Available online at www.sciencedirect.com

ScienceDirect

Procedia CIRP 75 (2018) 285–290

www.elsevier.com/locate/procedia

15th CIRP Conference on Computer Aided Tolerancing – CIRP CAT 2018

FEA integration in the tolerance analysis using Skin Model Shapes

Andrea Corrado*, Wilma Polini

Department of Civil and Mechanical Engineering, University of Cassino and Southern Lazio, via G. di Biasio 43, Cassino 03043, Italy

* Corresponding author. Tel.: +39-0776-2993698. *E-mail address:* a.corrado@unicas.it

Abstract

Many research works on tolerance analysis have been carried out in the last thirty years. In this paper, a new idea is proposed, aiming to investigate the effect of form error and mechanical behavior of parts on the stack-up result by combining the recent novelties on the tolerance analysis of rigid and flexible bodies. The real parts are simulated considering the non-nominal Skin Model Shape and the mechanical properties in order to simulate the assembly of a realistic case study. A manufacturing signature model to generate the features with geometric deviations and Finite Element Analysis (FEA) are used.

Peer-review under responsibility of the Scientific Committee of the 15th CIRP Conference on Computer Aided Tolerancing - CIRP CAT 2018.

Keywords: Skin Model; Mechanical behavior; FEA simulations

1. Introduction

1.1. The motivation of this work

A mechanical assembly consists of two or more components or subassemblies. Owing to variations in manufacturing, it is impossible to completely avoid deviations from nominal in a component's dimensions and geometry. Tolerance analysis enables the prediction of the effects of these part deviations on assembly functional requirements.

Historically, tolerance analysis of rigid assemblies has always been separated by that of compliant ones. This is due to many reasons, such as the fact that rigid assemblies reason on features to which tolerances are applied, they do not use finite element models (FEMs) and they rarely allow to consider form deviations. The compliant assemblies deal with nodes, they use only shell elements in FEM, that do not allow to take into account deviations in the thickness of the component, and takes into account form deviations too.

The aim of this work is to provide a method to numerically execute a tolerance analysis without dividing rigid from complaint assemblies. That is, to create a single tool where all functions, which characterize the two kinds of analysis, can be available at the same time: material properties, geometrical deviations along all the part dimensions, form deviations, stress and deformation evaluation.

To fully understand the novelty of the proposed approach, a deep analysis of the state of art was carried out in sections 1.2, 1.3 and 2.

1.2. Tolerance analysis of rigid assemblies

Over recent decades, a substantial number of mathematical models for tolerance analysis of rigid assemblies have been proposed, i.e. virtual boundary [1], variation model [2], TTRS model [3], matrix model [4], vector loop [5], Jacobian model [6], torsor model [7], Jacobian-torsor model [8], T-Map model [9], deviation domain [10] and skin model [11]. They reason on features to which tolerances are applied and, therefore, they are not able to deal with form deviations; moreover, they hardly handle with free form surfaces or assemblies with many components and tolerances.

Moreover, many commercial software packages exist and allow to make the tolerance analysis of rigid assemblies. These software packages are based on some approaches cited previously, such as MECAMaster®, Sigmund®, 3DCS®, VisVSA®, CeTol® and PolitoCAT®. They efficiently deal with simple geometrical feature of mechanical assemblies,

2212-8271 © 2018 The Authors. Published by Elsevier B.V.
Peer-review under responsibility of the Scientific Committee of the 15th CIRP Conference on Computer Aided Tolerancing - CIRP CAT 2018.
10.1016/j.procir.2018.04.055

such as plane, hole-pin, while they hardly treat free-form surfaces to connect.

1.3. Tolerance analysis of compliant assemblies

To analyze compliant assemblies many methodologies have been proposed, i.e. the Place-Clamp-Fasten-Release (PCFR) method [12], the flexible assembly statistical tolerance analysis (FASTA) [13] and the method of influence coefficients (MIC) [14]. Many other methods where the hypothesis of linearity in the mechanical behavior of the involved materials, in the contact among the bodies and in the influence of the fixturing have been exceeded [15–18].

