Residual strength validation of a composite stiffened panel virtually impacted

Gambino Benedetto*a, Iannuzzo Generosoa, Linari Maurob, Mottola Maracob, Scannavino Fabiob

aAlenia Aermacchi S.p.A., Viale dell’Aeronautica s.n.c, 80038 Pomigliano D’arco (Na), Italy
bMSC Software, Corso d’Italia 45, 00198 Roma, Italy

Abstract

Most modern structural system designs rely on a damage tolerance philosophy, which means that a structure can withstand some damage without failure.

The residual strength of a structure, i.e. the load that a damaged structure can still carry without failure, can be significantly affected by the presence of a crack or a damaged area and is usually substantially lower than the strength of the undamaged structure [1].

In the study presented in this paper MSC.Nastran sol700 explicit solver has been used to simulate different impact conditions on a stiffened composite panel [2]. This has allowed estimating and analysing the damage effects on the matrix and fibers of the composite panel. The virtual damaged panel has been loaded in a non-linear implicit simulation using MSC.Nastran sol400 solution to predict the residual strength [3,4,5,6]. The simulation results were in good agreement with experimental tests.

© 2014 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/3.0/).
Peer-review under responsibility of the Organizing Committee of DRaF2014

Keywords: Composite stiffened panel, Virtual impact, Residual strength,

* Corresponding author. Tel.: +39-081-8872252; fax: +39-081-8876042.
E-mail address: benedetto.gambino@alenia.it
1. Introduction

Certification of aircraft built of composite structures is a complex topic. Real commercial or military aircraft structures must operate satisfactorily when damaged. If the damage is severe, the damage tolerance requirement may mean only completion of the flight in which the damage occurred without injury to passengers and crew. Required structural capability as a function of damage extent varies from one category of aircraft to another and must be negotiated between the original equipment manufacturer (OEM) and the certifying agency.

Damage tolerance is the ability of the structure to resist catastrophic failure in the presence of damage without being repaired, for a specified number of operations (flights) or length of time in service. Damage tolerance is usually demonstrated by residual strength tests conducted after cyclic loading of a component that has been damaged in a well defined manner.

Residual strength must be greater than limit load by a factor, defined by the certifying authority, that depends on the ability to detect the damage during an inspection.

The residual strength of a structure, i.e. the load that a damaged structure can still carry without failure, can be significantly affected by the presence of a crack or a damaged area and is usually substantially lower than the strength of the undamaged structure. To have, therefore, an analysis technique able to accurately estimate the residual strength of a composite stiffened panel can be very beneficial for the aeronautical industries in terms of cost and time. In this context the Finite Element (FE) method can be used to investigate more in details the damage due to impact and its growth in composite structures.

The study presented in this paper shows an numerical approach to address the problem based on MSC.Nastran sol700 explicit solver to simulate different impact conditions. This allows to estimate and analyze the damage effects on the matrix and fibers of the composite panel.

After the virtual damages inflictions, the panel has been analyzed in a non-linear implicit simulation using MSC.Nastran sol400 solution to predict the residual strength.

2. Description of the structure used in the experimental test

2.1. Stiffened Panel

The stiffened panel used in the experimental test is composed by the following components:

- Skin;
- Four longitudinal stiffeners symmetrically placed with respect to the longitudinal skin center line;
- Two dummy transversal ribs;
- Eight stringer cleats to connect stiffeners and ribs.

![Fig. 1. Stiffened panel used in the experimental tests](image)
Skin and longitudinal stiffeners are made of composite material (unidirectional carbon fiber with epoxy resin) while ribs and stringer cleats are manufactured using Aluminium 2024-T3.

2.2. Constraints

To prevent the panel from bouncing upon impact, two belts are laid out on the panel on the side where impact occurs.

This panel is part of a wing and in the real structure there is continuity in the transversal direction. To simulate this condition, in the experimental test the longitudinal side of the panel is constrained to avoid vertical and horizontal motion.

Summarizing, the following constraints have been defined:

- The lower ends of the dummy ribs have been fixed. In this way the effects of the belts are in some way retrieved
- The longitudinal edges of the skin have been constrained in vertical and transversal direction.

2.3. Data for impact definition

The impactor consists of a structure that moves vertically and ends with a spherical tip with a diameter of 1 inch. It means that the impact occurs normally with respect to the skin plane. Impacts occur in the bottom side of the skin (that one opposite to the stringers location).

