Methodology for the analysis of tolerances in the assembly process of a wing torsion box through FE simulations

A. Chiminelli[1], F. Martín de la Escalera[2], R. Breto[1], E. Duvivier[1], J.L. Núnez[1], L. Lizarduy[3], M.A.Jimenez[1].

[1] Instituto Tecnológico de Aragón – ITAINNOVA, María de Luna 7-8, 50014, Zaragoza, Spain.
[2] Aernnova Engineering Solutions Ibérica, Avda. Manoteras 20, 28050, Madrid, Spain.
[3] Aernnova Manufacturing Engineering, Portal De Gamarra 40, 01013, Vitoria-Gasteiz, Spain.

achiminelli@itainnova.es

Abstract: *This work presents a methodology for the analysis of deviations that might be produced during the assembly process of aircraft components using the finite element method as calculation tool. The methodology allows determining the effect that different deviations in the constituent parts may have on the final tolerances of an assembly considering the influence of the joining techniques and the different operations that are usually carried out during such processes. This permits to evaluate if certain deviations in some parts would maintain the final assembly within the specified tolerances or, conversely, may accumulate/propagate negatively generating a non conformity. In addition, the simulations performed allow analysing the possible effect of deviations coming from the assembly tooling or different fastening sequences. In this sense, the work performed demonstrates that the finite element method can be used effectively to improve the processes commonly used in the manufacturing of components constituted by several parts and with tight tolerances.*

This work is part of the activities performed within the project "284961 CLEAN SKY, Simulation of the assembly tolerances for composite aircraft structures – SATCAS", led by ITAINNOVA and performed under the supervision of Aernnova Engineering Division as Topic Manager. SATCAS is a sub-project associated to the activity of Assembly Simulation defined in the work-package WP 3.2.1.2 of BLADE – SFWA - Clean Sky programme, led by Airbus.

Keywords: *Simulation Methodology, Assembly Processes, Tolerance Analysis, Deviations, Aircraft Structures.*

1. Introduction

Independently of the manufacturing processes and their sophistication, parts always present deviations and imperfections. The wear of the tools and moulds, misalignments in the fixation systems and jigs, errors that the operators may introduce, deformations caused by the own fabrication processes and many other factors; will inevitably generate defects and variations in the components. In systems constituted by many parts, the assembly processes acquire a special relevance, affecting the way on which the deviations might accumulate/propagate.

For the aeronautic and automotive industries, the study of accumulative tolerances has gained a significant relevance since its influence in the manufacturing costs is generally important [1]. These studies allow predicting deviations in assemblies, understand how they are produced, identify the main sources and, finally, obtaining recommendations to reduce the manufacturing non-conformities.

The conventional tolerances analysis techniques focus on the evaluation of the impact that geometric and/or dimensional deviations in the constituent parts may have in the dimensions of the resultant ensembles. In most of these methodologies the parts are considered as rigid bodies (i.e. the constituent components do not deform during the assembly processes).In many applications this hypothesis is inadequate because:

-The assemblies are strongly affected by the part compliances (the hypothesis of parts as rigid bodies is not acceptable because they are significantly deformed during the assembly).

-The assemblies have such tight tolerance requirements that, despite the fact that the parts behave almost rigidly, the deformations generated during the assembly processes cannot be dismissed.

In those cases, issues come into play such as the interaction between parts as a consequence of the deviations generated by the tools or the assembly jigs, the distortions generated in the joints, or the spring-back effects in the parts. From a methodological point of view, the incorporation of the part compliances in the tolerance analysis of assemblies is usually carried out through the integration of mechanical/structural calculation techniques with the methods of tolerances accumulation previously described [2-4].

The methodology presented in this work is based on the Finite Element Method (FEM) as calculation tool to analyse the accumulation and propagation of deviations considering the parts compliance, necessary taking into account the tight tolerances that apply for the NLF (Natural Laminar Flow) Wings object of study of SATCAS and the flexibility and size of these assemblies.

The numerical methodology developed covers, among other, the following aspects:

- Definition of FEM strategies for the simulation of the assembly process.

- Development of an application for the treatment of results.

The method has been applied to the analysis of a NLF Wing Box defined within the framework of the BLADE project previously mentioned.

