

Development of Numerical Methods for Simulation of Airframe Assembly Process

Sergey Lupuleac, Yulia Shinder, Margarita Petukhova, Sergey Yakunin, and Alexander Smirnov
St. Petersburg Politechnic University

Dmitry Bondarenko
EADS RTO Sk LLC

ABSTRACT

The paper is devoted to further development of numerical methods for simulation of riveting process during aircraft assembly (see [3, 4]). Algorithm modifications that increase computation accuracy, speed and complexity are given. These modifications involve the dual problem solving, incorporation of multiple calculation nets and special two stage procedure for calculation of deformations and stresses arising during assembly.

CITATION: Lupuleac, S., Shinder, Y., Petukhova, M., Yakunin, S. et al., "Development of Numerical Methods for Simulation of Airframe Assembly Process," *SAE Int. J. Aerosp.* 6(1):2013, doi:10.4271/2013-01-2093.

INTRODUCTION

During the assembly process it is important to control both gap between joined parts and stresses caused by installed fixture elements. On the one hand tight contact between parts should be achieved; on the other hand engineers should avoid cracks, composite layer delamination and part damaging. The main goal of the presented work is to develop special tool which allows engineers to perform simulations in order to evaluate displacements and stresses of aircraft parts on the assembly line. For this purpose specialized software complex was developed for solving the particular contact problem arising during the simulation of assembly process.

The software complex and its numerical algorithm were already presented in papers [3, 4, 5]. This paper is devoted to introduction of latest enhancements to the numerical algorithm that were implemented in order to ensure the possibility to work with complicated models of real airframe junctions.

THE BASICS OF NUMERICAL ALGORITHM

The concept of basic numerical algorithm as well as validation tests were described in [3]. Here we would like to give only the main idea of the method.

The main challenge for simulation the assembly process is necessity to solve the contact problem for determination of deformed stress state of the assembly loaded by the forces from fastening elements. This contact problem has following peculiarities to be taken into account in order to derive efficient algorithm:

1. The contact may occur only in junction area that is known a priori. Thus there is no need to implement complicated procedures for detection the zone of possible contact.
2. The installed fasteners and rivets restrict relative tangential displacements of assembled parts in the junction area. So the relative tangential displacements in junction area are negligible in comparison with normal ones. This special feature of the problem justifies implementation of node-to-node contact model that is much simpler than general surface-to-surface model.
3. Loads from fastening elements are applied inside junction area.
4. We are interested only in the stationary solution of the problem.
5. Friction forces between assembled parts in the contact zone do not play significant role due to smallness of relative

tangential displacements. So the friction can be omitted from consideration.

6. Stress state of each part in the assembly obeys linear theory of elasticity.

Let us consider artificial finite element model of upper wing-to-fuselage junction that is given in Fig. 1 and Fig. 2 (fragment). There are two parts in the assembly: the first is the wing (blue) and second is the fragment of center wing box (yellow).

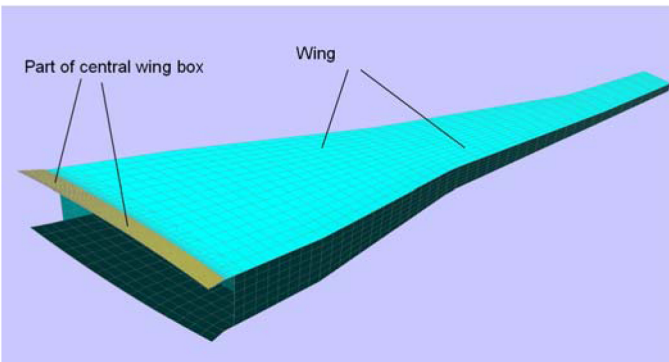


Figure 1. Finite element model of artificial wing-to-fuselage junction.

The nodes of possible contact (the nodes in junction area) are shown as green points in Fig. 2. We denote these nodes as computational. The set of all computational nodes is referred as calculation net.

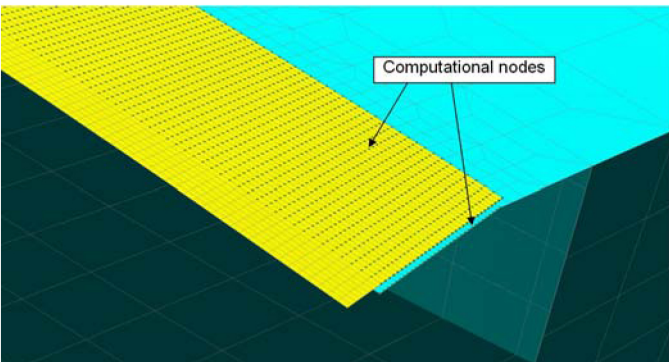


Figure 2. The nodes of junction area.

Using standard finite element modeling technique we formulate the contact problem in discrete variation form [6]:

$$\min_{U \in S_h} \left(\frac{1}{2} U^T \cdot K \cdot U - F^T \cdot U \right). \quad (1)$$

Here U is displacement vector of finite element nodes, K is stiffness matrix of finite element system, F is vector of

applied loads, S_h is admissible set that is determined with regard to boundary and non-penetration conditions.

In typical finite element model of airframe junction (e.g. wing-to-fuselage junction, as it is shown in the figures above) the number of nodes in junction area is much smaller than the total number of nodes. So the elimination of displacements outside the junction area dramatically reduces dimension of problem. To do that we divide all the finite element nodes in two groups containing nodes in junction area (green points in Figure 2) and all the others. Then

$$U = \begin{pmatrix} U_C \\ U_R \end{pmatrix},$$

where U_C is vector of displacement of nodes in junction area and U_R is vector of displacements of other finite element nodes. The same procedure can be implemented for matrix

$$K = \begin{pmatrix} K_{CC} & K_{CR} \\ K_{CR}^T & K_{RR} \end{pmatrix} \text{ and vector } F = \begin{pmatrix} F_C \\ F_R \end{pmatrix}.$$

Calculating Schur complement of K_{RR} , we get rigidity¹ matrix K_C from the formula $K_C = K_{CC} - K_{CR} \cdot K_{RR}^{-1} \cdot K_{CR}^T$. Now it is possible to derive the reduced problem:

$$\min_{N \cdot U_C \leq G} \left(\frac{1}{2} U_C^T \cdot K_C \cdot U_C - F_C^T \cdot U_C \right), \quad (2)$$

where N is linear operator which defines normal to contact surface, G is initial gap vector in junction area.

Also we mention that U_C contains only the normal components of node displacements. This simplification is possible due to the smallness of tangential displacements regarding normal ones. Thus we reduce initial contact problem to the problem of quadratic programming. Moreover the dimension of reduced problem is much smaller (e.g. the finite element model depicted in Figures 1 and 2 has around 130 000 degrees of freedom and reduced problem (2) for this model has only 16 000 unknowns). This approach is known in finite element modeling as substructuring.

Methodology given in this paper allows us to reduce problem dimension so that it is necessary to perform computations only inside of junction area. Similar approaches for contact problem solving can be found in [6, 7, 8, 9]. Several works of Pedersen [7, 8, 9] are devoted to application of reduction method to contact problems. However, it is assumed that contact is achieved everywhere in junction area. Then displacements of each considered body can be computed independently. Our approach removes this restriction: contact may occur anywhere in junction area.

THE OVERVIEW OF SOFTWARE

The described above numerical procedure was realized in software complex aimed at simulation of airframe assembly

¹Actually K_C is reduced stiffness matrix of finite element system but traditionally we call it rigidity matrix