Six impact location have been considered (see fig.4).
Two different impact energies have been considered: 50 and 70 joule. Each impact condition has one of these two energies assigned. As results for each impact the indentation on the skin surface has been verified. The energy associated to each impact location and the corresponding indentations are summarized in the table below.

| Location | Energy (J) | Indentation (mm) | Velocity (mm/msec) |
|----------|------------|------------------|--------------------|
| 1        | 50         | 0.15             | 3.3333             |
| 2        | 50         | 0.06             | 3.3333             |
| 3        | 50         | 0.05             | 3.3333             |
| 4        | 70         | 0.02             | 3.944              |
| 5        | 70         | 0.13             | 3.944              |
| 6        | 50         | 0.16             | 3.3333             |

From the estimated mass and the energy involved it was possible to calculate the speed associated to the impact for each of the six positions of the impactor.

- Impact energy = 50 joule (= 50 KN·mm)
  \[ E_{kinetic} = 50 = \frac{1}{2} M v^2 = 0.5 \cdot 9 \cdot v^2 \rightarrow v = \sqrt{\frac{50}{0.5 \cdot 9}} = 3.3 \text{ m/sec} = 3.3 \text{ mm/m sec} \]

- Impact energy = 70 joule (= 70 KN·mm)
  \[ E_{kinetic} = 70 = \frac{1}{2} M v^2 = 0.5 \cdot 9 \cdot v^2 \rightarrow v = \sqrt{\frac{70}{0.5 \cdot 9}} = 3.944 \text{ m/sec} = 3.944 \text{ mm/m sec} \]
3. Finite element model description

3.1. Mesh size determination

Considering that we are going to use an explicit approach for the numerical representation of the phenomenon, we can calculate the time required for a perturbation to propagate properly within the element of minimum size. The speed of propagation of the perturbation is given by the speed of propagation of sound within the material taken into consideration.

\[ l_{element} = c \cdot \Delta t = \frac{c}{20 \cdot f_{max}} \rightarrow f_{max} = \frac{c}{20 \cdot l_{element}} = \frac{\sqrt{E/\rho}}{20 \cdot l_{element}} \]

Considering as representative for the composite material the fiber mechanical properties it results:

\[ \text{element size} = 4 \text{ mm} \quad \text{follows} \quad f_{max} \approx 120 \text{ Hz} \]

![Finite element mesh size](image)

Fig. 5. Finite element mesh size

3.2. Elements, properties and materials

All the composite components have been modeled by solid elements. They have been defined starting from the tool surface in order to define a direction of the thickness that is congruent with the stacking sequence. Layered CHEXA elements have been used. The stacking sequence has been defined by associating them to the PCOMPLS property entry. The material for each of the ply is defined by MATD022 entry which allows the definition of orthotropic material properties.

Standard solid elements have been used for the filling material. Isotropic material data are sufficient to describe its mechanical behaviors, so MATD01 entry has been used. It has been associated to the CHEXA elements by using a PSOLID property entry.

2D elements have been used for the dummy ribs and the stringer cleats. CQUAD4 properties are defined by PSHELLD entries and the mechanical behaviors have been defined by MATD01 entry.

3.3. Impactor finite element model

The impactor has been modeled by considering a 2D spherical surface (CQUAD4 and PSHELLD) to which a rigid material has been assigned (MATD020).
The impactor has been located as close as possible to the panel surface in order to reduce the time to reach it at the specified velocity.

Fig. 6. Impactor finite element model

The mass of 9 kg has been defined by assigning specific thickness and material density to the elements. In particular a thickness of 6 mm has been defined by an arbitrary definition of an internal (Rint) and external (Rext) radius for the spherical surface. All these arbitrary data have been used to calculate the mass density useful for matching the target mass.

The other material properties have been defined as these of the steel, but in any case they are not considered because MATD020 entry allows defining a rigid body. It means that the only important data is the mass density. This type of material entry allows also forcing a constraint for the rigid body. In this case all the degrees of freedom are constrained except the vertical displacements.

4. Impact analysis setting

The analysis of the impact phenomenon assumes that high frequencies are involved in the system response. Considering the mesh density to be used, an implicit code could lead to very high calculation time. Explicit codes are more suitable for the solution of these problems even in case, like ours, where we talk about low-speed impacts. Furthermore we also need to build a model able to analyse damage location using for example Lamb wave approach where very high frequency excitations are involved.

The calculations have been performed by using the explicit Lagrangian approach available in MSC.Nastran solution 700

4.1. Damage/failure modeling in sol 700

Historically, damage studies of layered composite structure were based on the concept of the “First Ply Failure”. It means that the laminate is assumed to have failed when the first ply fails. This approach is good when degradation is unacceptable but no information is provided about the residual strength of the structure in the general situation. In order to determine the realistic design margin it can be helpful to know if a laminate that has started to fail tends to yield catastrophically or if the crack is only local and the structure does not yield as the redistribution of loads provides it a residual strength.

In a composite laminate, while individual plies may be brittle and fail catastrophically, a whole laminate usually doesn’t. This is because:

• When individual ply fails, the entire laminate may not;
• The load can be redistributed to the other intact layers avoiding the catastrophic failure occurring by reducing immediately the strength of the laminate to zero (FPF).

