Journal of Manufacturing Systems 37 (2015) 44–71

Contents lists available at ScienceDirect

Journal of Manufacturing Systems

journal homepage: www.elsevier.com/locate/jmansys

Technical Paper

Resistance spot welding process simulation for variational analysis on compliant assemblies

Sandro Moos*, Enrico Vezzetti

Politecnico di Torino, DIGEP – Department of Management and Production Engineering, Corso Duca degli Abruzzi 24, 10129 Torino, Italy

ARTICLE INFO

Article history:
Received 16 December 2014
Received in revised form
14 September 2015
Accepted 17 September 2015
Available online 9 October 2015

Keywords:
Compliant assembly
FEM
Variational analysis
Resistance spot welding
Plasticity

ABSTRACT

This work proposes a methodology that can be used to define a FEM simulation of the body welding process with the aim of evaluating compliant assembly deformations and spring-back, considering the effect of material plasticity, in order to improve the results of variational analysis methods, which so far have been based on a linear elastic material model. With reference to the automotive field, the simulation considers the effects of fixturing and resistance spot welding applied to sheet metal parts subjected to dimensional and geometrical tolerances.

1. Introduction

In order to rapidly meet the ever-changing needs of the customers, the automotive manufacturers need to reduce the time required to develop, produce and deliver a new product on the market.

One of the problems that arises in this context is related to the process and product design changes that are required to resolve the quality problems that can occur during production ramp-up. In order to make the project more robust and to solve any potential problems in advance, tolerance stack-up analyses are performed in the early stages of the product/process development to evaluate the impact of part tolerances on the quality and assemblability of products.

Several simulation models have already been developed to evaluate variation propagation in an assembly system; as pointed out by Maropoulos et al. [1] they range from 1D models that apply the worst case or the root sum square formulation, to 3D simulations that consider rigid parts and kinematic constraints, and which are integrated with the Monte Carlo statistical approach. However, their application is limited in the automotive sector, because of the complex geometry and compliance of the parts.

Compliant models are based on the influence coefficient method, which was first proposed by Liu et al. [2]. The method extracts the linear relationship between the deviation vector of the parts and the elastic spring-back of the assembly from a FEM analysis. Each part is considered to be subjected to deviations and to be forced into the nominal position by the clamps. The relationship between the deviation vector and the reaction forces of the parts is extracted from the FEA, as a sensitivity matrix: a unit force is applied to each source of variation, in the same direction as the variation, and the corresponding deformation of the part is then measured and ordered in a vector. The elastic spring-back of the assembled structure is extracted from the assembly stiffness matrix by considering it as being subjected to a force that is equal and opposed to the sum of the clamping forces required to close the parts in the nominal condition. This approach requires the matrices to be calculated only once, and then a Monte Carlo simulation is used to extend the validity of the linear relationships to arbitrary deviation vectors in order to obtain a statistical description of the tolerance stack-up, in terms of probability functions and contributors. A detailed application of the influence coefficient method in the aerospace field can be found in Lee et al. [3].

Other methods consider the effects of variations due to parts, fixtures and welding guns on the final assembly variations: Camelio et al. [4] proposed a methodology for the analysis of variation propagation, on multi-station compliant sheet metal assembly lines, based on a state space representation of the deviations of the parts.

Abbreviations: RSW, resistance spot welding; FEM, finite element method; d.o.f., degree of freedom; FE, finite element; B.C., boundary condition; W.P., weld point.

* Corresponding author. Tel.: +39 0110907267; fax: +39 0110907299.
E-mail addresses: sandro.moos@polito.it (S. Moos), enrico.vezzetti@polito.it (E. Vezzetti).

http://dx.doi.org/10.1016/j.jmsy.2015.09.006
0278-6125/

The influence coefficients and homogeneous transformations were used to obtain the relocation, deformation and sensitivity matrices of each station. The results showed that the optimal assembly sequence depends on the geometrical configuration (shape) of the assembly, and on fixture and tooling variations.

The body welding process was considered, in terms of tool variations, clamping deformations, joining and spring-back, by Liu et al. [5] and Camelio et al. [6,7]. Wärmefjord et al. [8] determined the best welding sequence by minimising the relative displacements measured in the positions corresponding to the remaining unwelded points, while Söderberg et al. [9] found that a variation in the weld point (W.P.) position from the nominal position, had a significant effect on the variations of the resulting assembly.

The effect of the contact between parts during the body welding operations has also been considered. For example, Tonks et al. [10] accounted for surface variation through a hybrid method that models surface covariance, considering three typical manufacturing process variation descriptors: warping, waviness and roughness. Xie et al. [11] considered the effect of a nonlinear frictional contact analysis in a multi-step FEM simulation of a welding process: the non-nominal parts were located on a 3–2–1 fixture, and other additional clamps closed the parts to the nominal position. The welding guns were then closed, thus producing additional deformations. Finally, the parts were joined together at the W.P. locations, the gun and clamps were released and the assembly was left free to spring-back. The authors used the Enhanced Dimensional Reduction Method to estimate the nonlinear response of the contact behaviour and reduce the computation time. They obtained a better prediction the variation propagation in nonlinear contact assembly processes.