In contrast to the models used for the tolerance analysis of rigid assemblies, a real distinction between these methods does not exist because all methods are characterized by a finite element analysis (FEA) to evaluate part deformations, forces and internal stresses due to assembly, etc. All these methods consider flat components that are schematically represented with a middle plane by means of shell elements.

The software packages for the tolerance analysis of compliant assemblies are: TAA® [19], 3DCS-FEA®, VisVSA-FEA®, RD&T and ANATOLEFLEX [20].

2. Status of Skin Model Shapes

The Skin Model is a recent concept of modern standards for the specification and representation of product [21]. Starting from it, the Skin Model Shapes concept has been developed to overcome the limits of established models for the computer-aided modelling and representation of rigid part geometry considering all different kind of geometric variations by discrete geometry representations [22].

The tolerance analysis based on Skin Model Shapes foresees three stages. The first stage is a pre-processing stage where the Skin Model Shapes of parts are generated. The generation of the Skin Model Shape foresees four steps:

- definition of the nominal model of part;
- discretization and segmentation of the nominal model in order to treat each feature independently;
- simulation of manufacturing deviations on each feature;
- combination of all geometric deviations and nominal model for a complete Skin Model Shape.

The discretization of the nominal model can be generated easily by FEA software or also by applying Delaunay algorithm [23]. The simulation of the deviations on each feature can be carried out by using different methods that can be classified into three categories: random noise [24], mesh morphing [25,26] and modal-based methods [11,26–30]. These methods do not necessarily lead to part representatives, which conform to specified tolerances so, the Skin Model Shapes are "scaled" [31] by using algorithms [32] for the evaluation of geometric tolerances from point clouds. The combination of all geometric deviations for a complete Skin Model Shapes can be modelled by FEA software [33].

The second stage is a processing stage where the Skin Model Shapes are assembled according to the defined assembly process employing the positioning scheme. To do this, two approaches are necessary [34]. The first approach identifies the contact points in a specified assembly direction between two mating parts and it enables to simulate the assembly through a 3-2-1 positioning scheme with iterations of the single assembly steps. In the second approach, the positioning problem is formulated as a constrained registration problem and solved using mathematical optimizations in order to minimize the sum of projected distances between the set of points in the moving part and their correspondences in the mating part, such that these projected distances do not take negative values.

The third stage is a post-processing stage where the assemblies are evaluated regarding the contact quality between the assembled parts, using point projection methods to obtain the projected distances between the parts, the functional requirements are measured such as minimal gap or maximum distance and finally the results are shown and analyzed [35].

The same scheme has been adopted for the new unified tolerance analysis tool that is deeply described in the following paragraph.

3. Integration of mechanical behavior of parts into the Skin Model

The tolerance analysis based on Skin Model Shapes model is a field where a lot of researches has been done in the recent years. As summarized in section 2, FEA software is much used in the pre-processing stage to discretize and to combine all deviations for a complete Skin Model Shapes representation. Moreover, a first integration of FEA software to study the deformation effects on the Skin model provoked by the thermal and working environment of a gas turbine blade has been presented in [36].

Therefore, instead of using the FEA software as auxiliary tool to discretize features and combine all deviations, the FEA software could be used as the main tool together to the Skin Model Shape concept that handles the deviations to be assigned to all discretized features. In addition, the FEA software generally may handle contacts between the bodies, therefore, many functions developed for the processing stage are already known and implemented in these software packages. Moreover, these kinds of software can visualize results as the stress, deformation, displacement, contact quality, etc.

The aim of this work is to provide a numerical tool that executes tolerance analysis by discretizing each component volume by means of brick elements whose geometry may be affected by geometrical deviations. This numerical tool is funded on a FEA software integrated with Skin Model Shapes concept. Its framework (see Fig. 1) is constituted by three main stages: pre-processing, processing and post-processing ones. They are described in details in the following paragraphs.

The entire numerical architecture is based on MSC Marc® solver and Matlab® environment that interact, as shown in Fig. 1, and this tool was called "CaUTA" (Cassino Unified Tolerance Analysis).

