NUMERICAL MODELING METHODOLOGY FOR AN ADVANCED LIGHTWEIGHT DEBRIS CONTAINMENT SYSTEM

G. Toussaint[1a], A. Gakwaya[2], D. Nandlall[1b], S. Caulfeild[3] and A. Shahkarami[4]

[1]*Defence Research & Development Canada – Valcartier, Qc, Qc, Canada,* *Genevieve.Toussaint@drdc-rddc.gc.ca*, *Dennis.Nandlall@drdc-rddc.gc.ca*
[2]*Dept. of mechanical engineering, U. Laval, Qc, Canada,* *agakwaya@gmc.ulaval.ca*
[3]*Pratt & Whitney Canada, On, Canada*
[4]*Composites Group, Dept of Civil Engineering and Materials Engineering, The U. of British Columbia, Bc, Canada*

Developing design methodologies based on experimentally validated predictive numerical simulation methods will enhance existing capabilities in predicting failure modes and structural design optimization for the high velocity impact problems. This paper is thus concerned with the set up of a methodology for modeling and simulation of the containment problem for the case of a real hybrid metallic/soft layered composite structure [1, 2]. To realize this new design, a debris protection fan case composed of a basic metallic shell structure with a dry Kevlar wrap around it is considered. The fan blade is modeled with ***Mat_piecewise_linear_plasticity, material model #24*** in LS-DYNA, while the metallic structure of the fan case is modeled with ***Mat_plastic_kinematic, material model #3*** in LS-DYNA. The remaining wrapped layered composite structure is modeled using a ***woven fabric material model*** developed in [3]. This material model can capture the ballistic response of multi-layer fabric panels and is implemented as a UMAT subroutine in an equivalent shell element corresponding to the representative volume element (RVE) for computational efficiency [3, 4]. Our objective is to assess how this material model can be applied in a real industrial application.

INTRODUCTION

Since the 70's, the numerical simulations of penetration of target by projectiles are subjects of interest. One such interesting application is the design and the analysis of efficient lightweight engine fan blade containment system [2]. In fact, in case of a severe engine malfunction, debris poses a serious threat to aircrafts and their occupants. Conventional methods of protecting the aircraft and their occupants rely on the use of heavy structures to restrict debris from escaping the engine containment case. However for better technical and economical efficiency, the need to strengthen and lighten aircrafts and to improve safety requirements has pushed the aerospace industry to explore alternatives to the metallic containment systems already in use. This paper is thus concerned with the set up of a methodology for modeling and simulation of the containment problem for the case of a real hybrid metallic/soft layered composite structure [1].

In order to simulate the impact between the blade and the hybrid fan case, a multistep analysis process is required. In the first step, a preloading stage using an implicit analysis is performed to generate stress and strain in the blades induced by the build up of the cruise rotational velocity. Then, using results of the previous step, the second step performs an explicit analysis to generate the debris and to study the high velocity impact problem between the blade and the hybrid fan case.

Some of the main issues to be considered in assessing the suitability of a computational model in industrial environment are: finding a constitutive law capable of capturing the complex physical interaction including intra and inter layers failure mechanisms as well as contact interaction related to the gap size between the adjacent layers, and finding the appropriate mesh size limited by available computer capacity and production time. It is the aim of this work to assess if theoretically recommended element size, resulting from representative unit cell homogenization process, is appropriate for practical industrial applications.