Taking into account the complexity of the processes used in aeronautics, their simulation becomes a major challenge. In this sense, it must be considered that the deformations in the final ensembles can be caused by the deviations in the constituent parts, deviations in the assembly jigs or by the assembly operations itself (for example deformations introduced by the temporary fasteners or the bolts). All these possible sources of deviations should be taken into account in the numerical simulations. The FEM strategy proposed in SATCAS aims to cover all these aspects using different techniques to simulate the operations and/or steps that may have an effect in the final deviations of the resultant component.

Finally, the tolerances analysis performed in this work focuses on the waviness that might appear in the Upper Cover surface, which can produce the lost of the laminar flow. This analysis of waviness requires a relatively complex treatment of the FE results that, as it will be explained later, has been implemented in a computer code and constitute part of the methodology proposed.

This work has been developed as part of the project SATCAS - Simulation of the assembly tolerances for composite aircraft structures. The main objective of the project was the development of a methodology for the analysis of assembly tolerances considering the propagation/accumulation of deviations coming from multiple sources (parts, assembly jigs and deviations associated to the assembly process) and the parts compliance. From the point of view of its application, the method is useful to: determine the key driving parameters in the processes, analyse the possible influence of the joining techniques (type of bolts, nuts, washers...), evaluate different assembly strategies, support the decision regarding the fixation sequences and determine how the components deviations can affect to the final assembly tolerances.

Within the methodology developed, this paper focuses in the simulation strategies proposed for the assembly processes and in the application developed for the treatment of results. In order to describe it and to show its operation, the results obtained from one of the cases analysed is presented: a worst case for a deviation in the cradles of the assembly tool.

2. Numerical simulation of the assembly process

The FE simulation strategies have been defined taking into account the assembly process proposed for the NLF Wings. Among others, the following aspects have been identified as critical in these processes:

- Simulation of initial fixations through temporary fasteners.

- Simulation of the bolts/rivets (including in-plane strains produced locally that, accumulated, might generate global deformations).

In assemblies where liquid and/or solid shims are used to compensate the clearances that usually appear between the constituent parts, the initial fixation through temporary fasteners becomes an important assembly phase in terms of tolerances, since the deformations that are produced by them are not avoided by the shims. Within the methodology proposed, the fasteners are represented through two simple connector elements and an additional set of 'shell' elements that represent the fastener head (surface for contact, Figure 1). The first connector introduces the load (with a low stiffness to guarantee that the entire load goes to fix the parts together) and the second one is used to finally introduce the axial stiffness of the main rod of the fastener. The force can be controlled through a load that operates in the main axis of the first connector. This simulation strategy allows considering that the position of the bolts may change during the assembly process (since the parts are being deformed and the drilling of the holes could be executed in the course of the process).

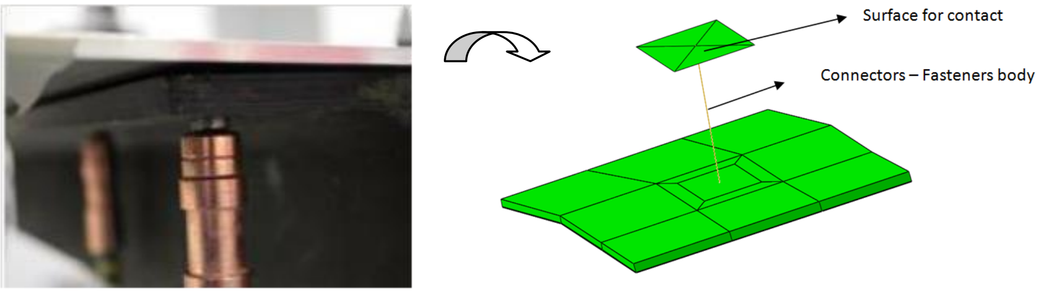

Figure 1. Representation of the temporary fasteners in the FE models.

The forces that are introduced by the temporary fasteners were determined experimentally, using a load cell as it is shown in figure 2.

Since hundreds of temporary fasteners might be required in a large aeronautical ensemble, a script has been developed by ITA that allows to introduce the virtual fasteners (connectors, shells, etc) automatically defining the positions of the pilot holes.

