Mathematical simulation of riveting process

Author: Margarita Kovtun
E-mail: margarita_kovtun@inbox.ru

Introduction

The theme of this report arose in the context of the project which results from the cooperation between Airbus and Laboratory of Applied Mathematics and Mechanics of Saint-Petersburg State Polytechnical University.

This project is aimed at simulating riveted junctions of aircraft parts. For this purpose, the special software product named ASRP was created in the mentioned laboratory. At the moment, it gives a possibility of modeling the particular case of junction: wing to fuselage. It is presumed that with the help of this tool user will be able to define the optimal configuration for riveting providing required sealant pressure and gap thickness between the riveted parts.

During the code development several models of junction have been already implemented in ASRP. But these models were quite simplified and couldn't be considered as models close to the real cases. As a result, I went to Toulouse for three-month internship to get basic knowledge about real assembly process.

The main objective of my work performed in Toulouse was to simulate the junction between A320 wing and fuselage.

It included:

- The preparation of all the initial data necessary for ASRP, i.e. problem statement, generation of finite element mesh of the joined parts, computation of the rigidity matrix of the wing, etc.
- The start of investigations on created model

1 Presentation of the theme

1.1 Description of real riveting process

The subject of my report is simulation of riveting process between wing panel and panel of fuselage. First of all, I would like to describe how it is done in reality.

On assembly line the process of joining wing to fuselage is realised in eight steps.

1. Determination of the best position of the wing referring to fuselage.
2. Holes drilling.
3. Decoupling of wing and fuselage, cleaning.
4. Applying the sealant on the surfaces of both details.
5. Joining wing to fuselage, installation of temporary fasteners in all the holes.

The last three steps include: removing of temporary fasteners, reaming and installation of final rivets. But, in order to provide sufficient pressure in the sealant, temporary fasteners are removed third by third, so that 1/3 of fasteners is removed, 1/3 of rivets is installed in empty positions, then second third is removed and operations are repeated.

Now, for A320, the process mentioned above is the longest one on the assembly line. In fact, the process of riveting is doubled: firstly temporary fasteners are installed, secondly - the final rivets. If it is possible to reduce the number of temporary fasteners but still providing the contact between parts, it will evidently accelerate the assembly process.

1.2 Description of software ASRP

ASRP tool is designed to simulate the process of riveting of composite parts with complex geometry. Also it is possible to take into consideration the existence of sealant between the riveted parts.

The objective of ASRP is calculation and visualization of the sealant pressure field and gap thickness between riveted parts with different configurations of fixations applied.

At present, the possibilities of this software are constrained to the case of junction wing-to-fuselage. The area where the wing covers the panel of fuselage is referred as a junction area. Fastening elements can be installed only in junction area and the computations are also performed there.

User is allowed to determine the locations of fastening elements and to define their parameters, i.e. diameter of fastener, its length, provided force.

The purpose of using ASRP is to reduce the number of temporary fasteners used for riveting, that means to find such configuration that the number of fasteners is minimal and the parts are still in contact (the gap between them is equal to 0).

1.3 Steps taken to make a simulation

One of the main advantages of ASRP is that the computations of gap and pressure are made in real-time mode; often it takes less than a second. However, the preparation of initial data to launch the simulation in ASRP is a process that requires much time and application of different skills.

Input data for ASRP are:

- Joint component geometry imported into software from .stl format file
- Rigidity matrices of joined components. Rigidity matrix displays geometry, component material properties and fastening area locations
- Initial gap vector imported into software from special format text file.
- Fastening map imported into software from special format text file. This file will be used for storage of point coordinates for future drilling to install fastening elements.
- Sealant mechanical properties imported into software from special format text file.

My internship in Toulouse was purposed to approbate ASRP on the real wing-to-fuselage junction of A320 plane.

2 Preparation of initial data for ASRP

2.1. Description of the model

The object of simulation is the junction between wing and fuselage of A320 plane.

As this aircraft is produced from 1980s, usually the data that describe its configuration are paper projects. Especially for studies presented in this document, the CAD model of junction was made from the measurements done on the real wing. The positions of several thousands points from the wing surface were determined with the laser device.

The junction is shown in Fig.1.