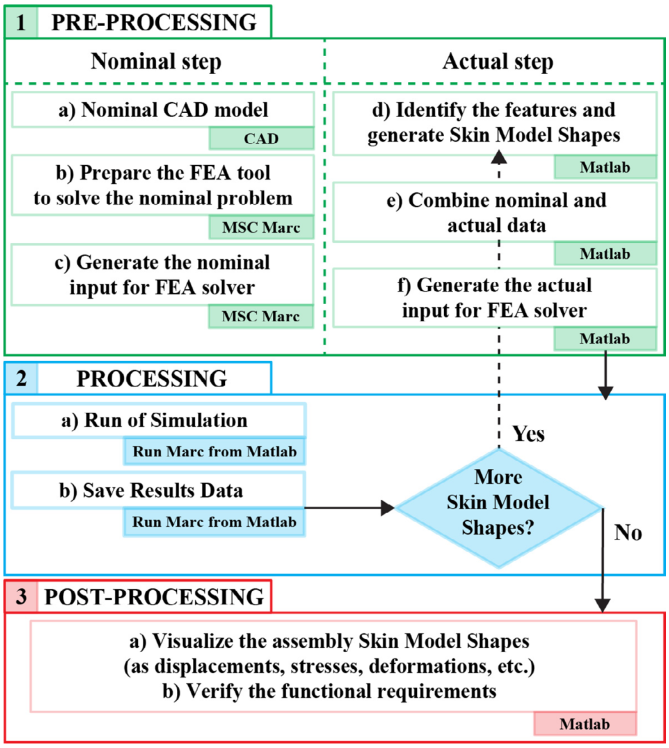

Fig. 1. Process flow to execute tolerance analysis with CaUTA.

3.1. Pre-processing: FEA and Skin Model Shapes integration

The pre-processing stage involves two steps: a nominal and an actual ones. In the first step, the FEA tool is prepared to solve the nominal problem without considering the geometrical deviations. Therefore, the nominal CAD model is designed and imported in the FEA software. In the user interface of MSC software, the part or/and the assembly are discretized by solid 2D/3D elements. During the mesh generation, the size of the generated elements has to be carefully considered because very small elements, together with the deviations of the Skin Model Shapes, could distort and, therefore, cause convergence problem during FEA simulations.
Afterwards, the material properties are defined. In case of a mechanical simulation, at least the density, Young moduli and Poisson's ratios are necessary. In complex cases, other information could be necessary such as the hardening law of materials, specific heat capacity, thermal conductivity, coefficient of thermal expansion, etc. In addition, the boundary conditions (BCs) and the loads that affect the parts of assembly have to be defined. Usually, user-defined functions, material models or load models can be added. Finally, the input for the FEA solver (.dat file) that contains all this information is generated.

In the actual step, the discretized features, where to apply the deviations, are identified and the Skin Model Shapes are generated on them. The generation of Skin Model Shapes can be carried out using one of the methods indicated in section 2. Once the Skin Model Shapes are completely generated, the generated errors are summed to the node coordinates of the nominal features without re-meshing the elements. At this point, the new input for the FEA solver is generated with the nodes of the actual geometry and it is sent to the processing stage.

3.2. Processing: FEA simulation

In the processing stage, the FEA software solves the structural analysis problem. The actual input file is executed in DOS by a Matlab script in order to perform the FEA simulation. The MSC software elaborates the solution, through the evaluation of the stiffness matrix, in terms of structural displacements, stresses, deformations. Moreover, the problem of parts assembly is solved by considering the contact properties between the bodies defined in the nominal step of the pre-processing stage.

Once the simulation is finished and the results are generated, the FEA results can be post-processed and/or saved. When the results are only saved, new Skin Model Shapes are generated for the features of parts; therefore, the process starts again from the generation of Skin Model Shapes (according to the second step in the pre-processing stage), and another simulation is executed. Once enough Skin Model Shapes are computed, the post-processing stage is carried out.

3.3. Post-processing: Visualization and analysis of results

In the post-processing stage, the numerical tool visualizes the solution as displacements, stresses, deformations and contact quality. The visualization of the assembly Skin Model Shapes is based on the magnitude of the calculated nodal value. The nominal mesh of CAD model is used as a reference frame where the distribution of these deviations (for example the magnitude of the nodal displacements) is plotted.