The methods described above are all based on the “historical” hypothesis of linear elastic model applied to the parts, while the plasticisation around the welding nugget has also been confirmed through process studies, such as those of Feulvarch et al. [12], Hou et al. [13], Nodeh et al. [14], Eisazadeh et al. [15]. Further confirmation of the importance of the spring-back evaluation with a plastic material model has been obtained from analyses of the stamping process: Panthi et al. [16] evaluated the spring-back dependency on material considering the properties of yield stress, Young’s modulus, strain hardening and on geometric parameters such as sheet thickness, die radius and sector angle. Friction was found to have a negligible effect on spring-back.

The importance of a plastic material model has been evaluated by Moos et al. [17], by means of a FEM simulation, with solid 3D elements along the part thickness, and structured as an electro-thermo-mechanical problem, using a plastic material model. The entire welding process has been simulated, considering the geometrical conditions of gap or interference. As a result, the fixture closure permanently deforms the part around the fixed locators, while the material melting caused by the welding current at the weld nugget makes the material plastic and reduces the amount of elastic energy stored in the parts, which is responsible for spring-back. Finally, Moos et al. [18] have outlined the FE methods that can be used to apply the complex interaction of the resistance spot welding process (RSW) to a shell model. The spring-back of the shell model resulted to be in good agreement with that obtained with the 3D elements, thus confirming the possibility of carrying out an accurate simulation while reducing problem complexity.

Fig. 1 shows the different spring-backs computed on the same model, using two material models: the case refers to a butt joint with one weld spot on the flanges which represent a gap condition. The sheets are constrained at the end opposing the flange in all directions, while the welding caps are constrained only in the radial direction for permitting the clamping motion and along the axial direction the clamping load is applied. Contact constraints are active at the sheet-to-sheet and sheet-to-weld-gun interfaces. Parts are modelled using C3D8 solid elements. Sheet parts are subjected to the welding process, consisting of weld-gun closure, heating, cooling, weld-gun opening and fixture release with the part on the right that remains constrained after release.

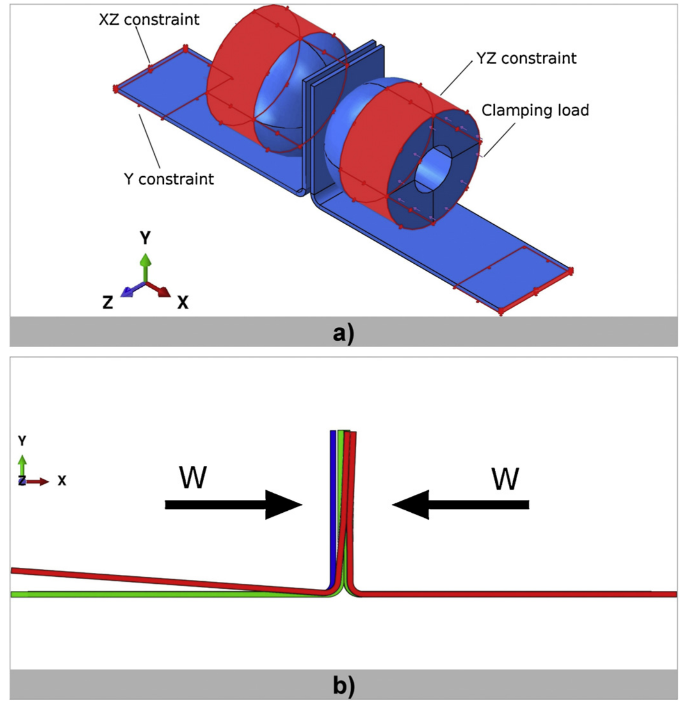

Fig. 1. Spring-back of a butt joint. (a) Model layout, constraints and load. (b) Deformed shape comparison: in blue it is shown the undeformed condition, in green the elastic material model shape, and in red the plastic material model shape. W is the welding direction. Results from [18]. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

In Fig. 1b different images are superimposed: the blue flange on the left corresponds to the un-deformed condition, the green shape corresponds to the elastic results while the red refers to the plastic model.

The elastic model seems to describe a final displacement of the left part that closes the initial gap. Instead the plastic model shows permanent deformations on the flanges that are not in contact towards the base, and also causes a rotation of the left part. The importance of analysing the plastic material model emerges from this result.

Apart from these works, no other evidence of developments in FEM techniques to tackle tolerance stack-up analysis has been found in literature. The focus of researchers seems to be oriented more towards fixture fault detection as in the works of Ceglarek et al. [19], towards layout optimisation, as in Liao et al. [20], or towards part by part tolerance compensation on the fixture, as in Xie et al. [21]. In the aforementioned works, the parts were assumed compliant and the required FEM simulations considered an elastic material model.

2. Proposed method

Sheet metal parts are subjected to geometrical and dimensional tolerances which are caused by the stamping process. When parts are placed on a fixture during body welding operations, the two situations described in Fig. 2b and d can occur: in the first situation, the welding flanges are not in contact and present a gap condition; in this case, the fixture locks the parts, the action of a welding gun closes the flanges and the weld point is then made.