METHODOLOGY FOR THE NUMERICAL SIMULATION OF A FAN BLADE CONTAINMENT PROBLEM

Stress Induced in the Blades by the Rotation

The first step of the simulation methodology is to generate the stress and strain built up in the blades by rotating the engine from rest up to a certain cruising rotational speed, set in our application to 1047rad/s. To perform this pre-stress analysis, an implicit dynamic finite element solver is required and we used the LS-DYNA code, which implement both the implicit and explicit algorithms for transient dynamic structural problems involving high velocity loadings. The generation of the model data involves the creation of the blades geometry and the assignment of material properties to the parts. The LS-DYNA material model used for the blade is ***Mat_piecewise_-linear_plasticity, option #24***. The material chosen is titanium and the properties used are: ρ = 0.004424g/mm^3; E = 115.142GPa; ρ = 0.35; σ_Y = 827.371MPa; ETAN = 951.476MPa; FS (failure strain) = 0.25. Also, a ***Hughes Liu shell element formulation, option #6***, is chosen with a selective reduced integration (S/R). During the implicit analysis, all the root nodes of the blades are fixed in space, and a velocity is assigned to each node of the blade according to a linear function starting from 0 to 1047rad/s. When the analysis is completed, we obtain a ***dynain file*** containing the strain and stresses distribution in the blades. This is then used as input in the following transient dynamic analysis of the blade release and fan case impact event using an explicit algorithm.

Debris Generation and Impact on the Fan Case

In this analysis phase, we have to simulate the blade release event (generation of debris) and to model and analyse the high velocity debris impact on the fan case. The material model for the blade is the same as the one presented in the previous section: ***Mat_piecewise_linear_plasticity, option #24***. The fan case is composed of two sections: the metallic ring part, to be impacted first and the dry Kevlar multiple layers woven fabric wrapped around it [2, 3]. We assigned a different material model to each of them. The metallic ring structure in Aluminium (Al) is modeled with the material model ***Mat_plastic_kinematic, option #3*** and the Kevlar fabric wrap is modeled with a new advanced material model, the ***Fabric Crossover UMAT model*** (***FCU model***) implemented by A. Shahkarami [3]. A continuum modeling approach is used to model the fabrics in such a way that the interaction of yarns in a unit cell is smeared into a single representative shell element. The Al material parameters are: ρ = 0.0027g/mm^3; E = 68.948GPa; υ = 0.33; σ_Y = 213.78MPa; ETAN = 77.951MPa; FS = 0.18 and the values for the time vs strain rate are 1:1; 100:1.1; 1000:1.3; 2500:1.5. The properties used for the Kevlar® 129 S-726 (fabric designation used by DuPont, Inc) are Yarn = 840 denier; Fiber elastic modulus = 96GPa; FS = 3%; Average count (warp) = 26 threads/in; Average count (weft) = 27 threads/in; Maximum crimp = 3.44%; Minimum crimp = 0.50%; Areal density = 198.6g/m^2.

The formulation chosen for the shell element is the ***Hughes-Liu formulation, option #1***. For the boundary conditions, rotation will be allowed around the z axis for the root nodes but the displacement will be fixed along the rotation axis. Also, one side of the fan case will be fixed in the 3D space. Also, in the explicit analysis, two types of contact are used. The first one is a ***Contact_tied_nodes_to_surface*** and allows for blade releasing. The second one is a ***Contact_eroding_surface_to_surface*** and defines the contact between each part of the model. The choice of the mesh size to be used for the Kevlar wrap depends on the theoretical model and the comparison with the experimental results. To make sure that the ***FCU model*** represents adequately the response of a fabric under ballistic impact, we compared our numerical results with the results obtained from an ***ELVS*** (***Enhanced Laser Velocity System***) test data. The next section shows that our numerical model reproduced very well these experimental data.

COMPARISON BETWEEN EXPERIMENTAL AND NUMERICAL RESULTS

The ***FCU model*** used to model the Kevlar wrap was developed in [3] and this numerical model was compared with experimental data (***ELVS*** Test). The experimental model was made of a target of 4 Kevlar® 129 plies represented by a panel of approximate dimensions 201mm x 201mm. The panel was impacted on its centre by a blunt-nosed projectile in RCC (42.1mm length, 5.5mm diameter and mass of 3g) with a striking velocity of 120m/s. To simplify the numerical model, a quarter of a plate was created and symmetrical boundary conditions were imposed to each side. The properties of each shell element were determined by the homogenisation of the properties of the fibres in each direction of the wrap. The theoretical element size of approximately 1mm x 1mm was used. The thickness of each shell element was taken as 0.3132mm and a distance of 0.45mm between each layer was considered. The projectile was modeled exactly like the experimental one.