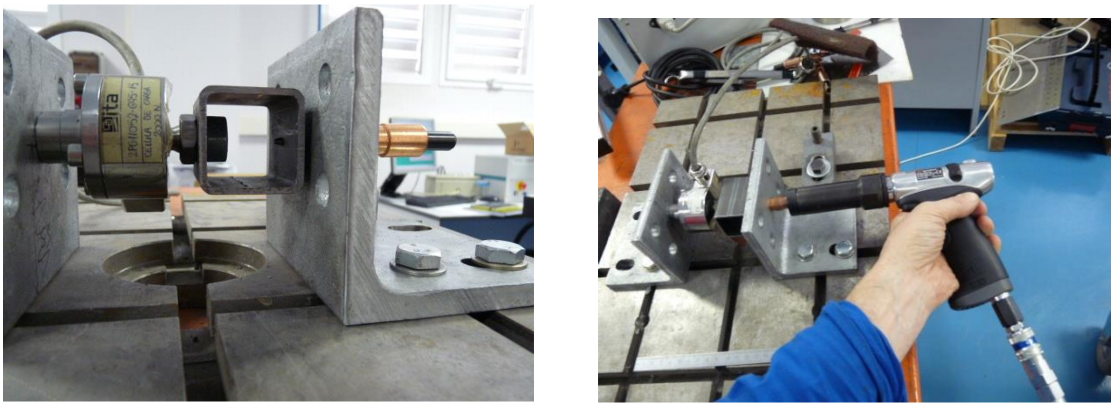

Figure 2. Measurement of the forces introduced by the temporary fasteners.

In relation with the bolts, they have been introduced in the simulations through:

- Connector elements, which allow to adequately reproducing the joints stiffness.

- A local field of in-plane strains, that is introduced to take into account the deformations that are generated locally in each bolt and that may accumulate to generate further ‘stretching’ on the parts.

The definition of the connector elements in the models vary depending on the type of bolt/nut to be considered. The adjustment of their stiffness has been performed through a correlation based on a detailed numerical-experimental study carried out considering different type of joints (Figure 3 and 4).

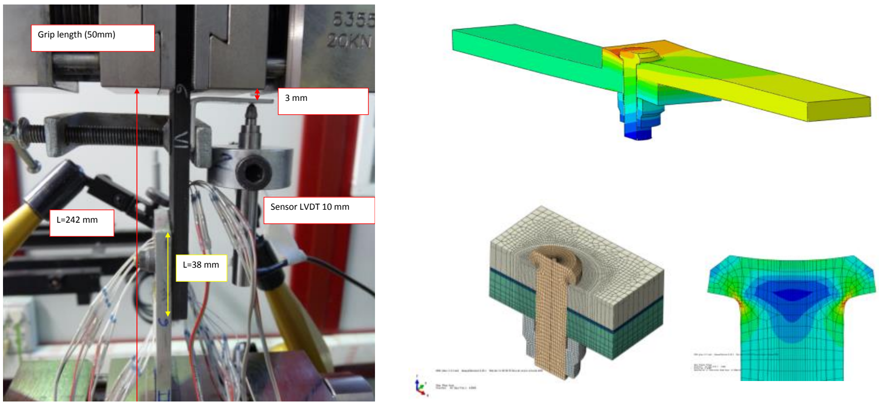

Figure 3. Detailed numerical-experimental study of bolted joints.

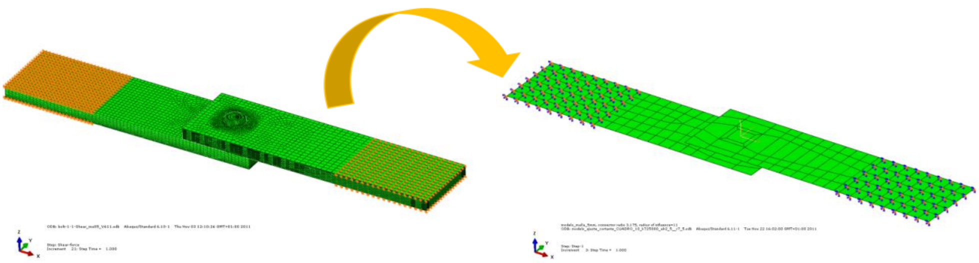

Figure 4. Detailed FE models vs. simplified FE models.