This mechanism results in a ‘progressive’ ply failure. It means that after one ply fails, the strength is reduced somewhat but not completely. Following this first ply failure, other plies may follow if the load is increased. In fact, even if with many failed plies, a composite part may still have significant and useful load bearing capability.

In sol 700 several material models dealing specifically with composites are present. For this study the Material Model 22 (Chang-Chang Composite Failure Model) has been adopted.

Five material parameters are used in the three failure criteria based on Chang and Chang:

- \( S_1 \) Longitudinal Tensile Strength
- \( S_2 \) Transverse Tensile Strength
- \( S_{12} \) Shear Strength
- \( C_2 \) Transverse Compressive Strength
- \( \alpha \) Nonlinear Shear Stress Parameter

In plane stress, the strain is given in terms of stress as:

\[
\varepsilon_1 = \frac{1}{E_1} (\sigma_1 - v_1 \sigma_2) \\
\varepsilon_2 = \frac{1}{E_2} (\sigma_2 - v_2 \sigma_1) \\
2\varepsilon_{12} = \frac{1}{G_{12}} \tau_{12} + \alpha \tau_{12}^2
\]

Three failure modes are considered by this material model. A fiber matrix shearing term augments each damage mode:

\[
\bar{t} = \left( \frac{\tau_{12}^2}{2G_{12}} + \frac{3}{4} \alpha \tau_{12}^4 \right) / \left[ \frac{\tau S_{12}^2}{2G_{12}} + \frac{3}{4} \alpha S_{12}^4 \right]
\]

which is the ratio of the shear stress to the shear strength.

The matrix cracking failure criteria is determined from:

\[
F_{\text{MATRIX}} = \left( \frac{\sigma_2}{S_2} \right)^2 + \bar{t}
\]

Failure is assumed whenever \( F_{\text{MATRIX}} > 1 \). If \( F_{\text{MATRIX}} > 1 \), then the material constants \( E_2, G_{12}, v_1 \) and \( v_2 \) are set to zero.

The compression failure criteria is given as:

\[
F_{\text{COMPRESSION}} = \left( \frac{\sigma_2}{2S_{12}} \right)^2 + \left[ \left( \frac{C}{2S_{12}} \right)^2 - 1 \right] \frac{\sigma_2}{C_2} + \bar{t}
\]
Failure is assumed to occur when $F_{\text{COMPRESSION}} > 1$. If $F_{\text{COMPRESSION}} > 1$, then the material constants $E_2$, $\nu_1$ and $\nu_2$ are set to zero.

The final failure mode is due to fiber breakage:

$$F_{\text{FIBER}} = \left(\frac{\sigma_1}{S_1}\right)^2 + \bar{t}$$

Failure is assumed whenever $F_{\text{FIBER}} > 1$. If $F_{\text{FIBER}} > 1$, then the material constants $E_1$, $E_2$, $G_{12}$, $\nu_1$ and $\nu_2$ are set to zero. Both shell and solid elements can be used.

Additional integration point variables have been added to store the documented damage data for the shell, thick shell and solid element. These additional variables are listed in the following table (‘i’ is the index for the ply number).

![Table 2. Integration point variables](image)

Fig. 7. MATD022 Nastran Card
Looking at the value for these results it is clear that they are in some way derived from the three failure criteria listed above. In fact for the failure criteria the failure condition is that the resulting value must be higher or equal to 1 while for the new history variables:

- Value = 1.0 means that the ply is still in the elastic field, it means undamaged;
- Value = 0.0 means that the ply is totally damaged;
- Intermediate values define the progression of the ply damage up to the failure.

In order to have a single data for each history variable in each element a medium value is calculated for each of them over all the plies according to the following relationships:

| Description | Integration Point |
|-------------|-------------------|
| $EF = \frac{1}{n_{ip}} \sum_{i=1}^{n_{ip}} ef(i)$ | 1 |
| $CM = \frac{1}{n_{ip}} \sum_{i=1}^{n_{ip}} cm(i)$ | 2 |
| $ED = \frac{1}{n_{ip}} \sum_{i=1}^{n_{ip}} ed(i)$ | 3 |

These components are stored as element component 7 in place of the effective plastic strain. In LS-PrePost this plastic strain or effective plastic strain must be selected as stress results. Then below the IPn option the three damage components can be displayed (as fringe plot or graph). Note that the legend of the displayed results refers to outer, inner and middle position of the shell. It is due to the fact that these additional data are stored in the place where plastic strain is stored normally.

The possibility to have a global result for the damage factors was one of the reasons that lead choosing this material model for this study.