COMPUTATIONAL RESULTS AND DISCUSSIONS

Composite or hybrid containment fan cases are in practice made from multiple layers of high performance fabrics to defeat the specific types of threats they are exposed to. Hence, as a result, a major part of the fabric target energy absorption stems from the interaction of individual layers with each other in the pack. To study such interaction existing in a real fan case, the shell crossover model is used to simulate the ballistic impact experiments on multi-ply Kevlar® 129 targets. Two key parameters are important in assessing the suitability of computational model in industrial environment: finding the appropriate element mesh size and the gap size between the adjacent layers. The determination of the gap between the adjacent layers (inter-fabric stiffness representing the interaction of adjacent layers during compression) is one of the challenges of modeling multi-layer fabric targets. Due to modeling constraints and in absence of experimental data, the distance between the Al ring and the Kevlar ply was set to 1.586mm instead of 1.622mm and the distance between the Kevlar plies was set to 1.244mm instead of 0.45mm. When an inter-ply gap is implemented, it was found efficient to activate the contact between the adjacent layers through the ***Contact_automatic_surface_to_surface*** option in LS-DYNA and this proved to be successful in maintaining the contact between the layers of the fabric pack, as well as between the projectile and the target. On the other hand, for industrial application, the mesh size of the fabric elements should be small enough to give reasonable resolution of stresses, but large enough so that full fan case simulations can be completed in a reasonable time. However the theoretical mesh determined previously [1, 3] generated too many elements in a real model. Thus, for modeling purpose and reasonable computational costs, we had to scale this mesh. This section presents the preliminary results obtained in assessing the suitability of the ***FCU model*** in curved environment. Figure 1 presents the results obtained with two different mesh sizes compared to experimental data. In all cases, the predictions seem to be slightly stiffer than the experimental measurements. The second mesh (≅ 8mm x 8mm) corresponds to what is dictated by a real application model and is 8 times larger than the first one (recommended theoretical model) deduced from homogenization procedures.

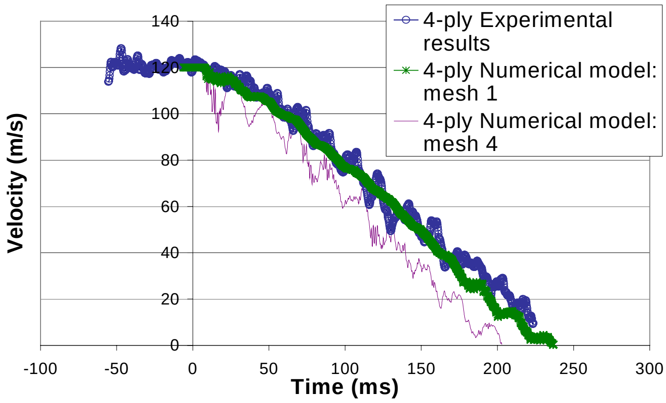

Figure 1: Comparison between the predicted (2 different meshes) and the ELVS measured velocity-time history for four-ply of fully fixed Kevlar® 129 fabric

Since the general trends are reproduced with the coarser mesh (8 times the theoretical mesh size), the simulations of multiple layers fabric wrap around an Al ring were thus conducted using the corresponding mesh. Figure 2 presents the results obtained for a blade impacting a metallic layer with a 2 Kevlar plies wrapped around it.

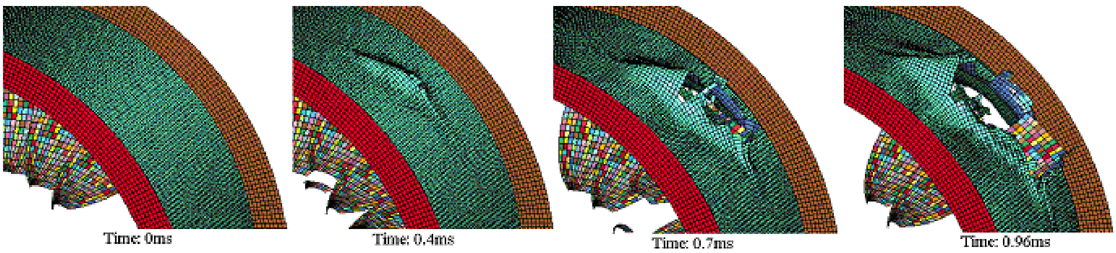

Figure 2: Numerical model of a blade impacting a hybrid structure (Al layer with 2 Kevlar plies)

