NON-LINEAR MULTI-SCALE ANALYSIS OF AEROSPACE STRUCTURES

Orlin Mintchev, Reinhard Diez
LASSO Ingenieurgesellschaft, Leinfelden-Echterdingen, Germany

KEYWORDS – non-linear FE analysis, pre-processing, decohesion, buckling.

ABSTRACT

A reliable analysis of aerospace structures submitted to extreme loads is still a challenge for the engineering community.The complexity of such structures due to their huge size and specific design of some parts requires the use of modern tools for pre- and post-processing and a robust non-linear solver, on one hand, and on the other - a multi-scale approach that allows a better description of the expected non-linear response avoids oversimplifications of the obtained FE-Models.

In our case, a complete section of central fuselage is taken as a representative example. The pre-processing was completely realized with ANSA, that clearly demonstrated its advantages when compared to all the other tools currently available on the market. Different representations were prepared in order to address the issues on two levels: a macro level, i.e. the complete section, and micro level - small assemblies suffering extreme loads. In such a way it is possible to describe phenomena like large deformation and buckling on macro scale and delaminations, decohesion and/or failure of junctions on micro level. The connection between these two levels of resolution is established by using a sub-modelling technique, that is suitable for this type of non-linear analyzes.

1. INTRODUCTION

Many computationally challenging problems that arise in science and engineering exhibit multiscale behaviour. Relevant examples of interest include: large-scale molecular dynamic simulations, fine-scale analysis of crystalline microstructures or composite materials, turbulent transport in high Reynolds number flows and many others.

A non-linear analysis of a complex modern aircraft structure is an other challenging issue now a days. These structures consist of many parts of different size scale and complicated junctions and joints. In case if the structure suffers extreme loads, phenomena like buckling (both local and global) or local damage and progressive faiure take place. The description of such mechanisms is possible if and only if the geometrical details of the strucrure and the interaction between the different parts are modelled in the proper way. The multiscale approach offers a good opportunity this goal to be acheived.

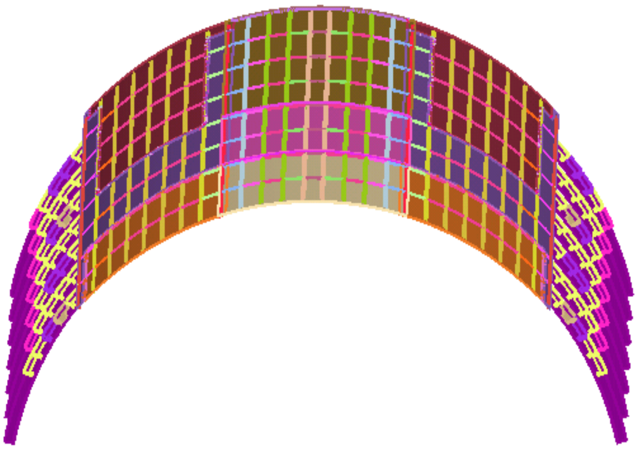
It is well known that classical computational methods for numerical simulations have been designed to operate at a certain preselected scale fixed by the choice of a discretisation parameter. In contrast, the multiscale behaviour needs a correct description on different physically relevant scales. As a result of this, even an attempt to represent all relevant scales in the physical model may lead to an extremely large set of unknowns, requiring a tremendous amount of computer memory, CPU time and excessive algorithmic complexity.

For certain multiscale problems (e.g. as the one considered in this article) one is not actually interested in the fine scale information in the early stage of deformation. In this case a relatevily coarse model provides correct description. However, when once the structure reaches a pre-buckling stage and/or the role of the nonlinearities in the model cannot be neglected any more – a refined model on a lower scale with precisely incorporated details is needed. For this reason, the submodeling technique leads us to the final solution of the problem on a reasonable price. The submodeling technique is used in our analysis because it allows to study local regins of the model suffering extreme deformation even when these areas are not known in advance but their appearence depends on the solution, resp. on the load case under consideration. After such areas are recognised by the global model - a refined mesh, representing the original geometry in a better way, can be used. In addition, some small parts like clips or junctions that were ignored or oversimplified in the global model, can be added. Based on interpolation of the solution from the coarse, global model the load case can be repeated for the local area only. It must be noticed, that this technique is most useful when the detailed modeling of that local region has a negligible effect on the overall solution. This condition implies some limitations on the use the submodeling technique that will be discussed later.