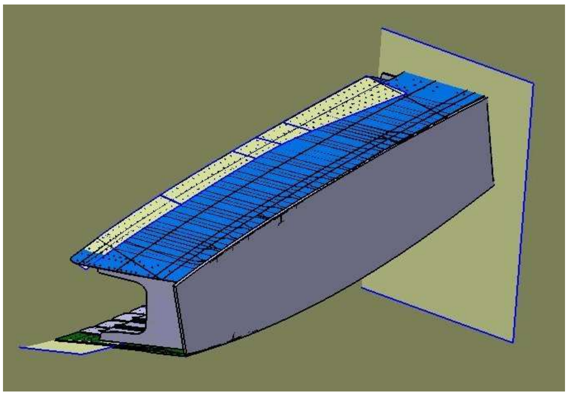

Fig.1 CAD model of the wing. Reference to fuselage

The geometry of the wing is very complicated as it can be seen from Fig.1, so I decided to simplify the model and to take into consideration only the junction between upper panel of the wing and panel of fuselage.

Both parts are considered to be isotropic and to be made of aluminum.

2.2 Finite element mesh

The first step to execute is to generate finite element mesh of joined parts.

There are two purposes for which mesh is made:

1. Representation of geometry in ASRP. The geometry is imported in ASRP from .stl format files, which are made from the mesh.
2. Calculation of rigidity matrix. Mesh should be exported to the any FEM (finite element modeling) system: Nastran, Ansys, Abaqus.

I have made the mesh using ANSYS ICEM CFD 10.0. View of mesh is shown in Fig. 2.

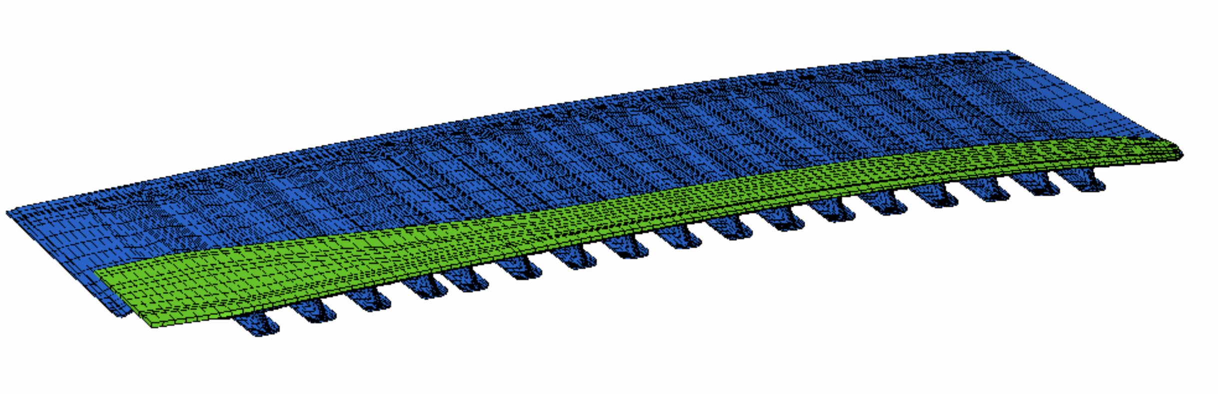

Fig. 2 Isometric view of the mesh

Three dimensional volume elements are used in this model. Mesh of the wing consists of 27880 elements; mesh of the panel of fuselage consists of 1221 elements.

For calculation of rigidity matrix it is necessary to choose the system of points in the junction area so that they belong to both parts. These points should coincide to the nodes of the mesh; they are so called computational nodes.

2.3 Calculation of rigidity matrix

The rigidity matrix is calculated in FEM system, I used ANSYS 10.0 for this purpose.

The calculation procedure is described below.

First of all, we calculate flexibility matrix ***R*** of the wing.

Let's denote system of computational nodes as $X^T = (x_1, \quad \dots \quad x_n)$, *n* – the number of computational nodes (*n* = 204 in my case). Let us pick out and fix a node x_j from the computational node system $X^T = (x_1, x_2, \cdots, x_n)$.

We impose the unite force into this node and zero forces into other nodes:

$$p_i = \begin{cases} 1, & i = j \\ 0, & i \neq j \end{cases}$$

The example is shown in Fig. 3(left image).

