CIRP Annals - Manufacturing Technology 64 (2015) 17–20

Contents lists available at ScienceDirect

CIRP Annals - Manufacturing Technology

journal homepage: http://ees.elsevier.com/cirp/default.asp

Variation simulation of stress during assembly of composite parts

Rikard Söderberg (2), Kristina Wärmefjord *, Lars Lindkvist

Chalmers University of Technology, Department of Product and Production Development, SE 412-96 Gothenburg, Sweden

ARTICLE INFO

Keywords:
Assembly
Composite
Tolerancing

ABSTRACT

Weight reduction requirements in aerospace and automotive industry lead to an increased use of composite materials. However, composite parts cannot be bent like sheet metal parts. Hence, only low forces can be applied to close gaps between parts, caused by geometrical variation in parts and assembly fixtures. Shimming is therefore used to compensate for bad fitting, with increase cost as a consequence. This paper investigates how variation in assembly fixtures and parts give rise to variation in gaps and thereby also to variation in stress. Monte Carlo simulations are used to find the distribution of stress, which supports shimming strategies.

1. Introduction

Sustainability requirements in aerospace industry drive reduced product weight, which is facilitated by an increased use of composite materials. Composite parts do not behave like sheet metal parts and extra care need to be taken in order to not bend the parts in such a way that they are damaged during assembly. In order to find out how much bending that can be allowed, the level of stresses during assembly must be calculated. The stress is however caused by geometrical *variation* in parts and in assembly fixtures. To predict the stress it is therefore necessary to simulate the statistical distribution of stress, not only a value of stress for a specific loading. The maximum value of the stress is, due to a nonlinear behavior of the assembly, not necessarily reached when all tolerances are at their absolute maximum. This nonlinear behavior is caused by rotations of the parts due to variation in part geometries and fixtures, leading to bending due to contacts between parts during assembly. Therefore, in this paper a Monte Carlo (MC) simulation approach is used to calculate the statistical distributions of stress during assembly of non-rigid composite parts. In this way, the maximum value of stress due to geometrical variation can be estimated with high accuracy. The simulation procedure is similar to the one for statistical variation simulation, where levels of geometrical variation in assemblies are predicted.

Variation simulation is extensively used in automotive industry in early phases of the product realization loop to compare different designs and to analyze different tolerancing strategies. There are two main approaches to statistical tolerance analysis: the MC simulation-based approach and the deterministic methods, often based on Taylor's series expansion. The MC simulation-based approaches can be done using direct Monte Carlo (DMC) simulation or by a linearization. For a DMC-based variation simulation distributions for all input parameters are defined. In each DMC iteration values of the input parameters are randomly sampled from the defined distributions. For analysis of non-rigid parts, finite element analysis (FEA) is used to calculate the response in the output parameters. Usually, thousands of iterations need to be run to get a good accuracy in this kind of simulation, which will be very time consuming, since a FEA must be run in each iteration. Therefore, the method of influence coefficient (MIC) [1] is used in most MC based variation simulation approaches. The main idea of MIC is to find a linear relationship between part deviations and assembly deviations after spring-back. A sensitivity matrix, constructed using FEA, describes that linear relationship. The sensitivity matrix is then used to calculate the response in each MC iteration. The method was used by Camelio et al. [2], who applied it to a multi-station system. Dahlström and Lindkvist [3] investigated how to combine MIC with contact modeling. Contact modeling was also further developed by Wärmefjord et al. [4]. Variation simulation for non-rigid sheet metal parts and assemblies is described in Refs. [5,6]. Variation simulation for composites is treated in Ref. [7].

Deterministic stress analysis is frequently used in a variety of engineering application. However, simulation of statistical stress distributions, taking variation in different input parameters into consideration, is not as frequently treated in the literature. In later years, some non-deterministic work has though been presented. Simulation of stress distribution based on MIC for one single plastic part is treated by Lorin et al. [8]. They also compare the MC method for calculating stress distributions with DMC. DMC for calculation of stress distributions is treated in Refs. [9,10]. DMC for stress is however a time consuming task and the number of possible MC iterations are strongly limited.

1.1. Scope of the paper

In this paper a novel method for computation of the distribution of stress in unidirectional composites during assembly is

* Corresponding author.
E-mail addresses: rikard.soderberg@chalmers.se (R. Söderberg), kristina.warmefjord@chalmers.se (K. Wärmefjord).

http://dx.doi.org/10.1016/j.cirp.2015.04.048
0007-8506/© 2015 CIRP.

presented. The method is based on MC/MIC simulation and contact modeling between parts is included. The stress levels are presented as percentage of failure stress, using the Tsai–Hill criterion.

In Section 2 the Tsai–Hill failure criterion is presented. Aspects of assembly of composite parts are discussed in Section 3. The suggested method is described in Section 4 and illustrated by a case study in Section 5. Finally, conclusions that summarize the suggested procedure are found in Section 6.

2. Failure criteria for composites

Composite materials are a combination of two or more different materials. One of the most common kinds of composite material is unidirectional CFRP (Carbon Fiber Reinforced Plastic). It consists of a number of thin plies of a polymer matrix reinforced with carbon fibers in one direction. The plies are stacked and different plies have different fiber directions. Each ply is anisotropic, meaning its strength is different in different directions, see Fig. 1. An overview of composites can be found in, for example [11].

