved either through direct relation or through load transfer. In the second case, the interaction is realized by using the results of one analysis as the loads in the other analysis.

In the thermoelectricity problem it is assumed that the thermal and structural analysis are performed successively. Interdependent thermoelectricity equations describe body deformation which occurs under mechanical and thermal influence, as well as the reverse effect – its thermal field change due to deformation. However, in most cases temperature deformations are insignificant and, therefore, they have no significant influence on temperature distribution. It will be one-way link in this case.

If one-way load transfer occurs, the following steps should be taken to solve thermoelasticity problem:
- thermal problem defining and solving;
- returning to preprocessor and modifying the database. The finite elements types should be changed (from thermal to structural ones), additional material properties should be set (such as Young's modulus of elasticity, Poisson`s ratio, linear expansion coefficients etc.) and structural boundary conditions need to be defined;
- reading the temperatures from the Jobname.RTH thermal analysis results file (LDREAD command);
- structural problem solving, that is thermal strain and stress calculation of the structure.

2. Brief description of the structure

«Aist-2D» small spacecraft structure is designed to accommodate target-oriented, scientific and supporting devices inside and outside the spacecraft. This design is intended to create and support both the supporting systems and the specified conditions for the normal functioning of all onboard systems at all stages of operation. The small spacecraft structure in the form of a rectangular parallelepiped is non-sealed and it consists of a frame, six external panels and one central panel which carries visual observation range electrooptical equipment.

The frame is designed as a box-shaped section load-bearing structure made of aluminum alloy, which consists of three horizontally oriented structural rings, interconnected by struts. Each panel is a sandwich plate with a lightweight honeycomb core made of metal foil and two thin shells made of aluminum alloy. The layers are joined together with film adhesive. In the places where the panels are attached to the frame and in the mounting seats for the onboard equipment bushings made of D16 material are used.

To ensure heat removal from the places where onboard equipment is installed all the panels except from the central one have embedded internal heat transfer tubes. The external heat transfer tubes of smaller diameter are installed perpendicular to them.

3. Thermal analysis

The goal of the thermal analysis is temperature distribution calculation in the structure under investigation and the other output values which are associated with it. This analysis is performed using the commercial program ANSYS® which is based on finite element method. Two types of heat transfer are considered - thermal conductivity and heat emission.

The following element types are used to build the thermal model. To idealize honeycomb panels a 4-node thermal element of the SHELL131 multilayered shell is used. For the three-layered shell with the linear temperature variation through the thickness of each layer (KEYOPT(3) = 1) the following options are taken as the nodal degrees of freedom: TBOT - temperature of the bottom surface; TE2 - temperature of the coupling face of the first and the second layer; NE3 - temperature of the coupling surface of the second and the third layer; TTOP - upper surface temperature.

A 2-node thermally-conductive bar element LINK33 is used for the modeling of the frame, heat transfer tubes and fasteners. Its each node has only one TEMP (temperature) degree of freedom. An 8-node thermal solid body