2. MODEL CREATION WITH ANSA

Global Model

At first, the CAD-data was transferred into ANSA (1) by using the Catia to ANSA translator. The obtained ANSA database had a size of about 600 MB and, understandingly, it was difficult to start immediately with the meshing. In order to proceed further, several main sub-groups were created and saved in separate ANSA files: upper and lower shell, side panels, the floor and the support structure in the lower shell area. These sub-groups are shown schematically in Figures 1 and 2.



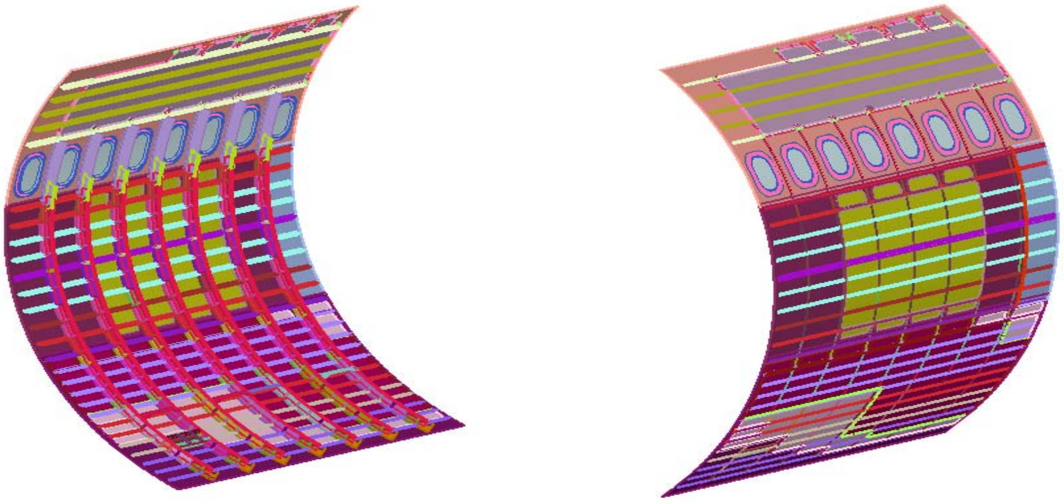
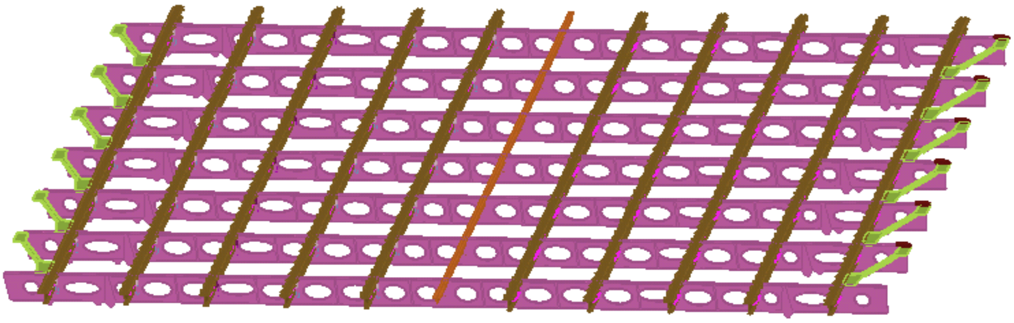
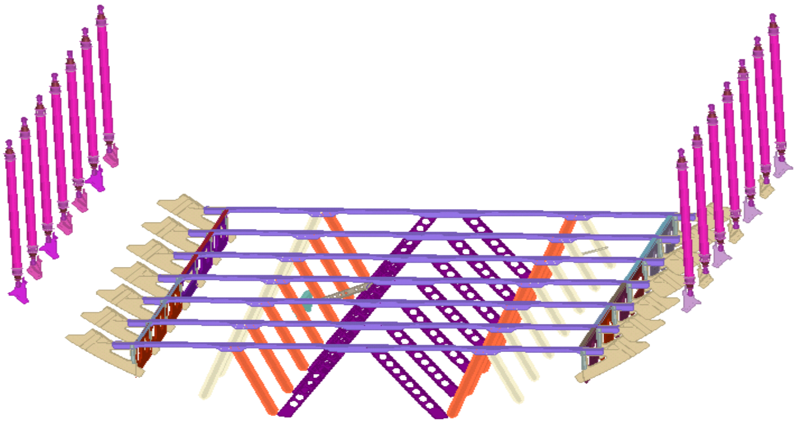


Figure 1 Sub-groups of the upper shell and the two side panels, including clips and frames.





