

Assembly Simulation of Riveting Process

Sergey Lupuleac, Margarita Kovtun and Olga Rodionova
Saint-Petersburg Polytechnic University

Benoit Marguet
Airbus

Copyright © 2009 SAE International

ABSTRACT

The presented paper introduces the new software complex aimed at simulation of the riveting process as applied to aircraft parts. The software complex implements the novel mathematical model based on minimization of the potential energy. The paper gives the detailed description of the mathematical model and particularizes the main features of the software. The physical and numerical tests aimed at validation of the software are also described in the paper.

INTRODUCTION

The temporary fastener installation is one of the most time consuming operations during the whole aircraft assembly process.

The overall objective of presented work is to improve the current assembly process by development of a fast simulation tool in order to analyze and visualize the riveting process in an aircraft junction for optimizing the number of temporary fasteners.

The methodology that is implemented in the software enables user to perform riveting process simulations in very fast and user friendly manner. The main difficulty faced in simulation is the necessity to take into account the contact between fastened parts. The contact problem is solved by minimizing the system potential energy with regard to restriction of displacements aiming to prevent the penetration of the riveted parts. The rigidity matrix needed for determination of the potential energy is

computed in the external finite element solver NASTRAN.

The following approach is implemented in the software tool: data preparation process is separated from simulations of riveting process. Thus the software complex is divided in two parts: Preprocessor and Simulator.

Preprocessor is designed for preparation of the model for Simulator on the base of imported finite element mesh. Preprocessor generates all data structures required for the calculations in Simulator.

Simulator is the part of software complex designed for the riveting process simulation. Simulator is designed for a person not familiar with finite element modeling. This tool permits the computation of the gap between parts in 0.1 sec. Also the Simulator has the functionality of automatic fastening element arrangement in order to minimize the number of temporary fasteners.

The special program of software validation was realized both in Airbus and St. Petersburg Polytechnic University. The results of simulations were tested by physical experiments and by commercial FEM code ANSYS. It was shown that presented complex provides the same accuracy as the commercial codes and good correlation with experimental results.

Very small computation time, user friendly interface and separation of data preparation from simulation process make the presented tool very convenient for solving such type of problems.

MAIN SECTION

1 DESCRIPTION OF ASRP MATHEMATICAL MODEL

The computations of gap between the fastened parts in ASRP (Assembly Simulation of Riveting Process) Simulator are very fast. At the same time the obtained results are very close to those obtained with commercial codes. This result is achieved due to implementation of the special mathematical procedure. The present section is devoted to the description of this procedure as well as other approaches used in ASRP complex.

1.1 Rigidity Matrix

In this paragraph we consider the concept of a rigidity matrix that is implemented in ASRP project.

The structural behavior of elastic body is determined by such factors as its geometry, mechanical properties of its materials and the location of the fixation areas (that are the places where displacements are knowingly restricted). All these properties are taken into account in the rigidity matrix.

Let's consider a system of two overlapping elastic bodies fixed in some areas (Fig. 1). Our goal is to simulate the process of riveting junction of these bodies.

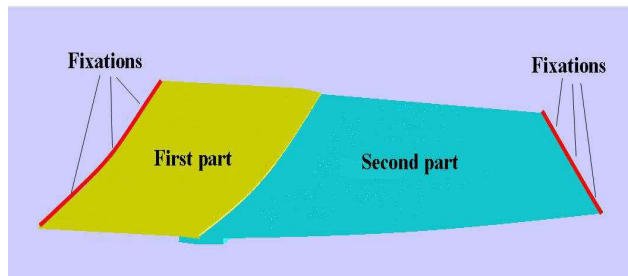


Fig. 1 The system of bodies to be fastened

The zone where these bodies overlap is called junction area (Fig 2).

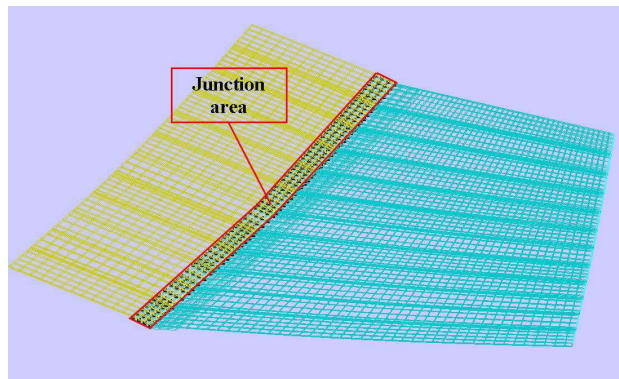


Fig. 2 Junction area

Let's choose some set of nodes (points) $X = (x_1, x_2, \dots, x_n)$ in the junction area, e.g. as it is shown in Fig. 3. We will refer these nodes as computational ones. And we will call the system of computational nodes by the calculation net.

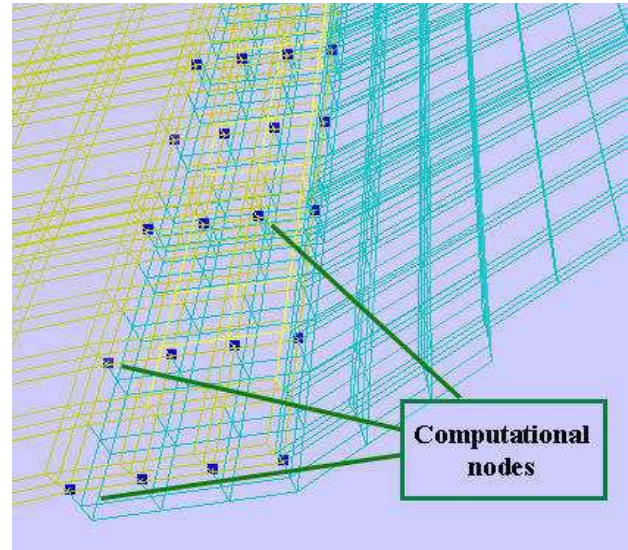


Fig. 3 Calculation net

Let's consider first and second parts separately.

We apply forces $P^T = (p_1, p_2, \dots, p_n)$ in the computational nodes to the first part. Here p_i is a force applied to node x_i in the normal direction, superscript T denotes transposition. These forces are related to displacements caused by them $(U^{(1)})^T = (u_1^{(1)}, u_2^{(1)}, \dots, u_n^{(1)})$ (where $u_i^{(1)}$ is displacement of the node x_i in normal direction under the action of force P) with the following equation written in a vector form

$$U^{(1)} = R^{(1)} \cdot P \quad (1)$$

Here $R^{(1)} = \begin{pmatrix} r_{11}^{(1)} & r_{12}^{(1)} & \dots & r_{1n}^{(1)} \\ r_{21}^{(1)} & r_{22}^{(1)} & \dots & r_{2n}^{(1)} \\ \dots & \dots & \dots & \dots \\ r_{n1}^{(1)} & r_{n2}^{(1)} & \dots & r_{nn}^{(1)} \end{pmatrix}$ is the flexibility matrix of the first part.

Next we apply the forces P^T to the second part. The relation between forces and displacements $(U^{(2)})^T = (u_1^{(2)}, u_2^{(2)}, \dots, u_n^{(2)})$ (where $u_i^{(2)}$ is normal