Mechanical Systems and Signal Processing 98 (2018) 382–401

Contents lists available at ScienceDirect

Mechanical Systems and Signal Processing

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

Obtaining manufactured geometries of deep-drawn components through a model updating procedure using geometric shape parameters

Vamsi Krishna Balla [a,b,*], Laurens Coox [a,c], Elke Deckers [a,c], Bert Plyumers [a,c], Wim Desmet [a,c], Kannan Marudachalam [b]

[a] Machine Design and Automation (PMA), Department of Mechanical Engineering, Celestijnenlaan 300 – box 2420, 3001 Leuven, Belgium
[b] Research & Development, TVS Motor Company, Hosur, Tamil Nadu, India
[c] Member of Flanders Make, Belgium

A R T I C L E I N F O

Article history:
Received 17 January 2017
Received in revised form 10 April 2017
Accepted 7 May 2017
Available online 16 May 2017

Keywords:
Modal updating
Modal Assurance Criterion
Optimization
Deep drawing

A B S T R A C T

The vibration response of a component or system can be predicted using the finite element method after ensuring numerical models represent realistic behaviour of the actual system under study. One of the methods to build high-fidelity finite element models is through a model updating procedure. In this work, a novel model updating method of deep-drawn components is demonstrated. Since the component is manufactured with a high draw ratio, significant deviations in both profile and thickness distributions occurred in the manufacturing process. A conventional model updating, involving Young's modulus, density and damping ratios, does not lead to a satisfactory match between simulated and experimental results. Hence a new model updating process is proposed, where geometry shape variables are incorporated, by carrying out morphing of the finite element model. This morphing process imitates the changes that occurred during the deep drawing process. An optimization procedure that uses the Global Response Surface Method (GRSM) algorithm to maximize diagonal terms of the Modal Assurance Criterion (MAC) matrix is presented. This optimization results in a more accurate finite element model. The advantage of the proposed methodology is that the CAD surface of the updated finite element model can be readily obtained after optimization. This CAD model can be used for carrying out analysis, as it represents the manufactured part more accurately. Hence, simulations performed using this updated model with an accurate geometry, will therefore yield more reliable results.

1. Introduction

Numerical computational tools like the Finite Element Method (FEM) proposed by Zienkiewicz [1] have become very powerful in aiding the design of structural systems. As some very important decisions – cost, weight and design finalization – will be made based on the results of the simulations using the finite element (FE) models, it is extremely important to develop FE models that are accurate representations of their physical counterparts. Therefore, correlation and validation

* Corresponding author at: Machine Design and Automation (PMA), Department of Mechanical Engineering, Celestijnenlaan 300 – box 2420, 3001 Leuven, Belgium.

E-mail address: vamsikrishna.balla@kuleuven.be (V.K. Balla).

http://dx.doi.org/10.1016/j.ymssp.2017.05.009
0888-3270/

of the behavior of real structural systems with corresponding numerical models is very important. The crucial importance of physical experiments through which one can understand the behavior of actual systems cannot be overstated. Numerical simulations can only complement physical experimentation but cannot totally replace them. Once a modeling methodology has been proven to mimic experimental observation, numerical simulations using the proven methodology could be a powerful route to design of actual systems.

Building reliable FE models of mechanical and structural systems to carry out dynamic analysis can be quite complex. The sources of errors in modeling these systems may have frequency dependence. The impact of these errors – inaccurate loads, boundary conditions, and system characteristics – on the system behavior gets amplified with increasing frequency. Further, though one could obtain reasonably good quality numerical models of individual components, getting a good quality system-level models is more difficult due to poor interface (connector) modeling.

