

Methodology for Solving Contact Problem during Riveting Process

2011-01-2582

Published
10/18/2011

Sergey Lupuleac
St. Petersburg Politechnic Univ.

Margarita Petukhova and Yulia Shinder
St Petersburg Politechnic Univ

Bertrand Bretagnol
Airbus

Copyright © 2011 SAE International

doi:[10.4271/2011-01-2582](https://doi.org/10.4271/2011-01-2582)

ABSTRACT

The paper describes the methodology of contact problem solving that is used in specialized software code aimed at simulation of aircraft assembly process. For considered class of problems it is possible to radically reduce the number of unknowns without loss of accuracy.

The results of validation of developed code against physical experiments and commercial FEM codes are also given.

INTRODUCTION

During the assembly process it is important to control both the gap between joined parts and stresses caused by installed fasteners. On the one hand tight contact between parts should be achieved; on the other hand engineers should avoid cracks, composite layer separation and part destruction.

The 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 ASRP software complex was developed for solving the particular contact problem arising during the simulation of riveting process. The structure and features of this software are described in [1] and [2].

The main peculiarity of considered approach is complete separation of data preparation process from simulations. The next important feature of ASRP is reduction of problem dimension using a priori information about possible contact zone and computing all values during simulation process only inside this zone.

The present paper gives the detailed description of developed mathematical algorithm as well as some numerical tests for its validation.

MAIN SECTION

1. METHODOLOGY FOR GAP COMPUTATIONS

Let us consider finite element model representing several panels to be assembled (e.g. blue, green and red ones as it is shown in [Fig. 1](#)). These panels overlap in so called junction area. The contact between panels is possible only inside the junction area.

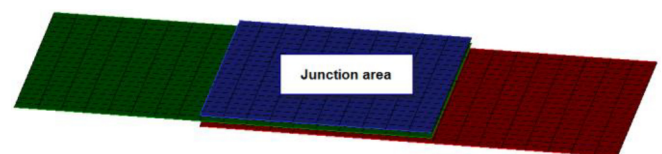


Figure 1. Considered model

Initially panels are separated by distance called initial gap (see Fig. 2).

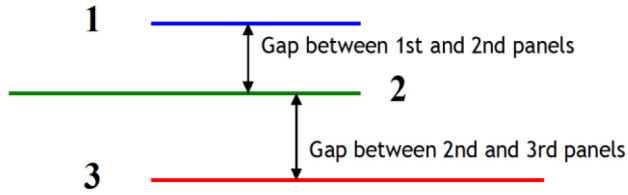


Figure 2. Initial gap

Let M be a number of finite element nodes in the model and $\{x_j\}_{j=1}^M$ is a set of finite element nodes. The unknowns are displacements in all the nodes $\{U_j\}_{j=1}^M$. We can consider displacements both in tangential and normal directions or just normal ones. Here we assume that the set $\{x_j\}_{j=1}^M$ includes the nodes of upper, middle and lower panels.

If we don't take into account possibility of contact between panels, the determination of $\{U_j\}_{j=1}^M$ needs only solving the system of linear equations:

$$K \cdot U = F, \quad (1)$$

where $K = \{k_{ij}\}_{i,j=1}^M$ is global stiffness matrix, $F = \{f_j\}_{j=1}^M$ is vector of loads.

Also it is necessary to impose the boundary conditions to avoid rigid body motion:

$$U_j = U_{0j}, \quad x_j \in \Gamma_{Uh} \quad (2)$$

where Γ_{Uh} - part of the boundary where displacements are specified. We take note that Γ_{Uh} contains the nodes belonging to all the panels.

The following restriction needs to be added to the system (1), (2) to prevent interpenetration of the panels:

$$(A \cdot U)_j \leq G(x_j), \quad x_j \in \Gamma_{Ch}, \quad (3)$$

where A is linear operator determining the relative displacement of parts in normal direction, $G(x_j)$ is a function of normal initial gap in the node x_j ; Γ_{Ch} is junction area that is the set of nodes that can come into contact. We take note that node-to-node contact is considered.

To finalize problem statement we pose conditions on contact stresses: in the contact points the normal stresses are contradiirectional and have the same value. The friction between parts is not taken into account, so tangential stresses in contact points are zero.

The equivalent variational formulation is used to determine the displacements. The displacement vector $\{U_j\}_{j=1}^M$ under the load $F = \{f_j\}_{j=1}^M$ provides minimum to the energy functional

$$E(U) = \frac{1}{2} U^T \cdot K \cdot U - F^T \cdot U \quad (4)$$

on the set of all admissible displacements A_C :

$$A_C = \left\{ U \mid U = U_0 \text{ in } \Gamma_{Uh}, A \cdot U \leq G \text{ in } \Gamma_{Ch} \right\} \quad (5)$$

Let us restrict ourselves by the special case when the number of nodes in junction area is much smaller in comparison with whole model. It usually takes place in problems of simulation of aircraft part riveting. In this case we can principally reduce the problem dimension by eliminating all unknowns outside the junction area. It will lead to significant decrease of computational time without loss of accuracy.

Further in the paragraph we are to formulate the reduced minimization problem (with unknowns in junction area) equivalent to (4)-(5).

For that we choose the system of computational nodes $CN = (cn_1, cn_2, \dots, cn_I)$ in junction area, for example, as in Fig. 3 (I is the total number of nodes in junction area). We denote it calculation net. It consists of nodes where displacements will be computed. We will build the reduced rigidity matrix K_C , which allows to pass from whole model to calculation net.