SIEMENS

COSSMAS

Composite Space Structure Modelling and Analysis Software

Executive Summary Report

(ESA Contract No.4000110033/13/NL/CBi/fk)

September 2016

Rev. 0

	NAME AND FUNCTION	DATE	SIGNATURE
PREPARED BY	F. Cugnon, C. Lequesne, S. Paquay, C. Martin, L. Ballere	27/9/2016	
CHECKED BY	F. Cugnon	27/9/2016	
PROJECT APPROVAL	F. Cugnon	27/9/2016	
APPROVAL			

DOCUMENT CHANGE LOG				
ISSUE	**DATE**	**TOTAL PAGES**	**MODIFIED PAGES**	**REMARKS**
Rev 0	Sept 2016	15		Final release
Rev 1				

TABLE OF CONTENTS

Abbreviation list

COSSMAS : Composite Space Structure Modelling and Analysis Software

ASL : Airbus Safran Launchers

OENG: Open Engineering

LIST: Luxembourg Institute of Science and Technology

dof: degree of freedom

Reference Documents

[RD1] SAMCEF V17.1 html documentation

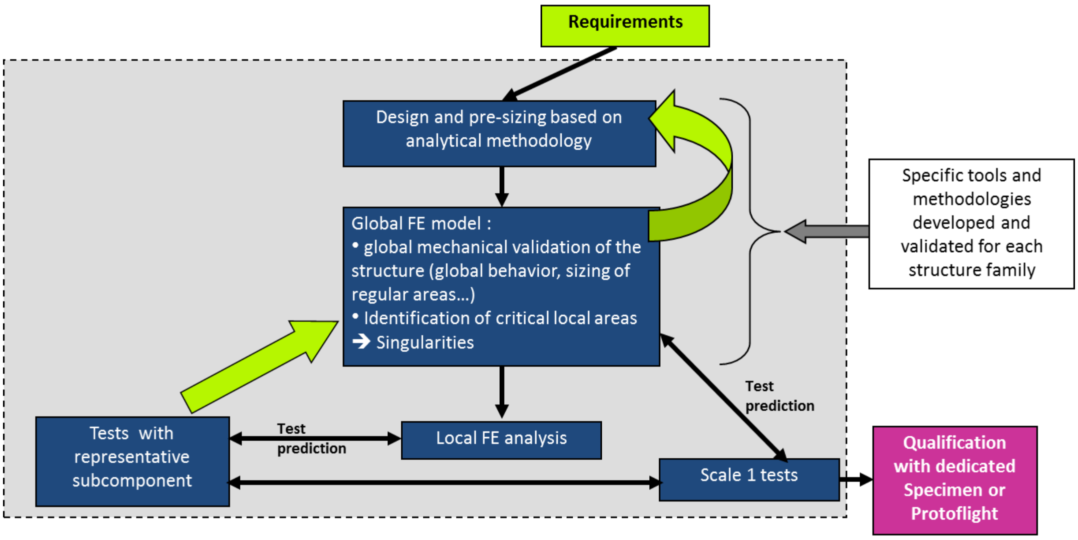
[RD2] S. T. Pinho, C. G. Davila, P. P. Camanho, L. Iannuci et P. Robinson, «Failure models and criteria for FRP under in-plane of three-dimensional stress states includin shear non linearity,» *NASA Technical Memorandum,* vol. 213530, 2005.

1 Introduction

This report is about the COSSMAS project. It resumes the main achievement of the project.

Launcher composite structures are widely made of CFRP composites, even if, in some specific cases, glass or Kevlar fibres can be introduced. Various types of reinforcements can be used (IM fibres, HM fibres …), depending on targeted applications and their design requirements (strength, stiffness …), but it mainly consists in long fibre reinforcements. Two types of resin systems are associated to those fibres, also depending on targeted applications and manufacturing processes: i) thermoset resins (mainly epoxy resins), ii) thermoplastic resins. Laminates used for the manufacturing of launcher composite structures can be either monolithic laminates, based on UD tapes or 2D woven fabrics (most of the time balanced ones), or sandwich laminates where the skins are made from the components previously described and where the cores are mainly aluminium honeycombs. For high pressure vessels, internal liners can also be introduced and they are generally metallic ones (usually made of titanium alloys). Several manufacturing processes are currently used in Airbus Safran Launchers for launcher composite structures, as filament winding (motorcases, high/low pressure vessels, interstage skirts …), hand lay-up, Automatic Fibre Placement and infusion.

