Numerical and experimental analysis of deformation and destruction of structurally heterogeneous joint assembly

Yu F Filippova and S V Doronin

Institute of Computational Technologies SB RAS, PO box 25515, Krasnoyarsk, 660049, Russia

E-mail: filippovajf@ict.nsc.ru

Abstract. This work is devoted to analysis of deformation and destruction of structurally heterogeneous joint assembly. The mathematical formulation of interacting parts of joint assembly is investigated. Logical and computational model of joint assembly presents various modes of its destruction and describes impairment conditions because of different damage states.

1. Introduction

The joint assemblies of parts and elements of machines and structures have features that largely form the nature of their systemic behavior. This is due to the following factors: differences in shape, size and materials of parts; variety of ways of parts interaction, including permanent (welds, adhesion, rivets) and detachable connections, friction; existence of intrinsic stress-strain state (SSS) being formed by assembly operation. Thereby joint assemblies usually should be considered as structurally heterogeneous and structurally complex objects. They are characterized by high level of uncertainty of characteristics and possible reactions in a wide range of external influences. This leads to general possibility for various scenarios of parts force interaction and conditions for the occurrence and development of damages to be realized. These scenarios cannot be described within the frameworks of simple models with small amount of parameters. It is necessary to justify the scenarios in connection with joint assembly structure, its parts properties and interactions. In the present work, we justified logical and computational model for deformation and destruction of a structural embodiment of joint assembly for beam skeleton parts of large scale precision reflector in parabolic antenna of terrestrial satellite communication systems.

2. Problem definition

Joint assembly for beam skeleton made of square cross-section pipes is the object of study. Force interaction of composite pipes and fittings is carried out by means of detachable (steel rings, bolts and nuts) and permanent (adhesive) connections (Figure 1). The nuts, the bolts and the rings are made of carbon steel with elastic modulus $E = 210000$ MPa, Poisson ratio $\nu = 0.3$. The fitting is made of woven polymer composite with elastic moduli $E_x = E_y = 60500$ MPa, $E_z = 6900$ MPa, shear moduli $G_{xy} = 19500$ MPa, $G_{xz} = G_{yz} = 2700$ MPa and Poisson ratios $\nu_{xy} = 0.03$, $\nu_{xz} = \nu_{yz} = 0.3$. The pipes are made of unidirectional polymer composite with elastic moduli $E_x = 60900$ MPa, $E_y = 27000$ MPa, $E_z = 15000$ MPa, shear moduli $G_{xy} = G_{xz} = 4700$ MPa, $G_{yz} = 3100$ MPa and Poisson ratios $\nu_{xy} = 0.07$, $\nu_{xz} = 0.4$, $\nu_{yz} = 0.27$. Coefficient of static friction of the interacting parts is 0.15.

Problem definition includes...
– analysis of possible damages of joint parts;
– formulation of mathematical model for joint assembly deformation;
– development and testing of a logical and computational model for deformation and destruction to
describe a set of possible scenarios of joint assembly behavior.

Development of computational model is carried out by integration of separate computational
modules based on theory-driven and field-tested algorithms of deformable solid mechanics, and
logical operators to take into account structure of joint assembly, interaction and damage of its parts.
The model includes above mentioned completely formalized algorithms as well as human-computer
procedure to change properties and conditions of interaction of joint parts.

Figure 1. Scheme of joint assembly for pipe and fitting: 1 is a pipe; 2 is a fitting; 3 is a ring; 4 is a
bolt; 5 is a nut; С1, С2 are threaded connections; C3 is glue joint; C4-C15 are frictional interactions;
∆ presents clearance.

3. Assembly joint testing
Typical damages and failures of thread connection parts and their causes are known from theoretical
engineering. Experimental tensile testing of joint assembly is carried out to clarify possible damages
of its composite parts. Testing machine Tinius Olsen 100ST was used. Two types of pieces were
subjected to testing: 1 – bolted composite parts (Figure 2, a); 2 – glued composite parts, two rivets
are not loaded and are for ease assembling (Figure 2, b).

Figure 2. Test pieces of bolted (a) and adhesive (b) joints.

The experiment allowed to reveal following possible damages:
– composite parts are crumpled by prestressed bolts (Figure 3, a);
– parts offset relative to each other due to adhesive layer damage (Figure 3, b);
– parts are crumpled and damaged by bolts in direction of tensile force (Figure 3, c).
From now onwards we shall take into account possibility of these damages occurrence when modeling joint assembly behavior.

4. Mathematical formulation of the problem
Any analysis of strength and failure of technical object is based on the finding static elastic SSS. Thereby the algorithm of static elastic analysis is the most important universal component of considered further computational models. In the most common formulation the algorithm of numerical (finite-element) solution of the problem (basic computational model for static elastic analysis – BCMSEA) is illustrated with Figure 4 [1].

![Algorithm diagram of base computational model for static elastic analysis](image)

**Figure 4.** Algorithm diagram of base computational model for static elastic analysis: \([k]\) is finite element stiffness matrix; \([K]\) is global stiffness matrix of the finite element model; \([U]\) is nodal displacement vector; \([F]\) is nodal forces vector; \([\varepsilon]\) is strain vector; \([\sigma]\) is stress vector; \([D]\) is differentiation matrix; \([A]\) is material stiffness matrix.