To validate the results, different approaches were used. First, an estimate of the ***kinetic energy*** was determined by hand calculation ($E_k = 2.0773x10^{14}$ N·mm) and the obtained value was compared to the one given in the ***status.out file*** (E_k' = 1.80418x10^{14} N·mm). We hence concluded that the kinetic energy was of the same order of magnitude as the one expected. At the same time, the mass properties of each component and of the body in the ***d3hsp file*** were verified to make sure it was modeled properly.

After this comparison, we verified that the ***total energy*** was effectively conserved during the simulation of a blade impacting a metallic layer with a 2 Kevlar plies around it.

The third step consisted in calculating the ***hourglass energy*** created by the distortion of the elements. Since we used uniformly reduced integration, hourglass, viscous and/or stiffness based stresses were added to the physical stresses at local element level in order to suppress the hourglass deformation modes that accompany the one-point quadrature. But since this is an artificial energy, we had to make sure it was negligible. The ratio of hourglass energy with respect to the total internal energy was approximately 10%. However this hourglass energy represents actually 0.2% of the total energy, which is a numerically acceptable result, hence we assume it to be negligible.

During our multi-ply fabrics analysis, the failure of the yarns seems to lead to premature perforation of the targets in many cases. Examination of the strains and stresses of the shell elements of the multi-layer models revealed the presence of a significant amount of numerical noise in the tensile response of yarns. Hence as a last model verification, we have to check that the tension developed in the shell element is within acceptable range. Hence one of the best ways to know about the status of the fabric is to plot yarn tensions (***T1 & T2***) and compare them against their tensile strength. In the ***FCU model***, two criteria are used to delete an element: either when the ultimate tensile strength (187N) is reached in both directions or when an element has reached its ultimate tensile strength in one direction and is distorted. On figure 3, we can see an example of the tensile strength given in both directions for one element deleted during the simulation. In this case, we can observe an important numerical instability: the maximum tensile strength reached is 116N instead of 187N.

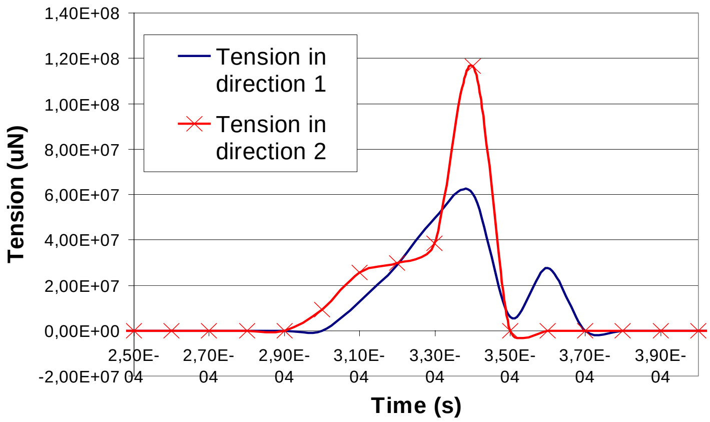

Figure 3: Tensile strength of a fabric element in both directions

The numerical oscillations observed in the case of multi-ply fabrics appeared more pronounced in comparison with the 1-ply model and this can stem from the interaction of the fabric layers in the simulations. The excessive noise resulting from the contact algorithms caused the premature failure of the yarns and hence the early perforation of the targets. This problem should be further explored and improved by implementing more advanced failure criteria, or through averaging or damping out of the oscillations. The verifications done previously are only meaningfull for results consistency but they do not guarantee the validity of the results. More work is still to be done.