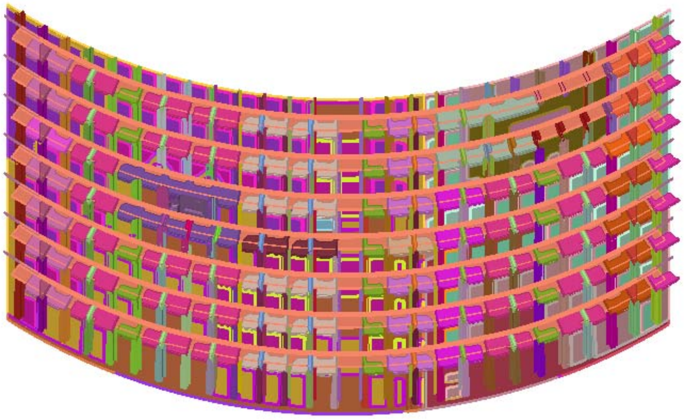


Figure 2 Sub-groups of the floor, the support structure and the lower shell.

These sub-groups were meshed separately, so the work on them was done in parallel by different persons. It must be noted that a very important role played the Data Management of ANSA and in particular the functionality Synchronize Representation. It is typical for aircraft structures that in addition to the simmetry of the structures, many parts have exactly the same topology but are widely spread throughout the assembly. So, using this funcionality, it was possible to mesh "at once" all families of frames, clips, windows and the support structure. This accelerated essentially the process of meshing. The resulting FE-mesh consisted of shell, solid and continuum shell elements. The continuum shell elements were used mainly for meshing the skins of the panels. The reason for this is that they do not have constant thickness, but each of them has some pockets (see Figure 3) that are milled from the inner side while the outer side remains smooth. So, the use of the classical middle surface approach appeared not to be very useful. The pockets are thinner by a factor of ca. 2 (different from pocket to pocket) when compared to the surrounding and the thickness changes nearby abruptly. It is obvious that the exact modelling of this jump would result in a very fine mesh thus causing serious convergence problems when running the global model. In order to avoid this, as a first approximation, a smooth transition was generated with one row of elements with non-constant thickness. For this purpose the continuum shell elements are very useful.

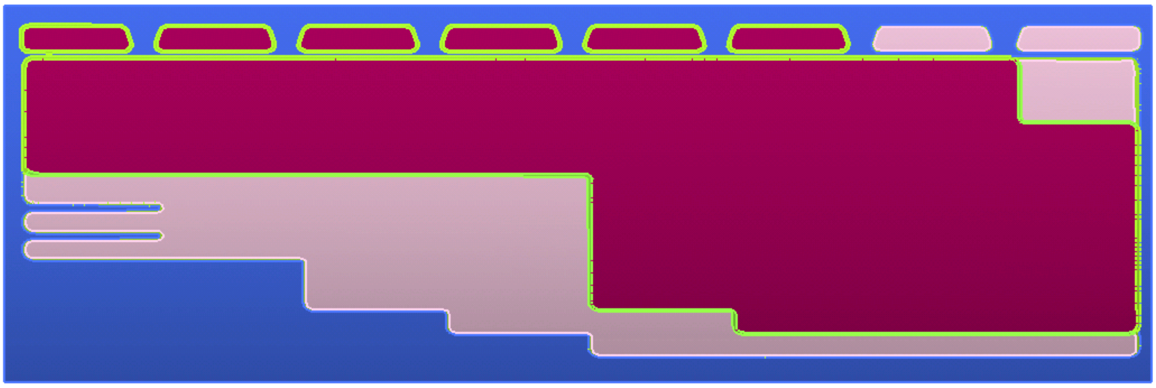


Figure 3 A typical panel skin with pockets of different thicknesses: the main thickness is marked in blue, the transitional rows of elements in greenyellow.

However, it must not be forgotten, that the continuum shell elements correspond in fact to standard shell elements with reduced integration. In case of coarse meshes this might lead to some inaccuracies. For the sake of comparison and completeness, a parallel model with shell elements with full integration instead of the continuum shells was created. For this purpose, an ANSA-script was created that transforms the continuum shells into shells. In the same time all the contact interactions that are defined already for the continuum shells are atomatically transferred to the shells. Some test runs were done, showing a negligibly small difference in the results obtained with both the models.