If the entire simulation foresees N iterations, i.e. N different Skin Model Shapes are generated for each feature, the simulation results can also be represented in terms of mean Skin Model Shape. Moreover, if the results are shown as displacements, the functional requirements, such as minimal gap, maximum distance, etc., can be measured using algorithms from the computational metrology.

4. Application example

4.1. Definition

The case study of this work consists of simulating the assembly between a bushing and a housing. The assembly between these two parts foresees an interference fit.

Dimensions and tolerances are shown in Fig. 2a. In particular, the dimensional tolerance range that contains a form error to be simulated characterizes the external diameter of bushing, as shown in Fig. 2b. A signature of the turning process is considered to represent the form error. The signature is represented by means of the autoregressive-moving average (ARMAX) model proposed in [37].

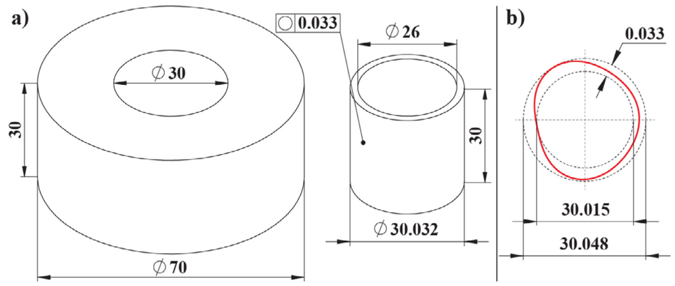

Fig. 2. a) Dimensions of the case study; b) the form error by ARMAX.

Once designed and imported the nominal CAD model in the FEA software, the parts are discretized by solid 3D elements (element type 7). Then, the discretized features where to apply the Skin Model Shapes are identified and the Skin Model Shapes are generated according to the form error to be simulated; finally, the nominal and actual data are combined. Fig. 3 shows the process to generate an actual geometry.

The Skin Model Shape of the external cylindrical surface of the bushing was done through the generation (by ARMAX model), the stacking and the random rotation of 16 profiles for each FEA simulation. Fig. 4 shows the steps from nominal model to actual one.

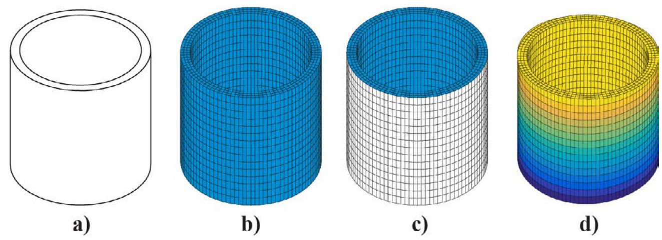

Fig. 3. a) Nominal model; b) Discretized model; c) Identification of the cylindrical feature and generation of Skin Model Shapes; d) Actual geometry.

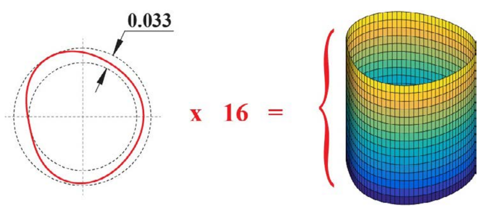

Fig. 4. Detail on the generation of Skin Model Shapes.

Tin-bronze QSn7-0.2 was used as material of both the bushing and housing. However, a raw material was used for the bushing, while a material treated by a power spinning process was used for the housing [38].

The boundary conditions are applied in order to simulate the clamping of the parts during the assembly, as shown in Fig. 5. The housing is constrained on four edges to avoid translations along the three main directions. The bushing is constrained on two edges to avoid translations along the X-Z directions and on other two edges to avoid translations along the Y-Z directions in order to stop the self-centering of it.

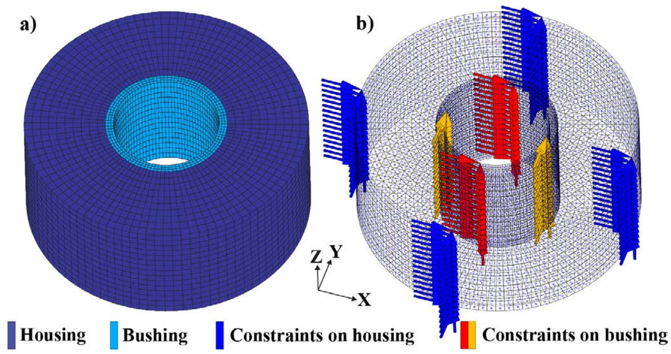

Fig. 5. a) Discretized model of the case study; b) Boundary conditions.