The approach proposed for the introduction of the in-plane strain fields that are generated locally in the bolted joints is based in the introduction of fictitious thermal expansions in the materials (Figure 5). Through them, and imposing adequate changes of temperature, a thermal stretching can be generated to simulate the mentioned phenomenon (depending again on the type of bolt, diameter, nut, etc).

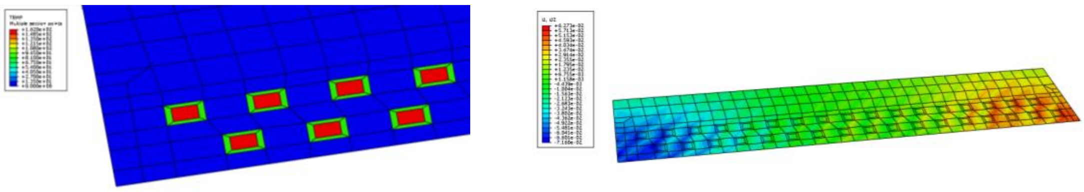

Figure 5. Strategy proposed for the introduction of the in-plane strains. Temperatures (left) and strains (right).

The thermal expansion coefficients and the jumps of temperature are adjusted depending on the substrates thicknesses, the materials involved and the type of bolts used. The radial strains has been obtained from simple 3D-solid simulations performed for the different materials/thicknesses and considering the preload introduced by the bolts (Figure 6.a.). The values of the preloads were obtained experimentally in a previous study performed by ITA. Then, the jump of temperature to obtain an equivalent radial strain (adjusted with the average value of the strains through the thickness obtained from the 3D simulations) has been determined by 3D-shell simulations considering the mesh shape (although with a higher refinement) defined for the FE models of the assemblies in the zones of the overlaps (Figure 6.b).

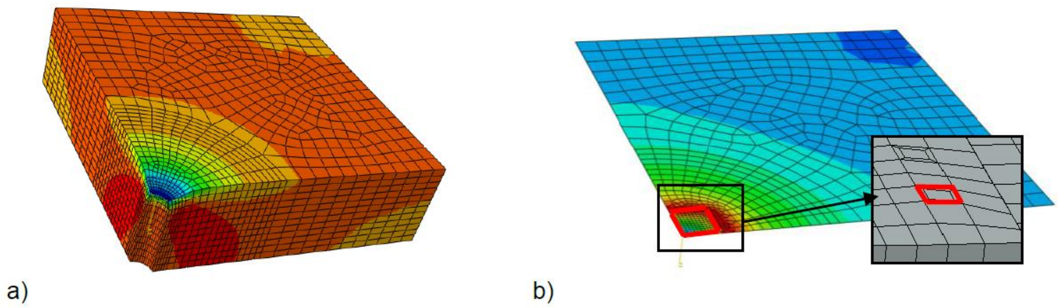

Figure 6. FE models used to adjust the thermal expansion coefficients and the jumps of temperature for the introduction of the in-plane strains.

It is worthy of mention that the strategies presented are focused on FE models of large ensembles, for simulations to determine/predict the deformations and the ‘waviness’ that might be produced during the assembly processes in a global sense. A detailed analysis of the deformations that might be generated locally near the bolts requires FE models as the ones shown in figure 3, in a different scale than the ones covered by the approach presented. However, the global deformations that can be produced by an accumulation of the in-plane local strains that are introduced by the bolts should be covered by the global FE analyses in some way. The strategy proposed is presented as a feasible approach to cover this particular source of deviations. In terms of meshes, the FE strategies should be suitable to be applied to models based mainly on ‘shell’ elements of reasonable size (in order to maintain the computational cost of the simulations within practical limits). This constitutes an additional challenge for the FE strategies proposed.

As for the temporary fasteners, the introduction of the connectors for the bolts and the definition of the temperatures and materials properties required for the simulation of the in-plane strains (through a proper definition of different sets of nodes and elements) have been simplified through a script developed by ITA specifically with these purposes.

The type of FE calculations proposed for the assembly simulations are lineal-elastic and without geometric or material nonlinearities. This assumption is based on two considerations:

- The deviations that will be analysed are in all cases small enough to guarantee that "small - displacement" hypothesis applies.