The anisotropic behavior of unidirectional lamina hinders the use of for example von Mises stress criterion, which is a frequently used stress criterion for sheet metal parts. Instead, a stress criterion adapted to composites is used. This is a large research area and a number of overviews and comparisons are given by, for example [12] and [13]. In this paper, the Tsai–Hill criterion [14] is used. The Tsai–Hill criterion, seen in Eq. (1), is a modification of the von Mises criterion and is merging the stresses in different directions and comparing them with the respective failure stresses. Under the assumption of a unidirectional composite and plane stress, the criterion can be expressed as [11]:

$$\left(\frac{\sigma_1}{\sigma_{1u}}\right)^2 + \left(\frac{\sigma_2}{\sigma_{2u}}\right)^2 - \frac{\sigma_2\sigma_2}{\sigma_{1u}^2} + \left(\frac{\tau_{12}}{\tau_{12u}}\right)^2 = 1 \quad (1)$$

Here, σ_1 is the axial tensile stress (i.e. in direction parallel to the fibers), σ_2 is the transverse tensile stress (i.e. orthogonal to the fibers) and τ_{12} is the shear stress, see Fig. 1. The values σ_{1u}, σ_{2u} and τ_{12u} are the values of the failure (ultimate) stress. The failure compressive and tensile stresses differ for anisotropic material, therefore the ultimate stresses are defined as:

$$\sigma_{1u} = \begin{cases} \sigma_{1uc} & \text{if} \quad \sigma_1 \le 0 \\ \sigma_{1ut} & \text{if} \quad \sigma_1 > 0 \end{cases}$$

$$\sigma_{2u} = \begin{cases} \sigma_{2uc} & \text{if} \quad \sigma_2 \le 0 \\ \sigma_{2ut} & \text{if} \quad \sigma_2 > 0 \end{cases}$$

The indices c and t correspond to compressive and tensile values. For the term $(\sigma_1\sigma_2)/\sigma_{1u}^2$ in Eq. (1),

$$\sigma_{1u} = \begin{cases} \sigma_{1uc} & \text{if} \quad \sigma_1\sigma_2 < 0 \\ \sigma_{1ut} & \text{if} \quad \sigma_1\sigma_2 \ge 0 \end{cases}$$

When Eq. (1) holds, the stress has reached a critical level. The equation defines an envelope in stress space, and if the calculated stress levels σ_1, σ_2 and τ_{12} are outside this envelope the stresses lead to failure [11]. If the left side of Eq. (1) is <1 for a large number of MC iterations the probability of failure due to stress is very low (typically a few ppm). The exact probability of failure is dependent of number of MC iterations and the distribution and capability of the part tolerances.

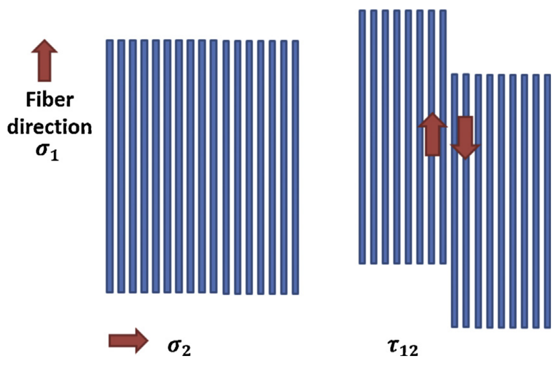

Fig. 1. Axial tensile stress, transverse tensile stress and shear stress.

3. Assembly of composite parts

Composite parts tend to deviate more from nominal values compared to sheet metal parts. This, in combination with the fact that they cannot be bent like sheet metal parts without risking failure, often make machining of parts necessary. This is however both difficult and expensive [15] and should if possible be avoided.

In the design of a composite assembly, interfaces between parts are often designed with gaps to handle geometrical variations stemming from the manufacturing process [16]. Those gaps need to be shimmed during assembly. If liquid shimming is used, the assembly cannot be moved during the, often quite long, curing time. Shimming is thereby often a bottleneck in the assembly workflow [16].

In conclusion, both machining and shimming are costly and the assembly process of composites need to be improved in order to deal with large part deviations.

4. Suggested method

To handle assembly variation induced by variation in parts and variation in the assembly process, it is necessary to do a detailed study of how these variation sources propagate during assembly. In this paper, variation simulation based on MC/MIC simulations is used as a basis. The simulation method uses an assembly model, containing meshes of parts, information about locating schemes and assembly methods, part tolerances, fixture tolerances, contact modeling, etc. The locating schemes describe how each part is positioned in its fixture during assembly. A non-rigid locating scheme consists of six master locating points (MLPs) and an arbitrary number of additional support points. For more information about non-rigid locating schemes, see [17].

This kind of model is normally used to calculate geometrical variation on assembly level. Here, the model is extended to also handle stress in parts during assembly due to geometrical variation. The method described in this section is implemented in the software RD&T [18].

MC/MIC based stress calculations are earlier presented by Lorin et al. [8] for a single plastic part. Here, that work is further developed to also handle composites and assemblies with contact modeling between parts. Contact modeling is a way to avoid that parts in the virtual model penetrate each other due to possible imperfections and variation in parts and tools. Instead, resulting forces due to collisions are transferred to the parts via contact points or surfaces. The contact modeling used here is point based, i.e. a number of point pairs are defined [4,19]. Each contact pair consists of one node from the master part and one corresponding node from the slave part. A negative distance between the nodes corresponds to penetration and must be avoided.

The parts are joined together (drilled and screwed) in a number of joining points. The joining points are treated like two contact pairs with opposite directions in the calculations described below. The two contact pairs will force the parts together in the joining point and the parts are then locked to each other in the joining point.

To handle stress calculations it is necessary to use a higher order elements compared to the normal kind of variation simulations where displacements are investigated, since stresses are a function of the derivatives of displacement [8]. Furthermore, small displacements and linear material models are assumed. The calculation procedure can be summarized as:

(1) The parts are positioned using their MLPs and support points.
(2) The stress support sensitivity matrix, $\mathbf{S}_{s\sigma}$, is calculated by applying a unit disturbance to the support points, one at the