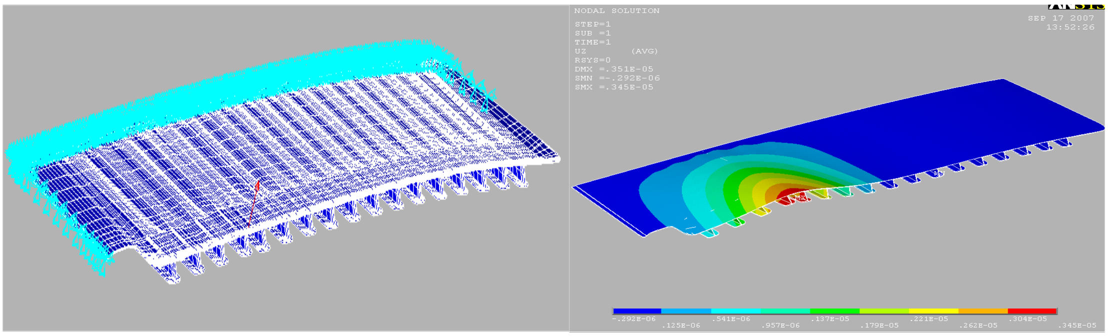

Fig. 3 The example of applying load on node.

By means of finite element model we compute displacements $U^T = (u_1, u_2, \cdots, u_n)$ in each computational node $X^T = (x_1, x_2, \cdots, x_n)$ and thus we obtain the *j*-th line of the flexibility matrix ***R***.

The example of solution: field of displacements *U* is shown in Fig. 3(right image).

This procedure must be performed for all nodes $X^T = (x_1, x_2, \cdots, x_n)$ for computation of the whole flexibility matrix ***R***. Therefore the defining of flexibility matrix needs the series of *n* computations, where *n* is the number of computational nodes.

After defining of flexibility matrix ***R*** the rigidity matrix ***K*** can be obtained from equation:

$$K = R^{-1}$$

2.4 Determination of boundary conditions

There is a critical point in calculation of rigidity matrix. It is the determination of proper boundary conditions when stating the problem in FEM solver.

As the initial model was reduced only to the upper wing surface, it is necessary to define the constraints on the wing flexibility according to the reality.

I have made series of numerical experiments applying several variants of boundary conditions. The goal is to compare the fields of displacements obtained as the solutions of problems almost similar to each other. The force is always imposed on the same node, but boundary conditions are changed.

There are four problems with different boundary conditions:

