



**HAL**  
open science

## Free-state shape of aeronautical components for assembly simulation

F Gringoz, F Thiebaut, C Lartigue, B Soufflet

► **To cite this version:**

F Gringoz, F Thiebaut, C Lartigue, B Soufflet. Free-state shape of aeronautical components for assembly simulation. International Joint Conference on Mechanics, Design Engineering and Advanced Manufacturing, Jun 2018, Cartagena, Spain. hal-02005508

**HAL Id: hal-02005508**

**<https://hal.science/hal-02005508v1>**

Submitted on 4 Feb 2019

**HAL** is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

# Free-state shape of aeronautical components for assembly simulation

F. Gringoz<sup>1,2,\*</sup>, F. Thiébaud<sup>1</sup>, C. Lartigue<sup>1</sup>, B. Soufflet<sup>2</sup>

<sup>1</sup>LURPA, ENS Cachan, Univ. Paris-Sud, Université Paris-Saclay, 94235 Cachan, France

<sup>2</sup>SAFRAN Nacelles, SAFRAN Group, R&T Methods Department, Route du pont VIII, 76700 Gonfreville-L'Orcher, France

## Abstract

This paper deals with a method to extract the free-state shape of aeronautical assembly components from part measurements independently of the assembly configuration of use. Knowledge of the free-state shapes enables to assess the geometrical conformity of an assembly through the assembly simulation using Finite Element Method (FEM). The component is measured, using optical means in a given configuration for which the set-up is well-known. A coarse cleaning is thus applied on the measured data to obtain manipulative data in the CAD model frame. Meanwhile, displacements due to the measuring set-up and to gravity are evaluated from the nominal geometry using FEM. Finally, the free-state shape, as a finite element mesh, is achieved by moving the nominal mesh nodes by a distance equal to the measured defect minus the evaluated displacement. The approach is applied to an aeronautical component.

**Keywords:** Geometrical deviations; Flexible parts; Optical scanning; Free-state shape.

## 1. Introduction

Within the context of aeronautics assembly structures, the geometry of an assembly is defined through the geometry of its components, described in their nominal configuration, i.e without form deviations and in their theoretical relative position. In practice, the geometry of manufactured parts differs from nominal geometry due to manufacturing process variations. Because of these imperfections, the inspection of assembly geometry is a major issue in industry.

During the assembly process, the geometrical conformity of components is verified by measuring the key characteristics (KC) that must respect given specifications to answer the functional requirements of the assembly. The classical method is to compare the actual component geometry to its nominal model, most often its CAD model [1, 2]. The actual geometry is generally obtained by surface meas-

urements, and the measured data are compared to the CAD model to determine if the manufactured geometry lies within the tolerance zone. When geometrical deviations out of the admissible tolerance interval are detected on the component, adjustment operations are carried out in order to get closer to the nominal geometry. This procedure turns out to be very expensive, especially when it comes to composite parts. However, in some cases, the assembly of non-conform components could result in assembly conformity. Moreover, an average assembly geometry, differing slightly from the nominal geometry could be just as functional as the geometry initially chosen. Therefore, analyzing the impact of the geometrical deviations of components on the geometrical functionality of the assembly is a challenging issue.

Difficulties arise from various issues problems. First, the assembly process generally includes many assembly stages. The output geometry of one stage is the input of the next stage, and thus, the assembly geometry is modified from station to station [3, 4] due to the succession of assembly operations [5]. In addition, the flexibility of aeronautical components makes their inspection arduous because of the deformations induced by the gravity load and by fixturing conditions [6]. The inspection of such components is generally performed when components are in the assembly configuration, at different stages of the assembly process. However, the assembly conditions are not necessarily the same as the conditions of use. R. Ascione and W. Polini [6] propose a method to control component geometry using coordinate measuring machines (CMM) in their conditions of use. These conditions are reproduced thanks to modular equipment. Another approach, described by P. Franciosa et al. [4], consists of coupling a prediction of the defect patterns of a component with a measurement step to find the real causes of these defects at different stages of the assembly process. These methods require the configuration of use to be reproduced, which turns out to be expensive in the context of large aeronautics components.