CONCLUSION

In this project, we considered several hypotheses. First, the real model was simplified to 5 blades with no hub. Also, one of the challenges in modeling multi-layer fabric targets was the determination of the gap between the adjacent layers. Due to modeling constraints and in absence of experimental data, the distance between the Al and the Kevlar ply and the distance between the Kevlar plies were set to approximate values according to the used model geometry and suggested industrial mesh size. However, in order to obtain an approximation of the distance between the mid-surfaces of successive layers, transverse compression test data on multi-ply fabrics should be used. Moreover, the shell element size modeled was 8 times larger than the one tested experimentally (≅ 1mm x 1mm). The experimental test was performed on a flat panel instead of cylindrical one, but preliminary sensitivity analysis showed that we could use reasonably this mesh. When we applied the preceding results to the real fan case structure, we encountered new numerical problems that had to be addressed first, one by one. In our assumption, we also supposed that the damage behaviour of the fabric model would be the same for a coarser mesh. Finally, the yarn failure criterion used in fabric model was simplified and the instantaneous strain-to-failure model incorporated in the shell element was highly sensitive to any oscillations in numerical noise introduced from a variety of source such as contact [5].

Thus, this model still needs improvements. For example, actually we used the cheapest ***Hughes-Liu element formulation, option #7*** (uniformly reduced integration, 1 integration point) to see how far we could go. However, in view of the results it is recommended to try on the same mesh, the ***option #6*** which uses a selectively reduced integration instead of just one integration point which requires a much more refined mesh. If the mesh is not successful, we can proceed to mesh refinement with a mesh size greater than the theoretical one until we find which mesh yield stable results. If the conclusion is that we have to use the theoretical unit cell mesh size (as provided by [3]), which is computationally prohibitive for real structure, then we have to modify the ***FCU model*** so as to use larger unit cell with a corresponding equivalent shell element. These include the generation of a new mesh of the fan case to get appropriate results. In any case, for better confidence in the results, we need to compare the results obtained numerically with experimental data. Also, another problem adressed in the paper, especially for targets with higher number of layers, is the premature failure of yarns, indicating that a more appropriate failure model, not affected by the oscillations in the response should be implemented for accurate predictions in penetrating impact simulations.

In short, the main purpose of this report was to develop a computational methodology for the fan blade containment with a hybrid fan case and make sure that the model at hand works properly. Simplifying assumptions have been made in order to obtain results within reasonable timeframe. However, in order to get more realistic results, the ***FCU model*** needs more improvements [5]; moreover for a real multilayer fabric simulation, a multiprocessor version of the code is required in order to obtain the results within a reasonable time in a production run.

ACKNOWLEDGMENTS

We fully acknowledge the financial support from the ***Consortium for Research and Innovation in Aerospace Quebec*** (***CRIAQ***), which made possible this collaborative work between industry and academia. Special thanks are also given to A. Shahkarami who provided guidance for using the ***Fabric Crossover UMAT model*** (***FCU model***).

REFERENCES

[1] G. Toussaint, *Modélisation du confinement des débris des pales d'un moteur d'avion dans un caisson hybride métal-composite*, Mémoire de maîtrise, Dépt. de génie mécanique, Université Laval (2006)

[2] J. Simmons, D. Erlich, and D. Shockey, *Explicit finite element modeling of multilayer composite fabric for gas turbine engine containment systems part 3: model development and simulation of experiments*, DOT/FAA/AR-04/40, P3, Technical Memorandum (2004)

[3] A. Shahkarami, R. Vaziri, A. Poursartip and N. Tajani, *An Efficient Mechanistic Approach to Modelling the Ballistic Response of Multi-Layer Fabrics*, The University of British Columbia and DuPont Advanced Fibers Systems (2005)

[4] A. Tabiei and I. Ivanov, *Computational micro-mechanical model of flexible woven fabric for finite element impact simulation*, Int. J. Numer. Meth. Engng. **53**, 1259-1276 (2002)

[5] A. N. Shahkarami, *An efficient unit cell based numerical model for Continuum representation of fabric systems*, PhD thesis, Dept. of civil engineering, The University of British Columbia (2006)