- The assembly process will never produce plastic deformations or damage in the parts.

Taking into account this, the main source of non-linearities in the simulations are the contacts/interactions between the constituent parts/components (including the heads of the temporary fasteners).

3. Treatment of results

In addition to the FE models itself, since as it has been mentioned the study is focused in the analysis of the deviations and 'waviness' that might be produced in one of the surfaces of the wings, a set of scripts/tools were needed to perform the analysis of results. These tools were integrated in a code and constitutes part of the methodology developed.

The waviness is evaluated mainly through the 'b/a' ratio, with 'b' and 'a' as defined in the following figure (measured against the nominal surface).

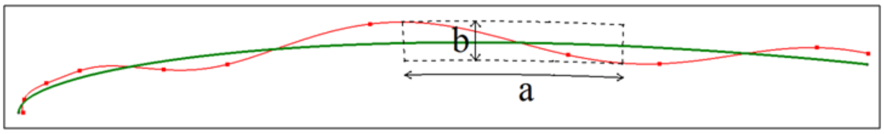

Figure 7. Parameters for waviness calculation.

Basically, the code/program developed performs the following tasks:

1.- Reading the FE results, obtaining the displacements of each node for the deformed configuration (after each FE simulation, with a certain scenario of deviations).

2.- Execution of a best-fit between the nodes of the un-deformed configuration and those of the deformed one.

3.- Reading of the original CAD surfaces (in order to be able to represent and manipulate them).

4.- Projection of all the deformed nodes on to the CAD surfaces.

5.- Sectioning algorithm. The zone of interest is divided in several chord-wise or span-wise sections where the profile and the b/a ratio are going to be calculated.

6.- Waviness calculation in the different sections.

The interface of the program developed is shown below.