Some studies propose to use the Finite Element Method (FEM) for assembly inspection. H. Radvar-Esfahlan and S.-A. Tahan [7] use FEM simulation that take into account the boundary conditions associated to the measuring configuration of the component geometry. This allows authors to compare the nominal geometry to the measured geometry in the same conditions, even if they are different from the conditions of use. A. Jaramillo et al. [8] developed an approach which consists in simulating the required deformations to match the reference points before comparing the partial view of the actual geometry restricted to regions that need to be inspected to the CAD model. To evaluate the assembly components geometry from scanned assembly geometries in the context of repair analysis, A. Yu et al. [9] proposed to assess component geometries through the simulation of a virtual assembly process.

I. Gentilini and K. Shimada [10] focused on FEM to predict the assembly geometry from measured component geometries through the simulation of the assembly process. Authors note an error between predicted and actual assembly geometries which is probably due to the variations of measuring conditions, which modify the loading conditions (including gravity).

To perform accurate assembly simulations whatever the configuration, the intrinsic geometry of components, called the free-state shape, must be known. The free-state is the shape a component should have in absence of loads [11, 12]. The free-state turns out to be difficult to identify when flexible parts are concerned, which is the case for aeronautical assembly structures, gravity loads and part fixturing indeed induce part deformations.

In this context, a method using finite element simulation to evaluate the geometrical conformity of assemblies is proposed, based on the free-state of components. The geometry of the components is extracted by measuring the components. Then, a FEM enables assembly simulation which is used to assess the assembly conformity.

This paper focuses on the step of geometry extraction from part measurements regardless of the assembly configuration of use. The originality of this method is to extract the free-state geometry of the component based on FEM simulations. The approach, summarized in Figure 1, is applied to the aeronautical component of an assembly defined by its CAD model.

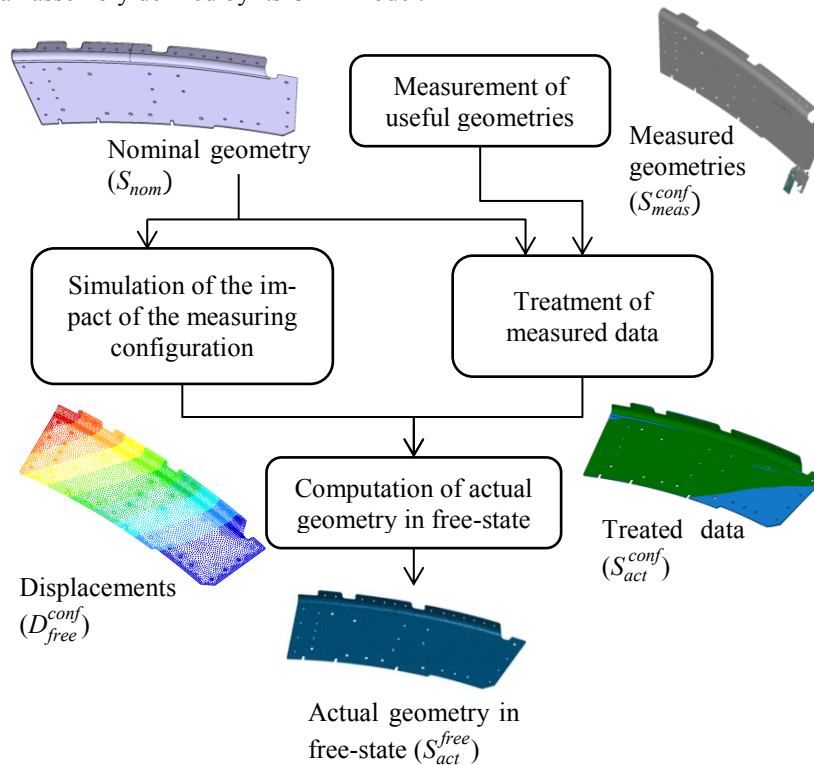


Figure 1 : Method to obtain the actual free-state geometry

Starting from the measurement of useful geometries only, a step of measured data treatment is performed in order to obtain a point cloud that is representative of the geometry in the measuring configuration. In parallel, a simulation of com-

ponent deformation under the gravity and loads (associated to the measuring configuration) is carried out using FEM. Finally, the actual free-state is obtained by comparing both geometries.