Airbus Safran Launchers overall logic for design, sizing and qualification of launcher composite structures can be summarized as follows:



This logic is based on different successive steps, from the customer requirements to the full qualification of the structure that could be obtained after full scale tests with a dedicated specimen or with a protoflight (i.e. qualification & flight) specimen. The first main objective of the COSSMAS project is the improvement of the modelling tools used for global and local FE analysis.

Next, in an industrial context on a subset of transducers and MEMS (Micro-Electro-Mechanical System) applications, the second main objective of COSSMAS project consists in developing simulation tools suitable for the efficient modelling of multi-physic applications using shell elements.

2 Mechanical Solver (SAMCEF)

2.1 Damage and advanced fracture mechanics models improvements

In this section, we describe the improvements that have been implemented in the damage interface material law used in the interface cohesive elements.

2.1.1 Damage interface material ("DAMINT" Behavior)

An enhanced model developed in Abisset's thesis with transverse isotropy has been introduced, where directions 1 and 2 are linked to the material direction and not to the failure mode; it includes a delay damage law. When the material model of the adjacent plies is the enhanced damage ply model and the hypothesis of the elements of the adjacent plies is `.HYP NON_LOCAL`, damages depend on the crack density of the two adjacent plies. If the maximum crack damage is reached in one of the 2 plies, we consider that the interface is fully damaged. The implemented Abisset's model includes also a specific way to compute in-plane damages from interface damage, material parameters, crack density and directions of adjacent plies.

2.2 LARC04 criterion

In the Samcef solver, the composite failure criteria of Langley Research Centre version 4 (LaRC04) has been implemented [RD1][RD2]. The criteria are composed with six conditions which correspond to composite failure modes. These criteria describe particularly the compression failure by taking into account the kinking of the fibre and the non-linearity behaviour of the matrix submitted to in plane shear loading. The criteria were implemented in the post-processing module of Samcef. After an analysis, the stress component in the plies are treated and the values of the different functions are provided. The failure in one element is predicted if the value reaches the unity. To validate the implementation, the results provided in the literature reference [RD2] are reproduced.

2.3 Shape function through the thickness of Composite Elements

For the traditional Mindlin shell/plate element, the displacement at any point of the laminate plate is expressed from the displacements of the neutral surface and the mean curvatures of the neutral surface. A direct application of this first order shear deformation theory leads to the so-called equivalent single layer theory. It produces inadequate predictions when it is applied to relatively thick laminated composites with material layers that have highly dissimilar stiffness characteristics. As an improvement, a shear correction factor is judiciously introduced to modify the stiffness for transverse shear as it is currently practiced in the Samcef code (.hyp Mindlin). Alternatively, a refinement of the equivalent single layer theory can be also performed. To this end, one might consider the partial layerwise theories that use layerwise expansions uniquely for in-plane displacement components. To be more precise, a piecewise linear in-plane displacement function is added, as it is suggested from the Nasa's research (Tessler et al., 2010), to the first order shear deformation theory. This piecewise linear function is also called a “zigzag” function. Depending on the formulations those functions are determined from the material and geometrical properties of the laminate.

2.3.1 Validation

The new zigzag element employs the zigzag functions while the existing Mindlin element in Samcef uses the shear correction factor in order to improve the performance of the original first-order shear deformation theory, when dealing with thick laminates. Consequently, both formulations should converge to the same prediction expected from the first-order theory in the absence of the transverse shear effect in a very thin laminate. For the case of thick laminates, where the shear effect cannot be neglected, a direct comparison with the reference results also issued from the Nasa formulation allows a good validation.

Through a series of academic test-cases (see deliverable D2.1), it is concluded that:

(1) For static analysis and with laminated structures, the new zigzag element should be used as an alternative to the existing Mindlin element only in the case of thick laminates.
(2) For dynamic analysis, the new zigzag element does not offer a true advantage.