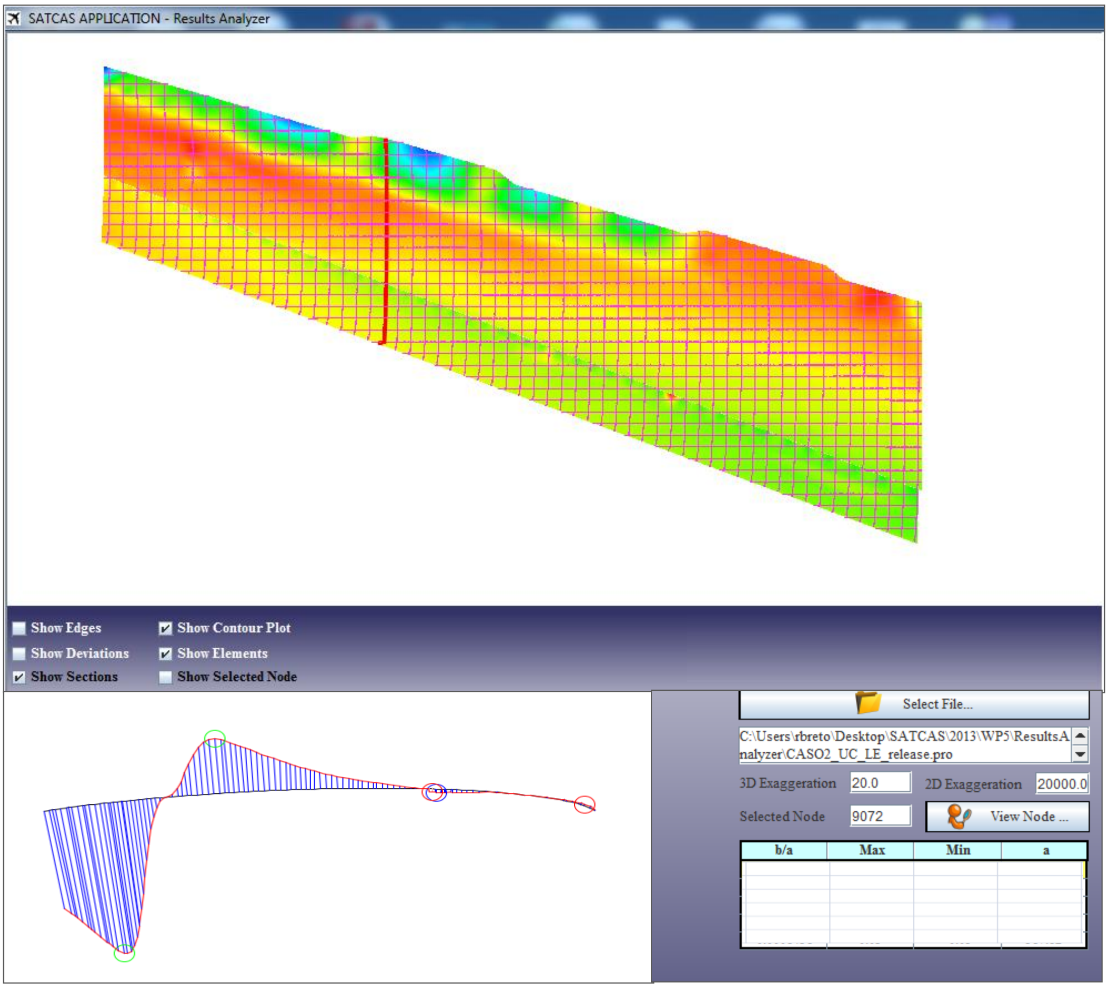

Figure 8. Interface of the code/program developed.

An example of the application of this procedure for the treatment of results is presented in the next section.

4. Application to an aircraft torsion box

As it has been previously introduced, the methodology has been applied to the analysis of the assembly of the Torsion Box of a NLF Wing. Since the assembly jigs can be an additional source of deviations, the FEM strategy defined also contemplates the simulation of these tools and the systems used for the fixation/positioning of the parts (cradles, vacuum vents/cups, etc).

Then, the FE model developed is composed by the following main parts:

- The assembly jig cradles for the Upper Cover and the Leading Edge, with a set of connector elements representing the vacuum vents/cups.

- The assembly jigs supports for the positioning/fixation of the Front Spar and the Rear Spar, with connectors to represent the fixation elements.

- Connectors for the positioning and guiding of the rest of parts.

- The constituent parts: Upper Cover, Leading Edge, Front Spar, Rear Spar, Ribs and the Lower Covers (Front, Central and Rear).

- The connectors and the rest of elements used to represent the bolts, the temporary fasteners and other fittings.

- The elements used to simulate the shim and sealant.

The parts mentioned are shown in the next figures (Figure 9):