This paper is organized as follows. In Section 2, we detail our method to extract the actual free-state shape of a component. A case study is then reported in Section 3. Section 4 presents the conclusion and perspectives of this work.

## 2. Obtaining the free-state shape of a component

The proposed method starts by the acquisition of the component's actual shape in a given configuration, denoted  $S_{act}^{conf}$ .

$$S_{act}^{conf} = S_{act}^{free} + D_{free}^{conf} \quad (1)$$

This shape is the superposition (equation (1)) of the shape that the component would have in its free-state  $S_{act}^{free}$  (free from any loads), and of the deformations induced by the loads associated to the gravity and the measuring configuration. These deformations  $D_{free}^{conf}$ , can be computed thanks to FE simulations. The following sections detail how the different terms of the equation are obtained.

### 2.1. Acquisition of the component's actual shape

The acquisition of the component's actual shape gives the numerical representation of the actual component geometry. This is performed through two stages: measurement of the geometry and processing of the measured data. The first stage yields data that are not directly exploitable, and the second stage makes them usable, thus leading to  $S_{act}^{conf}$ .

**Geometry measurement:** For the measurement, the stereovision sensor, ATOS Core, mounted on a tripod, is used (Figure 2).

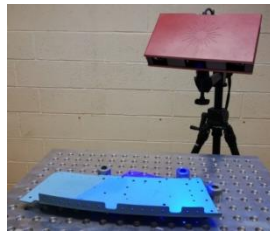


Figure 2 : Stereovision sensor ATOS Core

The measuring set-up plays a major role, as it may ensure part accessibility, but it also defines the boundary conditions that will be used in FE simulations. Therefore, once the measuring set-up is defined, it must be scanned so that its position is completely defined in the reference frame. Then, the component is set in position to be scanned in the same frame. Measuring these two elements in the same frame enables to evaluate the position of the contact points between the measuring set-up and the component. At the end of the measuring stage, the acquired data, expressed in a STL format, represent the measured shape in the given configuration,  $S_{\text{meas}}^{\text{conf}}$ . This measured shape is a raw point cloud, which is generally dense, non-homogeneous and contains unnecessary elements. Data processing is thus necessary to obtain the geometry of the component in the same frame as the CAD model frame.

**Measured data treatment:** To compute the actual free-state shape of the component, both its actual shape in the given measuring configuration  $S_{\text{act}}^{\text{conf}}$ , and the configuration, must be known. Data treatment thus consists in two main steps. At first, points that are not representative of the shape are removed from the measured point cloud. The measured shape is thus registered to its CAD model. This registration is carried out by matching the common geometrical elements of both shapes. The position of the contact points between the component and the measuring set-up has to be identified for FE simulation. As the measuring set-up is measured in the same frame as the component, the registration previously performed permits to detect the position of the supporting points directly in the CAD frame.

## ***2.2. Displacement computation by FE simulation***

The measuring configuration induces deformations due to the gravity load and the component's positioning on its measuring set-up. These deformations are evaluated thanks to FE simulation. First, the simplification and the sampling of the CAD model are carried out to obtain a finite element mesh. Due to the thin shape of aeronautic components, shell elements are used for the meshing. Then, the contact between the part surfaces and the set-up are expressed as boundary conditions, whereas gravity is considered as an oriented acceleration. To achieve the simulation, the mechanical behaviour of the component is assumed to be elastic linear. The result of this simulation, carried out in the finite element module of CATIA V5©, is a displacement field,  $D_{\text{free}}^{\text{conf}}$ , which represents the displacement of each node of the mesh in the CAD frame.