2.4 Performance of SAMCEF for large composite models

In this project, several tasks have focus on the performance of the modelling tools used for large composite models. Main improvement concerns the following points.

- The optimization of the bank files written by NX CAE GUI, and the reengineering of several SAMCEF Bacon commands allow a significant improvement of the pre-processing phase.
- Improved storage strategy, suppression of unnecessary writes in the post-processing files, and optimization of elements outputs have yielded to a drastic size reduction of the post-processing files
- New restart capabilities after a parallel simulation (or a co-simulation) complete tools provided to the user to benefit of HPC computing.

2.5 Sub-modeling in SAMCEF

In this section, we will distinguish 3 ways to connect detailed models of components to a global model of a structure. The first one, is currently used in ASL. It consists in including the local refined model in the global one, connecting them using gluing techniques, and solving the whole monolithic model. This technique can yield to very heavy models if several local components are refined. The second approach consists in using the structural zoom capability of SAMCEF that allow importing boundary conditions from a global model to a detailed model of a given component. This approach is a chaining of two simulations, as the second one (detailed model) doesn't influence on the first one. The resulting coupling is not as strong as the first method. The third method consists in using the SAMCEF Supervisor to strongly couple global and local models. As this coupling is done at iteration level, the solution is identical to the first method (monolithic solution). The main advantage is that the computation can be very efficiently parallelized, and that no huge model as to be solve. The 3 above methods have been validated solving the holed cylinder problem with local reinforcement.

3 Multi-layer multi-physic shell element in OOFELIE

3.1 Hypotheses and supported layer types

The implementation of multi-layer multi-physic shell elements inside OOFELIE::Multiphysics was mainly based on:

- Some multi-layer shell theory aspects that were developed by LIST
- Some previous implementations of mono-layer mechanical shell elements that were already available in OOFELIE::Multiphysics (MITC formulation).

The main hypotheses and particularities of this new shell element are the following:

- Shear and membrane locking are contrasted by means of the Mixed Interpolation of Tensorial Components (MITC formulation).
- The geometric description is given by the surface finite element supporting the shell and the thicknesses and eccentricity properties of the element.
- Displacement field inside the element is interpolated from the 3 translations and the 2 out-of- plane rotation at its nodes.
- The interpolation of scalar fields (electric potential and temperature) is linear or quadratic across each layer's thickness.
- The classical element connectivities are supported: Quad4, Quad8, Tri3 and Tri6.
- The non-linear geometric aspects are not considered.

The supported layer types in the multi-layer shell are:

- Mono-physic layers:
  - MECHANICAL_LAYER
  - THERMAL_LAYER
  - DIELECTRIC_LAYER
  - ELKN_LAYER
- Multi-physic layers
  - THERMOMECHANICAL_LAYER
  - PIEZOELECTRIC_LAYER
  - ELECTROTHERMAL_LAYER
  - ELECTROTHERMOMECHANICAL_LAYER

3.2 Implementation

The main implementation challenges for OENG in the COSSMAS project were:

- Reworking of OOFELIE::Multiphysics solver infrastructure to support variable number of dofs at layer level and stacking sequence definition of multi-layer shell
- The new multi-layer multi-physic shell itself
- Generalization of thermal and electrical boundary conditions to support dofs at layer level
- Implementation of 3D/shell and shell/shell connections

All these challenges were successfully addressed during the COSSMAS project.

3.3 Conclusions

A multi-layer multi-physic shell element based on MITC formulation was successfully implemented inside OOFELIE::Multiphysics in the framework of COSSMAS project.

The numerous successful validation cases permit us to be confident with the quality of the implementation that was performed.

4 Graphical user interface for Composites

The NX™ CAE environment for Samcef™ Solver Suite software enables engineers to build finite element (FE) models, define solution parameters and visualize results using the Samcef Solver Suite. This environment allows you to take advantage of the powerful geometry editing, meshing and general preprocessing capabilities in NX CAE to build analysis models for the Samcef Solver Suite faster than traditional CAE tools. The Samcef Solver Suite includes unique capabilities for the prediction of complex, nonlinear phenomena, such as progressive damage in the unidirectional and woven fabric plies of a laminated composite structure, and delamination with coupling to the damage inside the plies.
Using the Samcef Solver Suite environment with NX Laminate Composites to model and simulate delamination is illustrated on Figure 1.