The interference fit capability is treated in FEA software using the contact algorithms. In particular, five methods are available in MSC Marc® and the method called "Contact normal" has been used for this case study. This method is recommended for situations with small values of interference to be resolved along the normal direction to the touched interface. In this case, the nodes of the touching part are projected in the direction normal to the segments of the touched part. The individual touching entities are placed at a certain distance from the touched surface along the normal of the touched surface. This "interference" distance that signifies the maximum overlap between the parts has to be provided by user in order to make the detection algorithm works as follow:

$$\left(A+B\right) \text{ at a node} < C\cdot\left(1\pm D\right) \text{ on the in/outside} \tag{1}$$

where A is the overlap, B is the interference, C is the regular distance tolerance and D is a bias factor [39]. The algorithm detects the presence of interference and moves the parts to be assembled in order to bring them into contact.

4.2. Results and analysis

The assembly between the bushing and housing foresees an interference fit. The entity of interference depends, in this case study, only by the form tolerance on the external cylindrical surface of bushing.

Using the methods for tolerance analysis of rigid bodies, the results of this assembly would highlight the interference magnitude and the relative position of the bushing respect to the housing.

In reality, the parts to be assembled are not rigid but have mechanical properties and consequently different stiffness. Therefore, an interference fit deforms the bodies in order to ensure a final assembly where the parts are locked to each other, due to the arisen stress state, and the interference does not exist anymore.

In the CaUTA tool, once generated a Skin Model Shape for the external cylindrical surface of the bushing and solved the problem of interference with the detection algorithm, the arising stress state can be visualized as shown in Fig. 6. The generation of a single Skin Model Shape and the assembly simulation demonstrate the real influence that small deviations have on the stress level of parts. In particular, the stress state is not uniform on the assembly, especially on the bushing, due to the characteristic form error described by the ARMAX model.

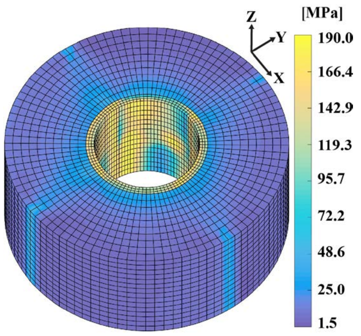

Fig. 6. Stress state considering only a Skin Model Shape.

Generating 1,000 Skin Model Shapes and solving the problem of interference, the arising stress state can be visualized in terms of mean and standard deviation values as shown in Fig. 7a and Fig. 7b respectively. In this case, the mean stress state is uniform on the parts of assembly because of the 1,000 Skin Model Shapes generated but the standard deviation values show again the influence of the form error on the not-uniform distribution of stress.

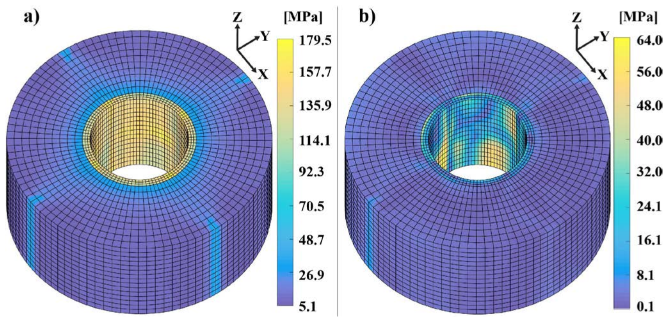

Fig. 7. a) Mean stress; b) Standard deviation of stress.

When a contact algorithm is used, FEA software provides an algorithm to check the contact and then evaluate the contact quality between parts. MSC Marc algorithm was used to evaluate the contact quality. The algorithm indicates whether the parts are more or less in contact using a scale ranging from 0 to 1. For the case study of this work, the parts appear to have a good contact as shown in Fig. 8.