### 2.3. Free-state shape computation

The free-state shape computation is expressed by equation (1). The first step gives the actual shape in the measuring configuration,  $S_{\text{act}}^{\text{conf}}$ . The second step leads to the displacement field due to the impact of the measuring configuration,  $D_{\text{free}}^{\text{conf}}$  calculated using FE simulation. However, although data are expressed in the same frame, both point clouds are not homogeneous; the measured point cloud is denser than the CAD mesh. In order to homogenize both point clouds, to each node of the nominal mesh,  $p_i$ , a corresponding mean point  $m_i$ , is computed on the point cloud considering a small neighborhood defined by a cylinder at the vicinity of the node. Furthermore, as the free-state will be the basis for FE simulations [11], it is represented by a finite element mesh.

For this purpose, our approach consists in moving  $p_i$  along its normal vector  $\vec{n}_i$  by a distance equal to the projection of the distance between the node  $p_i$  and its mean point  $m_i$  onto  $\vec{n}_i$ , minus the calculated displacement at the node,  $D_i$ , with  $D_i$  the component of  $D_{\text{free}}^{\text{conf}}$  for each node of the mesh. The method is summarized as follows (see Figure 3):

- The normal vector  $\vec{n}_i$  of each node  $p_i$  of the mesh is computed as the mean of the normal vector of the mesh elements sharing the node  $p_i$ .
- A cylinder, whose axis corresponds to the normal vector, and whose radius equals to 1mm is built.
- The mean point  $m_i$  of the points included in this cylinder is computed.
- The deviation between each node  $p_i$  and its corresponding mean point  $m_i$  is projected onto the normal vector  $\vec{n}_i$ , giving  $e_i$ .
- The displacement  $D_i$  of the node  $p_i$  is projected onto  $\vec{n}_i$ , giving  $D_{ip}$ .
- The deviation  $d_i = e_i - D_{ip}$  is calculated.
- The node is moved from its original position along its normal vector by a value equal to  $d_i$ .

The final result is a finite element mesh which is consistent with finite element software. This mesh is a representation of the actual geometry in the free-state, which enables to simulate various positioning configurations. Our approach is illustrated by an example in the next section.

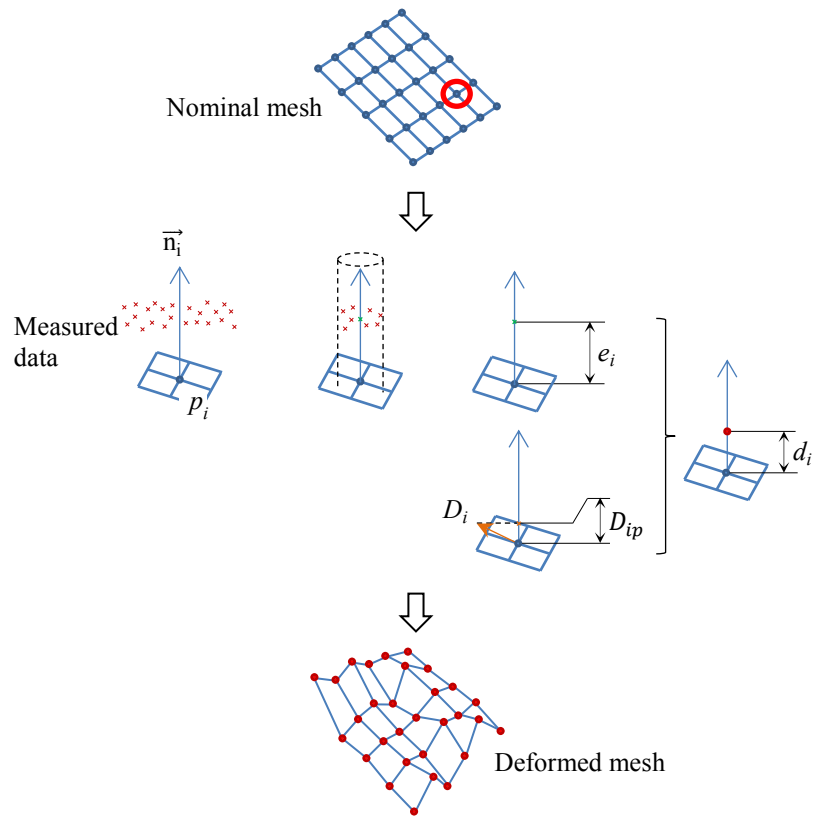


Figure 3 : Deformation of the nominal mesh

### 3. Case study: a flexible aeronautical component

In this example, the component used is a part of the forward frame of the thrust reverser of the airplane engine nacelle, the front panel (Figure 4). This component, made of composite material, is very thin, which makes it flexible. Currently, the whole geometry of this component is inspected before assembly, and the geometry of its external skin is inspected after assembly. Predicting the component geometry would be useful to improve the assembly process. In this direction, the use of the actual free-state geometry enables to get rid of the measurement configuration, and thus of the gravity, and is a good support for FE simulations.