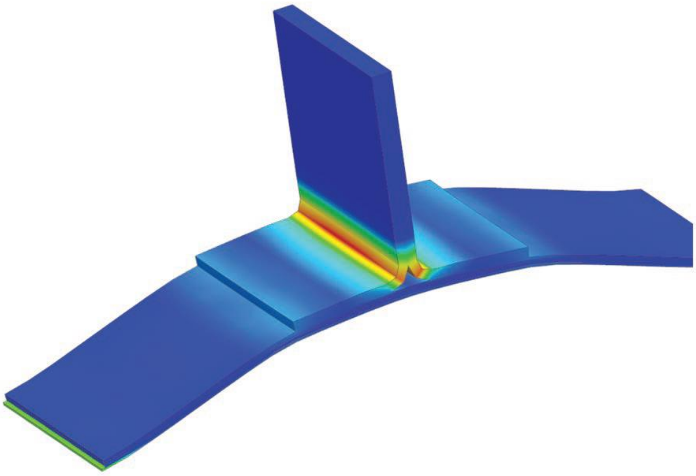


Figure 1: Composite model in NX CAE

To meet ASL needs, this generic tool has been completed by some NX OPEN code (customization tool) to manage multi-harmonic composite models.

Also, the implementation of a first prototype of “OOFELIE::Multiphysics - NX CAE” driver able to set up (by a generic procedure) models including multi-layer multi-physic shells was also supported by the COSSMAS project.

5 Composite cases studies for software validation

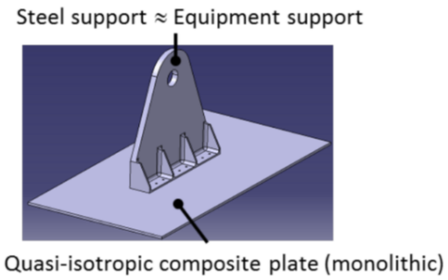
5.1 Introduction

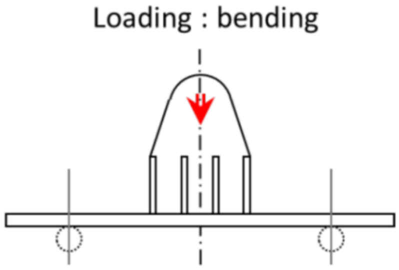
First, Airbus Safran Launchers chose three industrial cases dealing with launcher structures in order to validate the developments performed in the scope of COSSMAS.

Next, in order to complete the validation of new multi-layer multi-physic shell in OOFELIE and prove its usability in an industrial context, OENG has modelled two multi-physic applications with new implemented shell elements.

5.2 Validations on equipment support

The first studied case consists in a quasi-isotropic composite plate made of monolithic laminate which is subjected to local indentation when submitted to a “three points” bending. This indentation is due to a steel support bolted on the upper face of the composite plate (see following figures). This technological specimen is representative of equipment supports used in launcher composite structures.





This test case was chosen to evaluate the developments on damage models and failure criteria because the damage kinetics involved is highly complex and because experimental data are available. The loading imposed on this technological specimen (bending) enables to evaluate shape functions through the thickness developed for shell elements (zigzag).

5.2.1 Validation of damage interface material (“damint” behaviour)

This study permits to witness the differences in the computation of delamination between the previous coupling law (“damint mode 1”) and the Abisset’s formulation (“damint mode 2”) governing the interaction between matrix cracking and delamination. Here, delamination is more important with Abisset’s formulation, as expected considering enhancements brought to previous coupling law. As more energy is dissipated by delamination, with the new coupling law, fibre damage is delayed in some plies. Thus, the force drop associated to fibre failure in these plies happens later with Abisset’s formulation than with the previous coupling law.

5.2.2 Validation of the LaRC04 criterion

For this test case, LaRC04 criterion shows results that fit with the physical behaviour expected in terms of failures. Moreover, LaRC04 criterion appears to be more efficient than Hashin's criterion to describe failure mechanisms needed to be known during the mechanical analysis of structures (6 components versus 4). In particular, kink-band initiation can be evaluated, and in some cases this damage mechanism can lead to a catastrophic failure of structures.