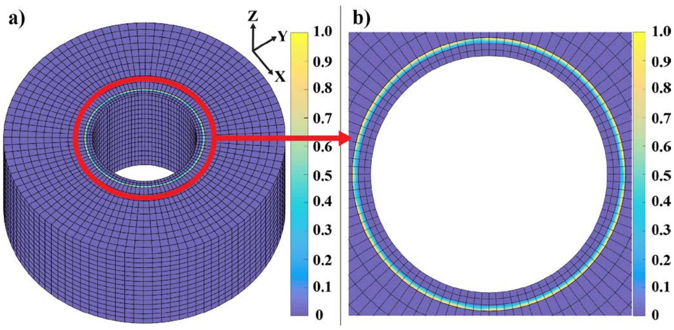

Fig. 8. a) Contact quality; b) Detail on contact quality.

The good contact is also evaluated by analyzing the values of the diameters recorded before and after the assembly of the parts, as shown in Fig. 9. The diameters are evaluated through the mean of different diameters for each cylindrical surface. The internal and external diameters of the bushing decrease while the diameters of the housing increase after assembly. In particular, the external diameter of the bushing is equal to the internal diameter of the housing, so the interference does not exist anymore.

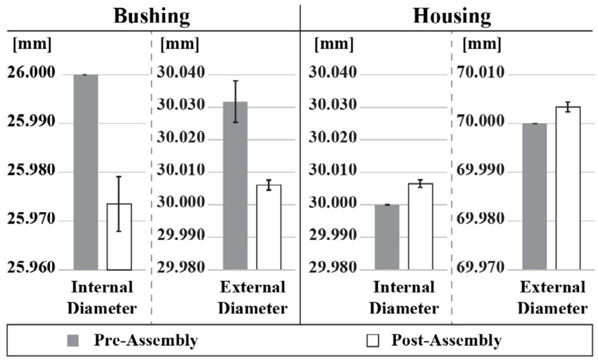

Fig. 9. Diameters of parts before and after the 1,000 simulations.

In this analysis, the 1,000 Skin Model Shapes are considered to be sufficient to capture the physics of the case study and no one computational problem occurred due to the distortions of mesh elements because the considered form error is small.

5. Conclusions

Some guidelines to execute a tolerance analysis without making distinction between tolerance analysis of rigid and compliant assemblies were provided by developing a single tool, called “CaUTA” (Cassino Unified Tolerance Analysis).

The use of FEA and Skin Model Shapes in the tolerance analysis through brick elements demonstrated the real influence that small deviations due to manufacturing errors have on the final geometry shape and stress level of assemblies.

The proposed method, that uses FEA simulations of a set of Skin Model Shapes, provides valuable information on mean value and variability of the interferences between parts that may be used to optimize the product geometry design.

This work seeks to pave the way for unifying the two kinds of tolerance analysis by integrating the FEA and non-nominal geometry captured by Skin Model Shapes in the mechanical assemblies where mechanical constraints exist, as in the case study of this work, and generally the high operating temperatures and loads.

Many aspects have to be explored and tested yet. In particular, the computational problem due to the possible distortions of mesh elements, the not negligible computational time of CaUTA tool and the application of more tolerances on different case study will be object of future works where a better integration between MSC Marc® solver and Matlab® environment will be carried out.

Acknowledgements

This research did not receive any specific grant from funding agencies in the public, commercial, or not-for-profit sectors.