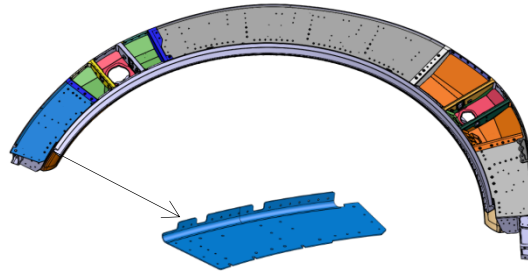


Figure 4 : The Front Panel, a part of the forward frame

The first step of the method is to measure the actual geometry of the component. One face only is measured, as the nominal mesh is a 2D mesh. The component is positioned on an equally geometry constrained set-up, and both the component and the set-up are measured in the same frame (see Figure 5). The coarse treatment is then applied to the 750 000 measured points. All the points that correspond to the set-up are manually removed from the measured data (Figure 5). Finally, the measured data are repositioned in the CAD frame. This alignment is achieved by matching common geometrical elements to both the measured and the nominal geometries. For the case study, the geometrical elements correspond to the front panel which positions holes on the forward frame during assembly. This stage gives an exploitable point cloud of 600 000 points expressed in the CAD frame.

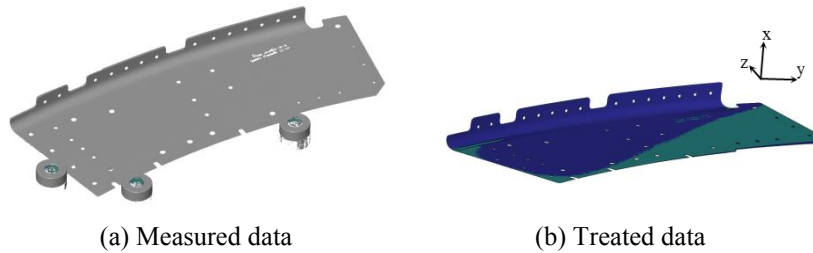


Figure 5 : Acquisition of the component actual shape

In parallel, the displacements cause by gravity and by measuring set-up are computed thanks to FE simulation. The CAD model is simplified as a surface model to build a 2D mesh composed of shell elements with 11503 elements and 6365 nodes. The contact points are identified from the measured data after their registration on the CAD model. At these points, boundary conditions impose that the displacements along the normal at the contact points be null. There are no additional loads to implement thanks to the use of an equally geometry constrained set-up. The gravity is implemented as an oriented acceleration along the  $\vec{x}$  axis (see Figure 5 (b)). The simulation yields the displacements of the mesh nodes, which are here relatively small compared to part form defects (see Figure 6).

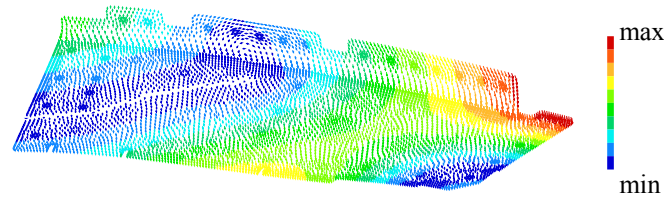


Figure 6 : Displacements

Finally, the free-state shape can be computed according to the method detailed in section 2.3. The mean points in the neighborhood of each node of the finite element mesh are computed, and then projected onto the normal vector at the node. The evaluated displacements at the nodes are removed from the projected distance between the node and its mean point. Nodes are then moved of the computed distance along its normal. The result is a finite element mesh, which is representative of the actual geometry of the front panel in the free-state, with geometrical deviations (see Figure 7).