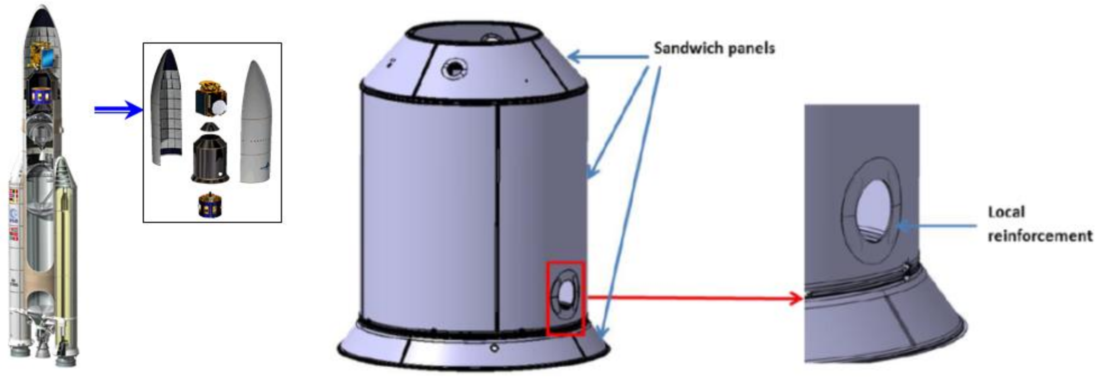
5.2.3 Validation of shape function through thickness of composite elements

Displacements fields obtained with a composite plate made of volume elements, mindlin shell elements or zigzag shell elements are close to each other. The assessment of experimental values of displacements is better with volume elements, then with mindlin shell elements and eventually with zigzag shell elements.

For an industrial use, for now, a lot of restrictions exist concerning the use of the "zigzag" hypothesis. The more penalizing one is that stresses and strains cannot be obtained.

5.3 Validation of sub-modelling methods on SYLDA access holes

The SYLDA is the dual launch system of Ariane 5. It is made of sandwich panels. To have an access to the lower payload, it is required to have a circular hole at the bottom of the cylinder of the SYLDA. This hole is reinforced locally by monolithic layers on the inner and the outer face of the sandwich panel. Because of the size of the hole compared to the size of the whole structure, this test case is particularly appropriate to validate sub-modelling techniques proposed in this project.

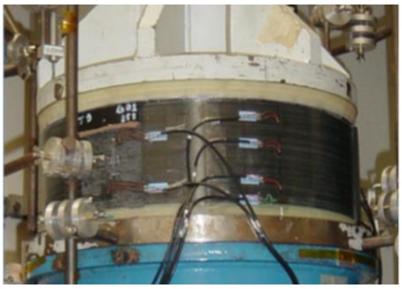


In the present test case, the weak coupling method (.ZOOM) seems to be the more convenient for displacement assessment: the computation time is the lowest by far (3h compared to 19h with .APS and 22h with co-simulation) and results are satisfying (quite close to those obtained with the .APS command which is the method currently used by ASL). Nevertheless, the co-simulation method (using the supervisor) improves accuracy concerning stresses assessment (thanks to strong coupling). For both methods, it appears that best results are obtained when the border between the global model and the local model is the same as the one used with .APS (i.e. between shell elements used for the global model and 3D brick elements used for the local model).

A priori, the co-simulation method could be used by Airbus Safran Launchers instead of .APS. Especially for cases where detailed patches have a strong influence on the global model and for cases with a lot of detailed patches (slaves) (in that case, a proper parallel configuration could lead to a lower computation time than using .APS).

5.4 *Validation of pre- and post-processing of multi-harmonic models on a motor case skirt test case*

The application proposed is a motor case skirt. The skirt is a cylindrical composite structure integrated on a motor case vessel by the use of bonded junctions. It makes the link between the pressurized vessel and launcher inter-stage structures, allowing the transmission of loads from one stage to another one. The studied skirt is a reduced scale skirt (ϕ450) made of 2D woven fabrics.



5.4.1 Multi-harmonic model pre-processing

The 6 UDOs implemented in NX Open have been successfully tested in NX10 and NX11. Their use is instinctive and simple. The associated epilog is well written.

