**FINITE ELEMENT MODEL FOR IMPACT ON COMPOSITE STRUCTURES**

**C. Garnier**, ML. Pastor† and B. Lorrain

* Daher-Socata, Research and Technology department, 23 toute de Tours 41401 Montrichard cedex, France  
  e-mail: christian.garnier@enit.fr

† Université de Toulouse, Université Paul Sabatier, Institut Clément Ader, IUT de Tarbes  
  Rue Lautréamont, 65000 Tarbes, France  
  e-mail: ml.pastor@iut-tarbes.fr

‡ Université de Toulouse, Institut National Polytechnique de Toulouse, Ecole Nationale d’Ingénieurs de Tarbes, Laboratoire Génie de Production  
  47 avenue d’Azereix, 65000 Tarbes, France  
  e-mail: bernard.lorrain@enit.fr

**Key words:** Orphan meshing, 3D shell, cohesive surface.

**Summary.** *The aim of this study is to create a finite element model, using Abaqus FEM code, to predict the behaviour of composite structure under low-velocity impact. Results of this model, in terms of damaged area, are compared to NDT results obtained by IR Thermography. Then, this model is used to modeling Compression After Impact. These second results are compared to experiment.*

**1 INTRODUCTION**

Nowadays, in aerospace industry, composite materials are used but their behaviour is not well known. During their life in aerostructure, these materials are subjected to many different loads as maintainability impacts, hail, birds, and others. These loads can be classified according to their energies or strain speed. Under dynamics loads or medium loads, the structure is affected and many damages are visible. In this case, this structure is automatically replaced. In case of quasi-static loads, the surface of the structure is few affected but damage can occur in it. The DET and GVI (General Visual Inspection) inspection do not reveal systematically these defects. If intrinsic damages exist and are not revealed, aircraft crashes can occur. One of the main problems is to know how the structure responds on impact in terms of damage produced in it and residual strength after impact.

In case of FEM model to design an impact on plate with the FEM code Abaqus, many possibilities exist. UMAT or VUMAT exist and allow the definition of the material and...
failure criteria. Some other studies show the utility of the Virtual Crack Closure Test (VCCT) to shows the behaviour of the composite plate subjected to low-velocity impact.

In this study, the 3D FEM model is build with conventional shell for plies and with cohesive surfaces for the ply interfaces. For each ply, the used damaged criterion is Hashin for tensile fiber failure, compressive fiber failure, tensile matrix failure and compressive matrix failure. For the cohesive surfaces, 2 criteria are used. This 3D FEM model is based on a standard mesh. This mesh is the basis of all the plies and interfaces. The plate is designed using orphan mesh to create plies and interfaces. Element used for the plies are SC8R and cohesive elements are used for the interfaces.

The plate is made of HR carbon fiber/epoxy resin commonly used in aerostructure. The mechanical properties are caught from experimental tests.

<table>
<thead>
<tr>
<th></th>
<th>E11 (GPa)</th>
<th>E22 (GPa)</th>
<th>G12 (GPa)</th>
<th>G21 (GPa)</th>
<th>v12</th>
<th>v12</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>63.3</td>
<td>62.5</td>
<td>4.8</td>
<td>4.8</td>
<td>0.049</td>
<td>0.051</td>
</tr>
</tbody>
</table>

Table 1 : Tensile properties of the carbon/epoxy material

Results of the Output Data Base are managed in order to separate elements with zero properties and other elements. This set of element is compared to the damaged area obtained by NDT using active IR thermography.

Results correlate well between FEM and experiments. Further works are in progress in order to use the ODB file, impacted plate, to simulate the Compression After Impact and correlate it with experiments.

REFERENCES