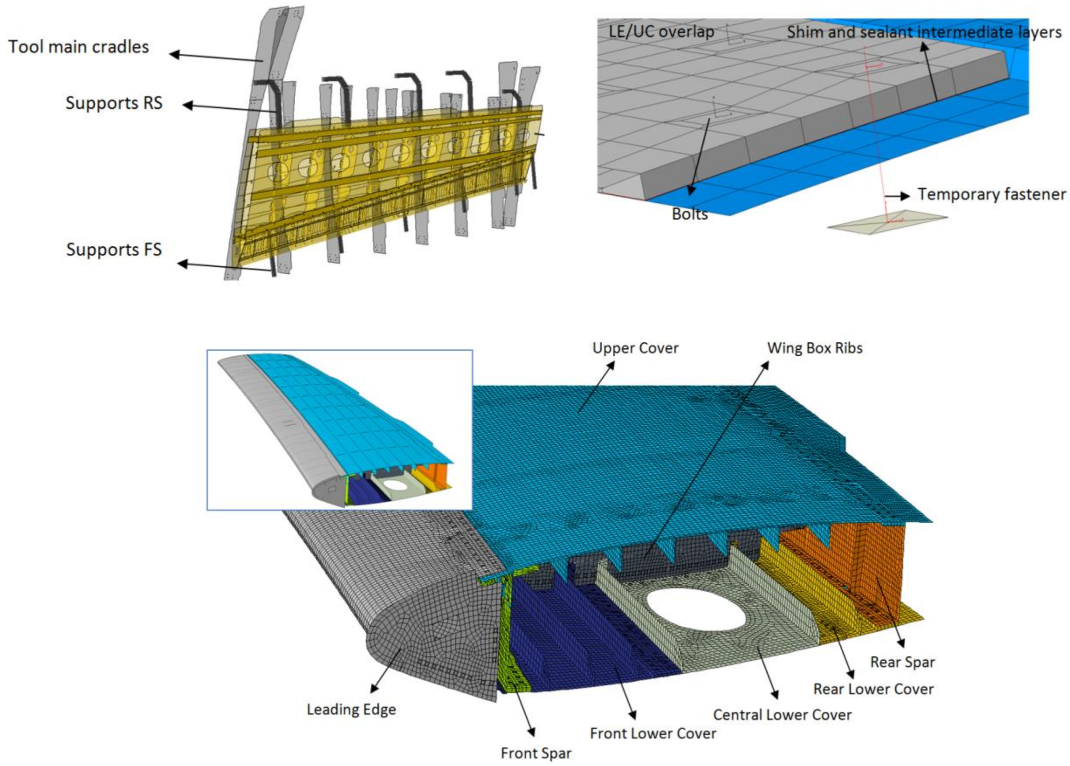

Figure 9. Main parts/elements constituting the FE model for the RH Wing.

The full model developed is composed by 409243 elements and 341034 nodes. The materials involved in the assembly have been defined as linear-elastic, isotropic for the aluminium and the steel components and orthotropic for the Upper Cover that has been designed with a carbon fibre – epoxy resin composite. The total amount of bolts in the model is 3113 and the amount of temporary fasteners is 433. Their distribution is shown in the next figure.

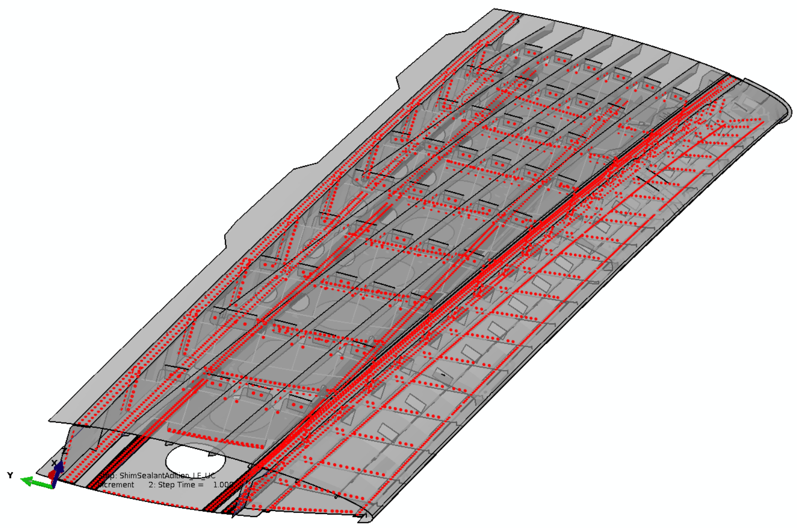

Figure 10. Elements used for the temporary fasteners & bolts in the Wing Torsion Box model.

The exact assembly sequence defined for the ensemble is introduced in the simulations through different calculation steps. For the base assembly process, more than 600 steps are required.

The methodology has been applied to diverse cases: different deviations scenarios in the constituent parts (for the cumulative tolerances analysis), deviations in the assembly tool, alternative fastening sequences, etc. In this paper, only one of these cases is presented.

The FE model has been prepared using Abaqus/Standard v6.12. The simulations have been performed using the same software.

4.1 Analysis of a deviation in the cradles of the tooling

The case presented is a deviation of +0.1 mm and -0.1 mm in the profiles of two subsequent cradles of the assembly tool. These deviations are shown schematically in the next figure. The figure also shows the results obtained from the FEM simulation of the process and the field of deviations after the treatment of results with a detail of one section (the one located in the zone where the deviations were introduced).