5.4.2 Multi-harmonic model post-processing

Concerning post-processing, no problem has been encountered in post-processing “3D existing mesh and 2D mesh”, neither in NX10, nor in NX11. However, there were some problems with the post-processing “3D built from multi-harmonics”. This last point would need further investigations.

5.5 The micro-bolometer

The micro-bolometer is a pixel of an infrared sensor (for imaging).

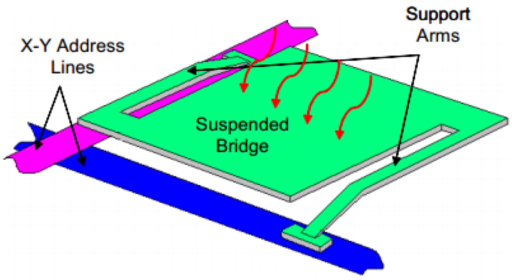


Figure 2: typical architecture of IR sensor pixel (micro-bolometer)

It is mainly made of a 3-layer plate supported by arms. In the intermediate layer of the plate, there is a high resistive layer (with high temperature dependency). Then, by a measure of the electrical resistance of the cell, it is possible to know its temperature that is related to the level incoming IR power. If we do that for all the micro-bolometers of the IR sensor, we are able to reconstruct the complete image of IR scene.

For this application, all the 3D and shells results were fully similar (cell resistance vs incoming IR power, temperature field, electrical potential field and displacement field)

5.6 MEMS Piezoelectric energy harvester

This kind of systems permits to retrieve power from a vibrating environment that will be delivered to systems located in the neighborhood. This system is mainly made of a cantilever plate with a piezoelectric layer and an added mass.

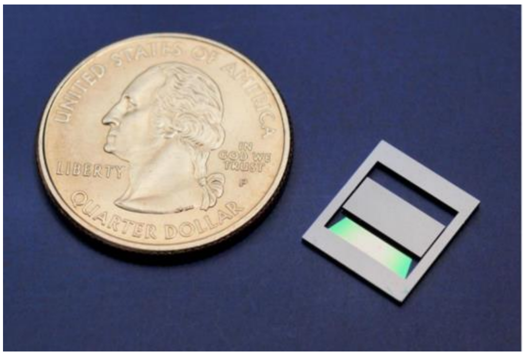


Figure 3: the piezoelectric MEMS energy harvester (From microGen presentation at Energy Harvesting & Storage 2012)

For this kind of systems, one important aspect is the extraction of eigen values. All modal 3D results and shell results were fully similar (eigen frequency values and related mode shapes)

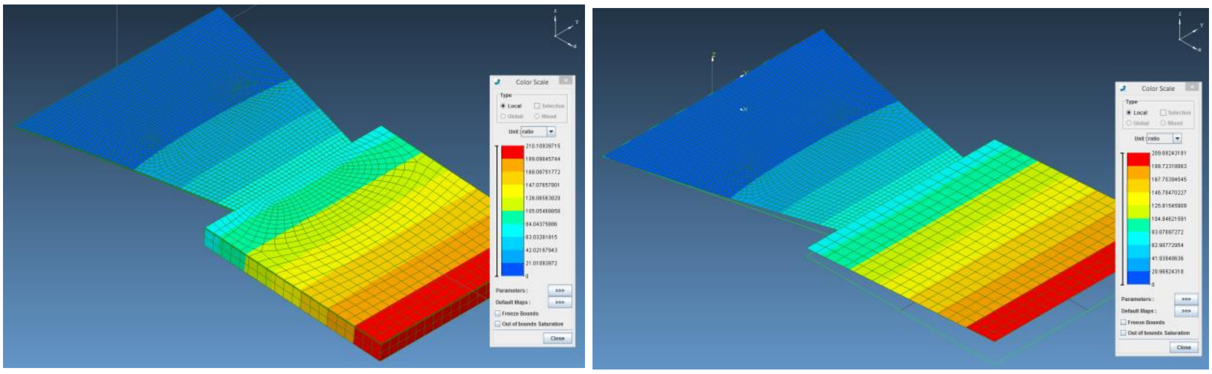


Figure 4: piezoelectric energy harvester - mode 1 – nodal displacement - 3D model vs shell model