An other issue was the definition of the contact pairs for ABAQUS (3). Due to the fact that the general contact just appeared in ABAQUS/Standard (but it is still not sure how expensive these algorithms are), the classical master-slave concept was used. The automatic flanges recognition in ANSA was very helpful as first step. After that some contact pairs were modified: the master surface was enlarged in a way to cover all the parts, e.g. clips, that enter in contact with it. Due to the fact that ABAQUS allows the use of discrete slave surfaces, the slave surface was modified by hand, thus including all the small discretely distrubuted contact surfaces. This simplified the control on the contact conditions and improved the datacheck performed by ABAQUS. In the global model all the contacts were defined using the option *TIE. This is a universal tied contact, restricting all degrees of freedom at the nodes (displacements and rotations).

As a result, a complete FE model of the fuselage section was created, having ca. 800 000 elements and ca. 5 million variables (degrees of freedom plus lagrange multipliers).

Submodeling

As noticed before, the submodels have to be created on the basis of the results of the global model. This allows some flexibility when chosing the size of the model, on one hand, and gives the opportunity to focus exactly on the areas of interest, on the other. Due to the expected large deformation and rotation, the node-based submodeling technique is applied in the example presented here. The mesh of the submodel was correspondingly refined and the contact conditions were changed from tied to small sliding contacts with cohesive behavior. Only a normal separation is considered in this example, but this can be easily generalized after adding a shearing debonding criterion. As next, the rivets were taken into account. They were generated with ANSA as ABAQUS fasteners, combined with connector element. This, together with the relaxed contact conditions, offers the opportunity to study in detail effects like debonding and deformation of rivets. In fact, this is one of the main advantages of the submodeling technique, because the study of such local phenomena with global models is not only expensive, but not always possible.

3. RESULTS AND DISCUSSION

As an example a global load case with a pre-defined vertical deflection and rotation of the one end of the section was chosen. The other end of the section is fixed (see Figure 4).

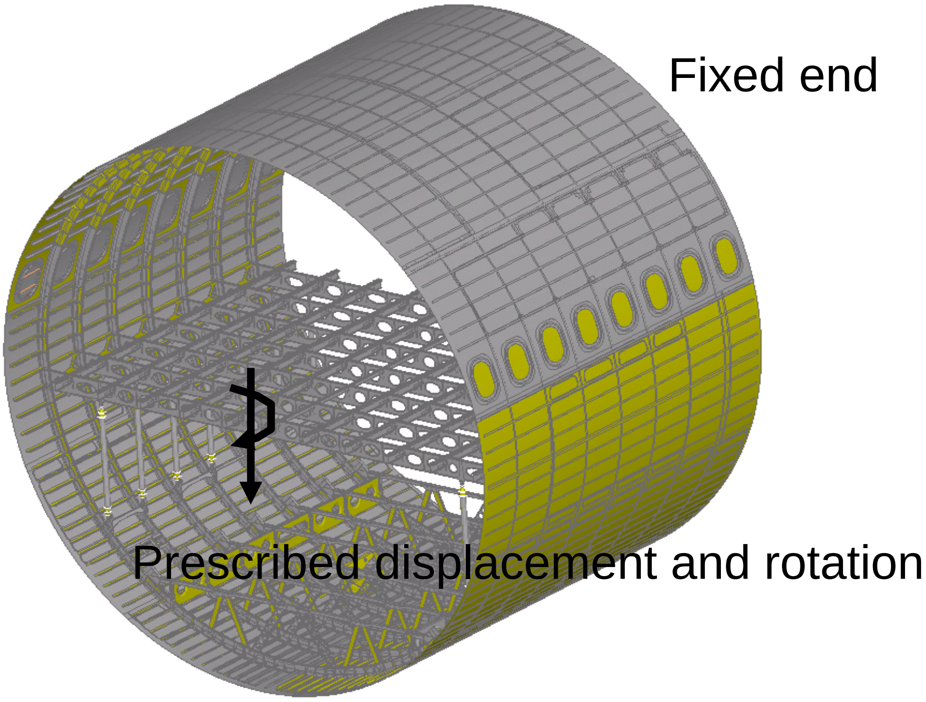


Figure 4 Global model and loading/boundary conditions.

The prepared submodel with defined rivets as fasteners in combination with connectors is shown on Figure 5.