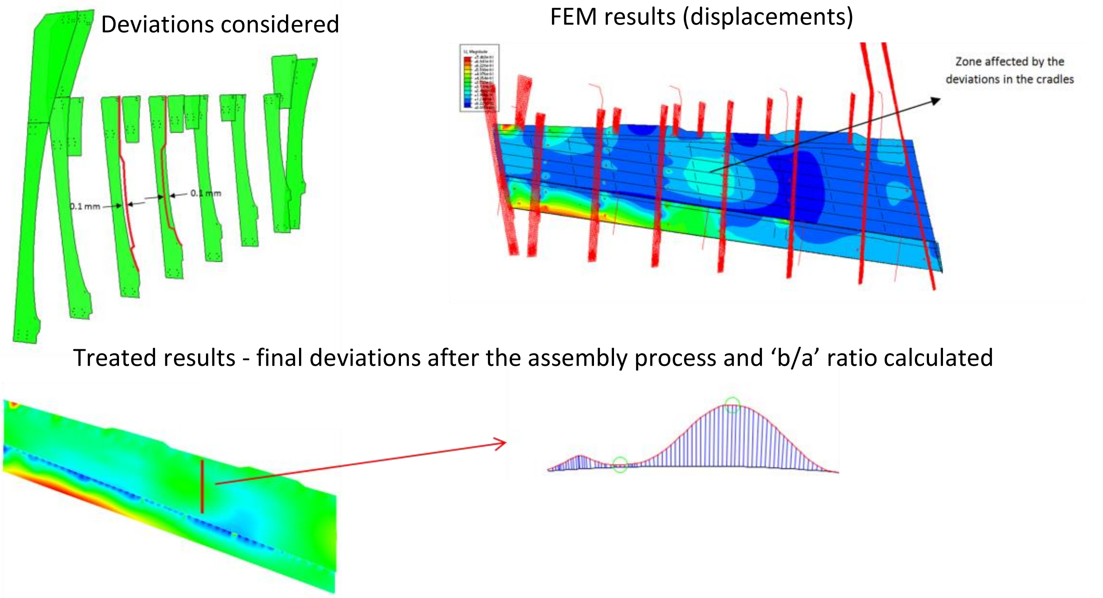

Figure 11. Case analysed. Deviations considered and results.

The main conclusions that have been obtained from this specific analysis are:

- The deformation obtained in the section in the zone affected by the deviations considered do not generate a waviness out of the acceptability range. Then, these deviations in the tooling are acceptable.

- Considering the maximum vacuum level for the suction cups used to fix/support the Upper Cover and the Leading Edge, a significant initial deformation might be introduced in both parts. This is produced by the forces done by the vents/cups and by the fact that there is a zone (corresponding to the zone in red in figure 11) where there is a nominal gap between the Leading Edge and the cradles, allowing the parts to deform freely. Moreover, these distortions are maintained throughout the entire process and their effects do not disappear after the assembly (once the suction cups fixations are released), since once the parts are assembled, they cannot recover freely (elastic recovery). In this sense, a redesign of the fixation strategy in these zones is recommended in order to reduce these deformations.

5. Conclusions

A methodology for the study of aeronautic assembly processes in terms of deviations and tolerances has been introduced. The methodology, based on the finite element method as calculation tool to take into account the parts compliances, has been applied to the assembly of an aircraft Wing Box. Within the SATCAS project, the FE strategies proposed for the simulation of these processes has demonstrated its feasibility, adequacy and potential for this type of studies. An example of this type of simulations has been presented in this paper.

The methodology developed has demonstrated to be useful for the analysis of deviations that may arise during assembly processes caused by deviations in the constituent parts, deviations in the assembly jigs and/or effects associated to the assembly operations itself. Moreover, it has proven that it might be helpful to improve the processes (through recommendations for the fixation systems, the assembly/fastening sequences, etc) or even the design of the assembly jigs.

6. References

[1] "Variation simulation of fixtured assembly processes for compliant structures using piecewise-linear analysis", Michael Leon Stewart Master Thesis, Brigham Young University, 2004.

[2] "An integrated methodology for statistical tolerance analysis of flexible assemblies", Alan J. Mortensen Master Thesis, Brigham Young University, 2002.

[3] "Deformation analysis of large aerospace components during assembly", M. Saadat, L. Cretin, R. Sim, F. Najafi, The International Journal of Advanced Manufacturing Technology, March 2009, Volume 41, Issue 1-2, pp 145-155.

[4] "Predicting Deformation of Compliant Assemblies Using Covariant Statistical Tolerance Analysis", M. R. Tonks, K. W. Chase, C. C. Smith, Models for Computer Aided Tolerancing in Design and Manufacturing, 2007, pp 321-330.

7. Acknowledgements

The research leading to these results has received funding from the European Union's Seventh Framework Programme (FP7/2007-2013) for the Clean Sky Joint Technology Initiative under grant agreement n° 284961.