References

[1] Jayaraman R, Srinivasan V. Geometric tolerancing: I. Virtual boundary requirements. IBM J Res Dev 1989;33:90–104. doi:10.1147/rd.332.0090.
[2] Gupta S, Turner JU. Variational solid modeling for tolerance analysis. IEEE Comput Graph Appl 1993;13:64–74. doi:10.1109/38.210493.
[3] Desrochers A. A CAD/CAM Representation Model Applied to Tolerance Transfer Methods. J Mech Des 2003;125:14. doi:10.1115/1.1543974.
[4] Desrochers A, Rivière A. A matrix approach to the representation of tolerance zones and clearances. Int J Adv Manuf Technol 1997;13:630–6. doi:10.1007/bf01350821.
[5] Gao J, Chase KW, Magleby SP. Generalized 3-D tolerance analysis of mechanical assemblies with small kinematic adjustments. IIE Trans 1998;30:367–77. doi:10.1080/07408179808966476.
[6] Laperrière L, Lafond P. Tolerance Analysis And Synthesis Using Virtual Joints. In: van Houten F, Kals H, editors. Glob. Consistency Toler., Dordrecht: Springer Netherlands; 1999, p. 405–14. doi:10.1007/978-94-017-1705-2_41.
[7] Bourdet P, Mathieu L, Lartigue C, Ballu A. The concept of the small displacement torsor in metrology. Ser Adv Math Appl Sci 1996;40:110–122.
[8] Ghie W, Laperrière L, Desrochers A. A Unified Jacobian-Torsor Model for Analysis in Computer Aided Tolerancing. In: Gogu G, Coutellier D, Chedmail P, Ray P, editors. Recent Adv. Integr. Des. Manuf. Mech. Eng., Dordrecht: Springer Netherlands; 2003, p. 63–72. doi:10.1007/978-94-017-0161-7_7.
[9] Davidson JK, Mujezinović A, Shah JJ. A New Mathematical Model for Geometric Tolerances as Applied to Round Faces. J Mech Des 2002;124:609. doi:10.1115/1.1497362.
[10] Giordano M, Pairel E, Samper S. Mathematical representation of Tolerance Zones. In: van Houten F, Kals H, editors. Glob. Consistency Toler. Proc. 6 th CIRP Int. Semin. Comput. Toler. Univ. Twente, Enschede, Netherlands, 22--24 March, 1999, Dordrecht: Springer Netherlands; 1999, p. 177–86. doi:10.1007/978-94-017-1705-2_18.
[11] Schleich B, Anwer N, Mathieu L, Wartzack S. Skin Model Shapes: A new paradigm shift for geometric variations modelling in mechanical engineering. Comput Des 2014;50:1–15. doi:10.1016/j.cad.2014.01.001.
[12] Chang M, Gossard DC. Modeling the assembly of compliant, non-ideal parts. Comput Des 1997;29:701–8. doi:10.1016/S0010-4485(97)00017-1.
[13] Mortensen AJ. An Integrated Methodology for Statistical Tolerance Analysis of Flexible Assemblies. Brigham Young University. Department of Mechanical Engineering; 2002.
[14] Liu SC, Hu SJ. Variation Simulation for Deformable Sheet Metal Assemblies Using Finite Element Methods. J Manuf Sci Eng 1997;119:368. doi:10.1115/1.2831115.
[15] Liao X, Wang GG. Non-linear dimensional variation analysis for sheet metal assemblies by contact modeling. Finite Elem Anal Des 2007;44:34–44. doi:10.1016/j.finel.2007.08.009.
[16] Xie K, Wells L, Camelio JA, Youn BD. Variation Propagation Analysis on Compliant Assemblies Considering Contact Interaction. J Manuf Sci Eng 2007;129:934. doi:10.1115/1.2752829.
[17] Lorin S, Lindkvist L, Söderberg R, Sandboge R. Combining Variation Simulation With Thermal Expansion Simulation for Geometry Assurance. J Comput Inf Sci Eng 2013;13:31007. doi:10.1115/1.4024655.
[18] Jareteg C, Wärmefjord K, Söderberg R, Lindkvist L, Carlson J, Cromvik C, et al. Variation Simulation for Composite Parts and Assemblies Including Variation in Fiber Orientation and Thickness. Procedia CIRP 2014;23:235–40. doi:10.1016/j.procir.2014.10.069.
[19] Sellem E, Rivière A. Tolerance Analysis of Deformable Assemblies. Proc. 1998 ASME Des. Eng. Tech. Conf., Atlanta: 1998, p. 1–7.
