

UDC 681.3

Borysov V. V.<sup>1</sup>, *senior lecturer*, Ahmadi Amir<sup>1</sup>, *bachelor*<sup>1</sup>*Igor Sikorsky Kyiv polytechnic institute*

## THE METHOD OF PRELIMINARY DESIGN THE CENTRAL COMPARTMENT OF CARGO AIRCRAFT FUSELAGE

One of the most important tasks to be solved in the process of designing aircraft of the transport category is the task of analyzing the distribution of stresses in the structural elements of fuselage. There are various integral methods for calculating the stiffness parameters of longitudinal structural elements. However, all of them have one fundamental drawback associated with the impossibility of accurately determining the loads acting on each element of the structure, because that do not allow taking into account the mutual influence of deformations of elements. Now the most effective method for solving the problem of determining the effect of deformations on the stress strain state (SSS) of a structure is the finite element method (FEM), which is a universal method for calculating the SSS of arbitrary structures of complex technical objects. However, FEM does not make it possible to determine the stiffness parameters of structural elements in a first approximation, based on the integral values of loads and permissible stress values. The report contains a description of the method of information technology for specifying the stiffness parameters of a finite element model (FE-model) of a structure based on the results of integral calculations.

The problem of implementing information transfer of rigidity parameters of structural elements from an integral calculation model to a FE-model includes three aspects:

- difference in the structures of integral and FE- models;
- insufficiency of finite element (FE) attributes in CAE systems to determine their correspondence to elements of a real structure;
- lack of connection between CAE systems and database management systems.

The structure of the integral calculation model is built on the basis of the principle of regularity, according to which the rigidity parameters of the structure are specified in the form of an array of data on the distribution of thicknesses of structural materials along the contour of the section, as well as an array of distances of points on the contours of sections (taking into account local thicknesses of materials) from the conditional axes of rigidity sections. To simulate changes in material thicknesses along cross section, the section contour is divided into separate parts. Sections of reinforcing elements (stringers and caps of spars) are included in the cross-sectional area of the corresponding parts. Loads are specified in the form of resultant forces and moments in cross sections of the model. The stress values are determined by the ratio of the re-

sultant bending moments and the section parameters. At the same time, it is not possible to model nodes of application of concentrated loads and irregularity zones.

In a FE-model, the design of a technical object is described as a set of conditionally elastic elements connected to each other through model nodes. In this case, structural elements are modeled as one or more FEs. In modern CAE systems, data on the coordinates of nodes and topologies of FEs are stored in one-dimensional arrays, which do not make it possible to determine the belonging of FEs to one or another section of the structure.

The monograph [1] describes a method for constructing a FE-model of the wing box structure of a transport category aircraft using the means and methods of object-oriented information technology, implemented in the form of a database management system (DBMS) "SPACE". The use of this technology made it possible to include in the model data structure arrays that describe the correspondence of the FEs to the real elements of the modeled structure:

**int P\_COUNT; //-> Number of cross sections in FE-model**

**int P\_FE\_COUNT[50][2]; //-> Number of FE in contours of cross sections**

**int P\_FE\_NUMBERS[50][2][300]; //-> FE numbers in cross sections**

**int FE\_FUNCTIONAL\_CODES[50][2][300][20]; //-> Functional codes of FE**

The correspondence between FEs and actual structural elements is realized by specifying functional codes for each FE, which are sequences of integers that are similar to drawing numbers. Based on the functional code, you can set the unit type, compartment number, assembly type, part code within the assembly, and part number of the part (if the part is modeled in the form several FEs). Automatic assignment of functional codes of FEs became possible thanks to the use of highly specialized software procedures implemented within the framework of object-oriented information technology.

**Algorithm for reading stiffness parameters of FEs.** The algorithm for reading FE stiffness parameters includes the following steps:

*Step 1.* Reading the FE number in the current cross section:  
**P\_FE\_NUMBERS[i][j][k].**

*Step 2.* Reading FE Functional Code: **FE\_FUNCTIONAL\_CODES[i][j][k][...].**

*Step 3.* Determining whether a function code corresponds to the type of unit.

*Step 4.* Determination, based on the FE number, the numbers of the nodes on which FE is based.

*Step 5.* Reading the coordinates of the nodes on which the FE is based.

*Step 6.* Calculation of coordinates of the FE center gravity.

*Step 7.* Appeal of the FE model to the integral model with transfer of the FE functional code and the coordinates of its center of gravity.

*Step 8.* Determination, based on the functional code of the finite element and coordinates of its center of gravity, the section number of the integral model and the number of the section contour part.

*Step 9.* Reading the stiffness and code of material for a specific part of the contour.

*Step 10.* Transferring the stiffness and code of material to the FE model.

**Data exchange format between the FE model and the integral model.** В рамках інформаційної технології "SPACE" обмін даними між КЭ-моделью і інтегральною моделлю кессона крила здійснюється через віртуальну структуру даних наступного формату:

```
struct _FE_INTEGRAL
{
    //- Source data for integral model: -----
    int FunctionalCode; //-> Functional code
    double Coord[3]; //-> Coordinates of the gravity center of FE

    //- The result data for FE model:
    double Stiffness; //-> Stiffness of the skin
    int MaterialCode; //-> Conditional of material code for FE
};
```

**Scheme of data exchange within object-oriented IT.** In the process of the read the stiffening parameters of the FE, sequential data exchange is carried out (Fig. 1) between two objects of the design database: the FE model of the outboard wing box (OWB) and the integral model. At the same time, the FE-model of OWB and the integral model are both part of the database of the FE-model of the airframe, which serves the design departments that design the airframe of the aircraft. The data exchange traffic format is described in the form of a **\_FE\_INTEGRAL** structure.

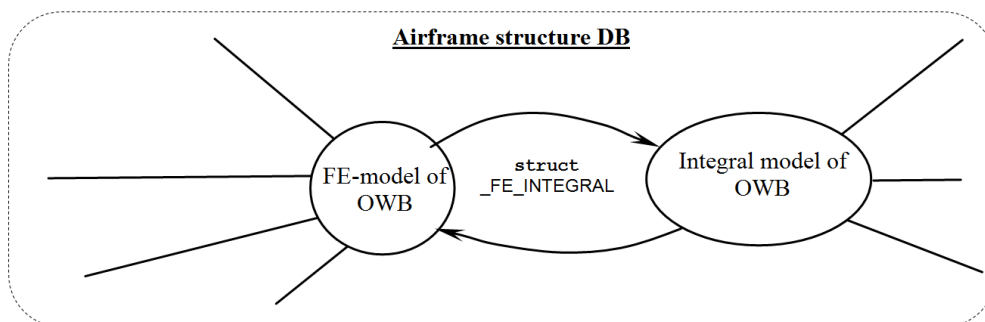


Fig. 1. Scheme of data exchange between the FE model and the integrate model

**Reference**

1. Borisov V. The methods of the synthesis of finite element model of the wing box, LAP Lambert Academic Publishing (ISBN 978-3-659-67887-5), Jan. 2015, -136 p.