Traditionally, modal updating using material properties – Young's modulus, material density and damping ratio – was used by various researchers in order to validate FE models. Mottershead et al. [2], published a literature review on modal parameter updating. Mottershead et al. [3] proposed an eigenvalue sensitivity approach to update FE models to model welded joints accurately. Yuan et al. [4] proposed a more accurate two stage modal updating. The stiffness and mass matrices are updated first, followed by updating with damping. More recently, Ning Guo et al. [5] demonstrated that the strain-based FRF updating procedure is more accurate compared to a modal parameter updating procedure. Maletta et al. [6] proposed a method, using a finite element model and Genetic Algorithms (GA), to obtain elastic constants of composite laminates. Meruane [7] proposed a methodology to identify anti-resonances using transmissibility functions. The anti-resonances of finite element model were updated using a GA based optimization method. Petrone et al. [8] carried out model updating of a composite panel made of unidirectional flax fibres embedded in a polyethylene matrix (flax-PE) flat panel. They updated the mechanical properties, used in the FE model, using an inverse modelling method based on parallel genetic algorithms in order to obtain the minimum difference between the numerical and experimental data. It can be seen that the modal parameter based approaches are well suited for updating of assembly-level FE models, so as to model interface properties of joints, bushings and welds properly.

In case of components (with high draw ratios) manufactured through a deep drawing process, where typically no joint exists, modal updating using material properties alone won't be sufficient to achieve the desired accuracy in predictions. Andersson et al. [9] carried out both numerical and experimental work to evaluate spring back of an automobile front side member. They studied the effect of different materials on spring back. They found up to 7.5 mm profile deviation from the original CAD profile happened due to spring back effect. Hancock et al. [10] showed that part-to-part variation contributed with 21% of total variation. This variation was primarily because of random spring back associated with the process. Kim et al. [11] showed that spring back of 4 mm was very likely for an automobile motor-case with 40 mm radius. Hence they resorted to process improvement by implementing multi-forming along with elliptical core for reduction of spring back. They were able to achieve an excellent reduction in spring back value of 0.038 mm with redesigned elliptical core. de Souza et al. [12] carried out stochastic simulations to quantify the contribution of process parameters – Blank Holder Pressure (BHP) and coefficient of friction – and mechanical material properties – Yield stress and Ultimate strength – on spring back of an auto-mobile component. They showed changes in process parameters resulted in changes in strain component thus effecting the magnitude of external work applied to forming system. Material property variation, on the other hand, effected stress component of the response. Padmanabhan et al. [13] studied thickness variation of a LPG bottle manufactured through deep drawing process. The main objective of their study was to reduce thickness variation in the deep drawn part, i.e. to maximize the minimum thickness. They proposed various blank holder force strategies – highest, lowest, optimal, linearly increasing, linearly decreasing and adaptable. Their study indicated that about 25% thickness reduction happened when the highest blank holder force was applied. The required component thickness was achieved when optimum BHP was used. Mouatassim et al.[14] used PAM STAMP software to analyze thickness variation of a dash-pot cup of the RENAULT Twingo. It was found from their study that about 10% thickening and 20% thickening happened at the critical locations.

It can be seen from the work done by above researchers that significant geometry profile and thickness variations occur in the components manufactured by deep drawing. Hence, modal updating of FE models using material properties alone will not be sufficient to achieve the desired accuracy in predictions. According to the best of authors' knowledge, there is no literature that addresses updating procedures taking profile and thickness variations into account which occur in deep drawing process. In this research work, a novel method of modal updating is proposed by incorporating geometry shape variables, defined using morphing of the finite element model obtained from the original CAD, which mimic changes happened during the manufacturing process, to construct the deviated profile of component. Modal Assurance Criterion (MAC) based optimization is carried out using combination of a FE solver and an optimization solver that uses the GRSM algorithm. Matlab [15] is used as a simulation manager in order to run the FE morphing software and optimization solvers. The advantage of the proposed methodology is that the CAD surface of the updated modal model can be obtained. This CAD model can be used to design die and punch for successive designs or for carrying out numerical analysis, as results of such updated part analysis will be more accurate than those of analysis results where the finite element model is obtained from the original CAD model. The outline of the paper is as follows – In the first and second sections of paper, a brief theoretical description of modal updating and geometry based shape parameter updating procedure is given. An industrial application oriented component – an oil-pan look-a-like – is designed and manufactured using a deep-drawing manufacturing process. It is demonstrated by incorporation of geometry shape variables, a reliable finite element model can be obtained through optimization. Finally, the procedure to obtain an updated CAD model of the manufactured component using geometry based shape parameter opti-