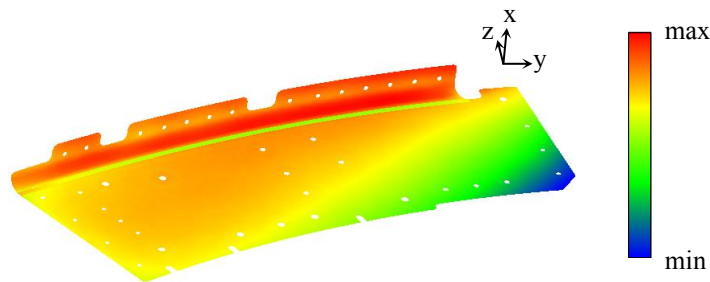


Figure 7 : The actual free-state shape of the front panel

The displacements and the actual free-state shape are shown (see Figure 6 and Figure 7) to have an idea of the expected results. However, the values of these displacements and the geometrical deviations cannot be communicated for reasons of confidentiality.

#### 4. Conclusion

In this paper, we proposed a method to extract the free-state shape of a component independently of the assembly configuration of use. This method relies on the component geometry measurements. For this purpose, the component is positioned on a set-up that is entirely defined during the measuring stage. When positioned on its set-up, the component is subjected to deformations cause by both the gravity load and the measuring set-up. These deformations, evaluated using FE simulations, are removed from the measured data to obtain the free-state shape of

the component. Finally, the nominal mesh is deformed to obtain the free-state shape of the component as a finite element mesh. Applied to an aeronautical component, the method provides a finite element mesh which is representative of the component's free-state shape that can be used to evaluate the geometry of the component in different configurations or for assembly simulations. If the whole approach seems relevant, each step of the method now has to be assessed. Specific attention will be paid in future work to the evaluation of the mechanical model, by using more than one configuration.

## 5. References

- [1] Bispo, E. M., Fisher, R. B.: Free-form Surface Matching for Surface Inspection. (1994).
- [2] Samper, S. et al.: Modeling of 2D and 3D assemblies taking into account form errors of plane surfaces. *Journal of Computing and Information Science in Engineering* 9, 041005-1 (2009).
- [3] Hu, S. J., Camelio, J.: Modeling and Control of Compliant Assembly Systems. *CIRP Annals-Manufacturing Technology* 55, 19-22 (2006).
- [4] Franciosa, P. et al.: Rapid Response Diagnosis of Multi-stage Assembly Process with Compliant non-Ideal Parts using Self-evolving Measurement System. *Procedia CIRP* 60, 38-43 (2017).
- [5] Liu, G., et al.: Study on analysis and prediction of riveting assembly variation of aircraft fuselage panel. *International Journal of Advanced Manufacturing Technology* 75, 991–1003 (2014).
- [6] Ascione, R., Polini, W.: Measurement of nonrigid freeform surfaces by coordinate measuring machine. *The International Journal of Advanced Manufacturing Technology* 51, 1055-1067 (2010).
- [7] Radvar-Esfahlan, H., Tahan, S.-A.: Non-rigid geometric metrology using generalized numerical inspection fixtures. *Precision Engineering* 36, 1-9 (2012).
- [8] Jaramillo, A. et al.: Fast dimensional inspection of deformable parts from partial views. *Computers in Industry* 64, 1076–1081 (2013).
- [9] Yu, A. et al.: Geometric Design Model and Object Scanning Mode Based Virtual Assembly and Repair Analysis. *Procedia CIRP* 44, 144-150 (2016).
- [10] Gentilini, I., Shimada, K.: Predicting and evaluated the post-assembly shape of thin-walled components via 3D laser digitization and FEA simulation of the assembly process. *Computer -Aided Design* 43, 316-328 (2011).
- [11] Thiébaud, F., et al.: Evaluation of the shape deviation of non rigid parts from optical measurements. *The International Journal of Advanced Manufacturing Technology* 84, 9-12 (2016).
- [12] Lacroix, C.: Caractérisation géométrique des assemblages flexibles par la mesure. Phd Thesis. École normale supérieure de Cachan, Cachan, France, 2015.