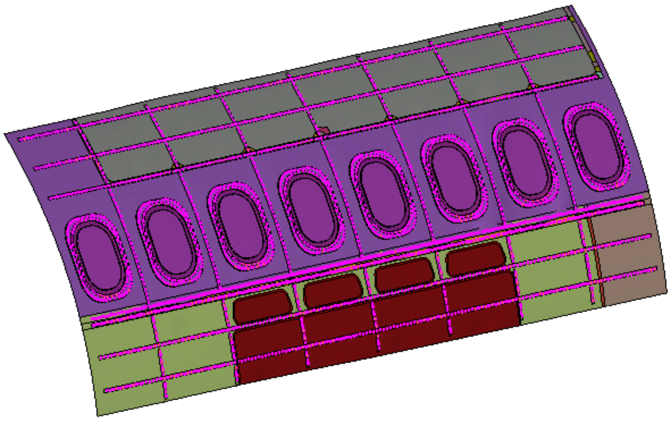


Figure 5 Submodel with rivets.

At first a run of the global model was done. It shows clearly which areas suffer large deformation and have to be examined in detail (see Figure 6).

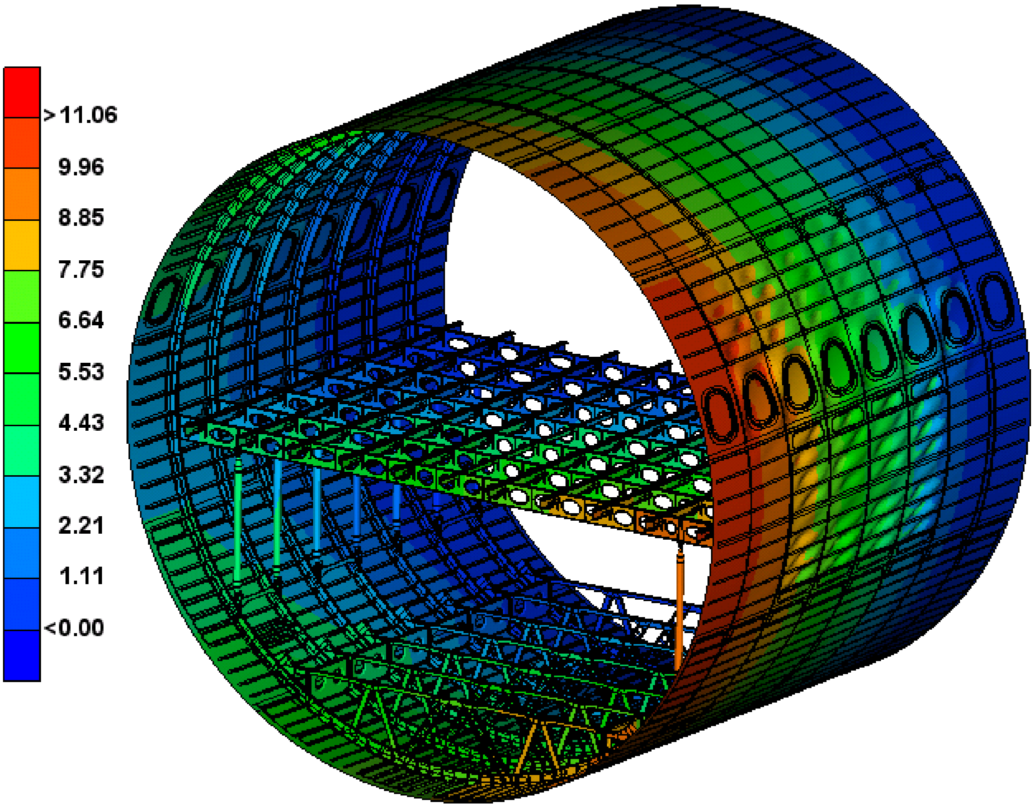


Figure 6 Global model – Displacement magnitude (magnification factor = 10).

On the figure above are seen the areas where the skin starts to buckle. The role of the pockets (where the skin is thinner) is obvious. For this area a submodel was created. At first a test run was done with the refined mesh, but without changing the contact conditions.