1. Only the border line of upper surface of the wing is fixed. (Fig. 4)

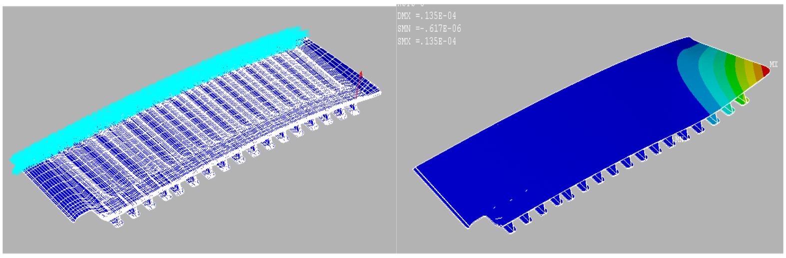

Fig. 4 First problem statement and solution

2. Three edges of the wing are fixed. (Fig. 5)

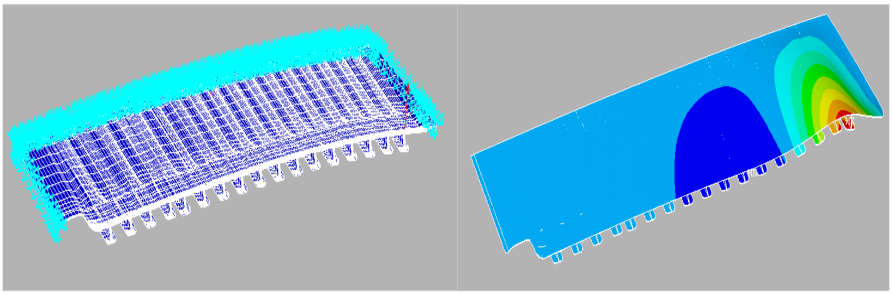

Fig. 5 Second problem statement and solution

3. The side edges are half-fixed. (Fig. 6)

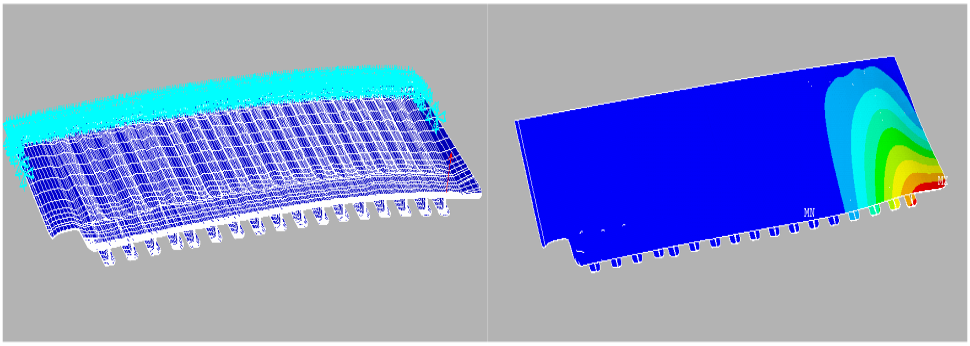

Fig. 6 Third problem statement and solution

4. Two walls with constant thickness are added to the model, it is the approximation of real model. (Fig. 7)

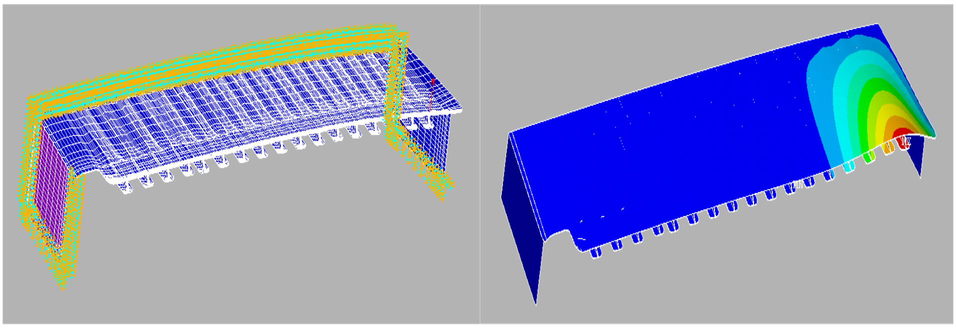

Fig. 7 Fourth problem statement and solution

Each time the solution is different, and it makes the stated problem more complicated. It is necessary to perform additional investigations concerning this subject.

3 Computations

3.1 Computations performed in ASRP2

The results that can be obtained in ASRP characterize the state of junction after installation of temporary fasteners, the fifth stage described in 1.1. Fastening element forces acting on assembling parts cause their displacements which eliminate initial gap between wing and fuselage. The problem is to find the current relative position of fastened bodies loaded by vertical forces of fastening elements.

Displacement field is computed for computational nodes $X^T = \left(x_1, x_2, \cdots, x_n\right)$. Displacements in other points of the upper wing surface are defined by interpolation.

The initial data for computation are the forces applied in computational nodes and the values of initial gap between parts also in computational nodes.

$$F = \begin{pmatrix} f_1 \\ \vdots \\ f_n \end{pmatrix} - values\ of\ \ forces\ applied\ in\ computational\ nodes$$

Initial gap is given by vector:

$$\Delta = \begin{pmatrix} \delta_1 \\ \vdots \\ \delta_n \end{pmatrix} - values\ of\ \ gap\ between\ wing\ and\ \ fuselage\ in\ computational\ nodes$$

The computation of wing current location is carried out by means of solution of following quadratic programming problem:

Find the displacement vector $U^T = (u_1, u_2, \cdots, u_n)$ that provides minimum to the positively defined function of n variables W given by the following formula: $W(u_1, u_2, u_3) = U^T \cdot K \cdot U - 2 \cdot F^T \cdot U$

where K is the rigidity matrix of the wing, U is displacement vector of wing points; F is the vector of forces on the upper wing surface caused by fastening elements. The minimization of this function is carried out with the following restrictions having form of linear inequalities: $U \le \Delta$