Stiffness matrices of isotropic (metals) and orthotropic (composite) materials are described respectively by equations (1) and (2):

\[
[k] = \int_{V} B^T \sigma B \, dV
\]

\[
[K] = \int_{V} B^T [A] B \, dV
\]
where

\[ \Delta = \frac{1 - \nu_x \nu_y - \nu_x \nu_z - \nu_z \nu_x - 2 \cdot \nu_x \nu_y \nu_z}{E_x E_y E_z}. \]

The solution of the three-dimensional contact problem of the theory of elasticity with a variable contact zone with friction [2] is carried out in two stages [3]:

1) determination of the mutual penetration of contacting bodies;
2) the definition of contact forces that prevent these mutual penetrations, based on the solution of the equations of equilibrium (motion).

When determining the geometry of the mutual penetration of the contacting bodies, the possibility of the penetration of active nodes through the surface of a possible contact of the passive body is checked. If virtual penetration occurs (i.e., possibly based on the stiffness of the active body), it is assumed that the active body is in contact with the passive body. To test the possibility of penetration of each active node through the surface of the passive body, an iterative procedure of the Newton-Raphson method is used.

Determination of contact forces that prevent mutual penetrations is carried out using the extended Lagrange method, based on the method of penalty functions with advanced control of indention. In this case, the role of Lagrange multipliers is performed by unknown (desired) concentrated contact forces. The numerical solution is obtained by the iterative method of conjugate gradients.

Then the computational model of contact interaction (CMCI) has the following form (Figure 5).
To analyze the possibility of parts offset relative to each other if the adhesive layer is damaged or the bolt joint is loosening, we use the friction model in the form of the Amonton-Coulomb law

\[ \tau_{\text{lim}} = \mu_s \cdot P, \]

where \( \tau_{\text{lim}} \) is limit frictional stress; \( \mu_s \) is coefficient of static friction; \( P \) is contact normal pressure.

The behavior of the contacting bodies is determined in accordance with the expression

\[ \tau < \tau_{\text{lim}}, \quad \tau = \sqrt{\tau_1^2 + \tau_2^2}, \]

where \( \tau_1, \tau_2 \) are frictional stresses in two mutually perpendicular directions.

When the inequality (4) is fulfilled, the contacting bodies are kept from displacement relative to each other by friction force. Otherwise, the mutual displacement of bodies begins. The condition for its continuation is the inequality

\[ \tau \geq \mu_d \cdot P, \]

where \( \mu_d \) is coefficient of sliding friction (\( \mu_s \geq \mu_d \)).

5. Development and implementation of logical and computational model

The numerical model of the joint assembly is developed taking into account the presence of one plane of symmetry. Boundary conditions include fixed support in the fitting holes, symmetry conditions and axial tensile force (Figure 6).

On the basis of a previously developed information model describing possible scenarios and conditions for the damage accumulation and fracture [4], the computational model for the deformation and destruction of the joint assembly was developed and practically implemented (Figure 7). It allows one to obtain quantitative estimates of the possibilities of transition from one state of damage to another, conditions for the occurrence of a particular branch of the scenario. The computational model allows analyzing the behavior of the structure both in the intact state and in various damaged states.
Figure 6. Boundary conditions.

Figure 7. Logical and computational model for deformation and destruction of joint assembly: \( \sigma', \sigma', \sigma' \) present bearing strength of the fitting, the ring and the pipe respectively; \( \sigma \) is maximum equivalent contact stress; \( \sigma_u \) is ultimate strength of material; \( \sigma_{\text{glue}} \) is maximum equivalent strength in the adhesive layer; \( \sigma_{\text{glue}_{cr}} \) is adhesive layer strength; \( \tau \) is frictional stress; \( \tau_b \) is maximum shearing stress in bolt cross-section; \( \tau_{\text{cut}} \) is bolt shear strength.
As applied to the considered object, the computational model of the intact structure includes the basic solving equations and the algorithm of the finite element method in displacements, as well as nonlinear procedures for solving the three-dimensional contact problem of elasticity theory with a variable contact zone with friction. The formulation of a numerical (finite-element) model of a joint assembly includes two load steps. At the first step, the preliminary tension of the bolts is set and the initial (intrinsic) stress state of the mechanical system is determined. On the second, the actual analysis of deformation and fracture under external loading is carried out.

5. Conclusion
Numerical and experimental analysis of deformation and destruction is the implementation of the technology of applied system analysis of structural strength and survivability, aimed at providing the ability to predict and prevent all possible damage states. Testing the technology by the example of the joint assembly of the beam skeleton of a large scale precision reflector in a parabolic antenna made it possible to establish the critical values of internal and external influences (pretension forces of bolts and tensile forces in the beams of the skeleton) corresponding to the occurrence of different damage development scenarios.

References

[1] Bathe K J 2014 Finite element procedures (Watertown, Ma: K.J. Bathe)
[2] Wriggers P 2006 Computational contact mechanics (Berlin-Helderberg: Springer)
[3] Korobeynikov S N 2000 Nonlinear deformation of solids (Novosibirsk: Publishing House SB RAS)
[4] Doronin S V and Filippova Yu F 2018 Modeling of the multi-level process of damage accumulation in a reflector skeleton with an inhomogeneous structure Diagnostics, Resource and Mechanics of materials and structures 6 191-202