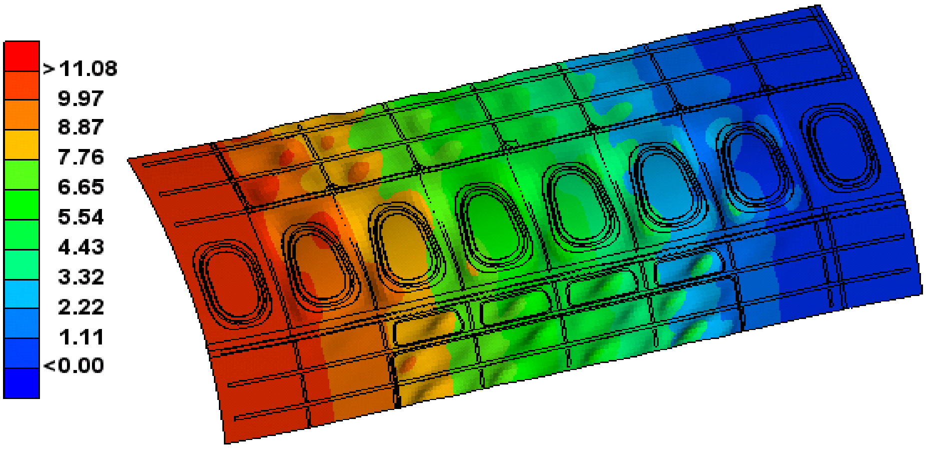


Figure 7 Submodel with refined mesh, but tied contacts (magnification factor = 10).

The comparison between the two results is very good. This demonstrates that the submodeling technique allows to be transferred results even when along the boundary of the submodel the deformation is large. This is a very imprtant advantage, otherwise the submodel had to be much larger.

As next, the submodel with rivets and relaxed contact conditions was launched. At the early states of deformation the coincidence with the previous submodel was very good. But after the debonding started, the deformation path was changed. Those regions where the contact surfaces started to debond, changed the deformation and new buckles were formed (see Figure 8). This illustrates, that a refined analysis is nessecary when additional effects on lower scale can be captured and described by using the submodeling technique.

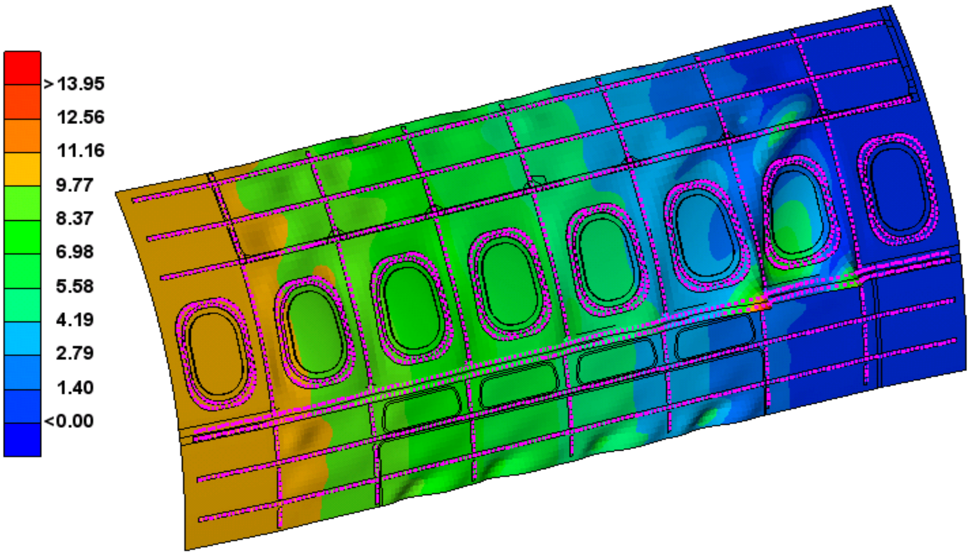


Figure 8 Submodel with rivets and relaxed contacts (magnification factor = 10).

4. CONCLUSIONS

The aim of the present contribution was to demonstrate the advantages of ANSA especially when working with big size models. The global model presented above confirms that.

The excellent performance of the data management of ANSA must be outlined. It accelerated essentially the process of work.

After the first test runs, ANSA was widely used in order to precise and improve the model, e.g. many contact pairs were redefined and the initial mesh was adjusted.

The creation of the submodel was very easy with ANSA. An advantage was the even after refinement of the mesh, the existing contact surfaces were usable immediately after minor changes.

REFERENCES

(1) ANSA version 12.1.5 User’s Guide, BETA CAE Systems S.A., July 2008

(2) μETA PostProcessor version 6.3.2. User’s Guide, BETA CAE Systems S.A., June 2009

(3) ABAQUS User’s Manual, Dassault Systemes 2009.