[20] Falgarone H, Thiébaut F, Coloos J, Mathieu L. Variation Simulation During Assembly of Non-rigid Components. Realistic Assembly Simulation with ANATOLEFLEX Software. Procedia CIRP 2016;43:202–7. doi:10.1016/j.procir.2016.02.336.
[21] Anwer N, Ballu A, Mathieu L. The skin model, a comprehensive geometric model for engineering design. CIRP Ann - Manuf Technol 2013;62:143–6. doi:10.1016/j.cirp.2013.03.078.
[22] Anwer N, Schleich B, Mathieu L, Wartzack S. From solid modelling to skin model shapes: Shifting paradigms in computer-aided tolerancing. CIRP Ann - Manuf Technol 2014;63:137–40. doi:10.1016/j.cirp.2014.03.103.
[23] Zhang M, Shi Z, Mathieu L, Nabil A, Yang J. Geometric Product Specification of Gears: The GeoSpelling Perspective. Procedia CIRP 2015;27:90–6. doi:10.1016/j.procir.2015.04.049.
[24] Zhang M, Anwer N, Mathieu L, Zhao H. A Discrete Geometry Framework for Geometrical Product Specifications. Proc. 21st CIRP Des. Conf., vol. 2, 2011, p. 1–7.
[25] Franciosa P, Gerbino S, Patalano S. Simulation of variational compliant assemblies with shape errors based on morphing mesh approach. Int J Adv Manuf Technol 2011;53:47–61. doi:10.1007/s00170-010-2839-4.
[26] Zhang M, Anwer N, Stockinger A, Mathieu L, Wartzack S. Discrete shape modeling for skin model representation. Proc Inst Mech Eng Part B J Eng Manuf 2013;227:672–80. doi:10.1177/0954405412466987.
[27] Schleich B, Anwer N, Zhu Z, Qiao L, Mathieu L, Wartzack S. Comparative study on tolerance analysis approaches. Int. Symp. Robust Des., 2014.
[28] Corrado A, Polini W. Manufacturing signature in variational and vector-loop models for tolerance analysis of rigid parts. Int J Adv Manuf Technol 2017;88:2153–61. doi:10.1007/s00170-016-8947-z.
[29] Corrado A, Polini W. Manufacturing signature in jacobian and torsor models for tolerance analysis of rigid parts. Robot Comput Integr Manuf 2017;46:15–24. doi:10.1016/j.rcim.2016.11.004.
[30] Formosa F, Samper S. Modal Expression of Form Defects. In: Davidson JK, editor. Model. Comput. Aided Toler. Des. Manuf., Dordrecht: Springer Netherlands; 2007, p. 13–22. doi:10.1007/1-4020-5438-6_3.
[31] Schleich B, Wartzack S. Evaluation of geometric tolerances and generation of variational part representatives for tolerance analysis. Int J Adv Manuf Technol 2015;79:959–83. doi:10.1007/s00170-015-6886-8.
[32] Srinivasan V. Computational Metrology for the Design and Manufacture of Product Geometry: A Classification and Synthesis. J Comput Inf Sci Eng 2007;7:3. doi:10.1115/1.2424246.
[33] Yan X, Ballu A. Generation of consistent skin model shape based on FEA method. Int J Adv Manuf Technol 2017;92:789–802. doi:10.1007/s00170-017-0177-5.
[34] Schleich B, Wartzack S. Approaches for the assembly simulation of skin model shapes. Comput Des 2015;65:18–33. doi:10.1016/j.cad.2015.03.004.
[35] Schleich B, Anwer N, Mathieu L, Wartzack S. Status and Prospects of Skin Model Shapes for Geometric Variations Management. Procedia CIRP 2016;43:154–9. doi:10.1016/j.procir.2016.02.005.
[36] Garaizar OR, Qiao L, Anwer N, Mathieu L. Integration of Thermal Effects into Tolerancing Using Skin Model Shapes. Procedia CIRP 2016;43:196–201. doi:10.1016/j.procir.2016.02.079.
[37] Moroni G, Pacella M. An Approach Based on Process Signature Modeling for Roundness Evaluation of Manufactured Items. J Comput Inf Sci Eng 2008;8:21003. doi:10.1115/1.2904923.
[38] Zhao J, Gu Y, Fan W. Variable-based Ramberg–Osgood constitutive model of power spinning bushing. Trans Nonferrous Met Soc China 2015;25:3080–7. doi:10.1016/S1003-6326(15)63936-X.
[39] MSC. Marc – Volume A: Theory and User Information. 2013.