5. Numerical test results and comparison with experimental data

Considering that local damages have been verified in the experimental test without a real failure for the structure, the time limit for the numerical tests has been defined as that one in which failure occurs; it means that considering the history variables EF (global fiber tensile failure index), CM (global matrix tensile failure index) and ED (global matrix compressive failure index) as the output parameters representing the damage status, composite failure is occurring when at least one of them reaches a null value. In fact in this condition a meaningful permanent deformation occurs determining a result that is not in line with experimental test. Note that the analysis has been interrupted as soon as the described failure condition is verified in the intermediate checks executed by LS-PrePost.

An analysis has been executed for each impact location. The following results have been taking into account:
• The elements damage indices (EF, CM and ED) have been visualized by fringe plots and that one for which the damaged area is larger has been considered as representative for the specific impact location. The size of the damaged area is calculated by considering its size in longitudinal ($\Delta X$) and transversal ($\Delta Y$) directions;
• Referring to the damaged area the node with the maximum Z-displacement has been detected. The relative diagram vs time has been built and filtered in LS-PrePost. The diagram of the filtered displacement tends to assume a constant value when the impact process ends, and the impacting body comes off from the panel. If the structure has been damaged (not necessarily broken), this final displacement could be non-zero representing the residual deformation induced by the impact process. This value was taken as representative of indentation.

To clarify how indentation is influenced by composite failure (at least one null damage index), the effects on filtered displacement time history is shown for a case with failure and another with only damage. The first diagram represents the calculated time history while the second one is obtained by a filtering process.

*Case 1: Composite Failure*

![Fig. 8. Composite failure](image)

*Case 2: Composite Damage*

![Fig. 9. Composite damage](image)

It is evident how composite failure increase a lot the residual displacement estimated by the almost asymptotical Z-displacement in the filtered diagram. In case of damage the asymptotic filtered Z-displacement is very close to zero.

In the following figure the comparison between simulation and experimental test has been reported.
Difference between the experimental and the numerical aspects has to be considered. In fact in the experimental case the several impacts are executed in sequence while in the numerical simulation they have been executed separately. It means that in the experimental case each impact following the first has act on a pre-stressed/pre-deformed/damaged structure. Differently In the numerical case each impact test start with a structure in its initial configuration.

In conclusion, taking into account all of these differences and lack of material data, the results have been obtained seems to be sufficiently good. Furthermore it has been possible to analyze all the modeling solution and material models usable to simulate impacts on composite structure.

6. Residual strength estimation

In order to evaluate residual strength index, compressive test simulation is executed on both undamaged and damaged model in order to verify if and how much the mechanical behaviors of the structure have been influenced by the several previous impacts.

The simulation has been executed by using MSC Nastran solution 400. Finite element model used for the compressive test simulation has been built in order to satisfy the following items:

- Allows to obtain good results in sufficiently limited time;
- Reproduce the numerical model used in the impact test simulation;
- Model the damages of the composite material due to the preliminary impact tests.

It has been decided to use 2D shell elements to model the panel

- The same mesh has been used positioning the grid points in the midplane of the skin;
- The identifiers of the 2D elements are exactly the same of the corresponding 3D elements in the impact test simulation model.

The material properties of the damaged regions have been degraded according to the damage indices calculated in each of the impact test simulations.
6.1. Damaged and undamaged compressive test results comparison

The results obtained in case of damaged and undamaged structure can be summarized in the following graphs:

The blue curve describes the results for the damaged structure. The following representative points can be highlighted:

- Failure indices are lower than one up to the 45% of the load. It should be noted how the damaged regions act as if holes are there. In fact they generate the typical stress concentration around an hole. At 45.5% of the load the structure start to fail.

- Around the 52% of the total load the failure has extended over the most of the regions close to the stiffeners. In particular at 52.5% these regions start to be connected each other. It means that the damage is probably extended also to the stiffeners.
Fig. 13. (a) Failure index shown at 52.5% of applied load  
Fig. 13. (b) Failure index shown at 68% of applied load

The maximum load that the damaged structure can lead is occurring at 68% of applied load

- After the maximum load is reached the structure start to deform as in a plastic condition up to a situation where it slip at almost constant load. At this point the complete transversal section is damaged.

7. Conclusions

In this study was presented an numerical technique based on simulation by finite elements for the determination of the residual strength after impact of a composite stiffened panel.

I has been identified a modeling technique, using the MSC Nastran sol 700, to determine virtual impacts and subsequently it was performed a nonlinear implicit calculation, using MSC Nastran sol 400, to determine the residual strength.

The technique showed good correlations with the experimental tests, highlighting the possibility of future investigations to more massive applications in the field of aeronautical composite structures.

References

[1] “Project Management Considerations: Design architectures for civil and military aircraft”, B.Gambino - Encyclopedia of Composites, 2nd Edition, Editors: Luigi Nicolais, Assunta Borzacchiello, publisher John Wiley & Sons, Hoboken, New Jersey, 2012;
[2] Explicit Nonlinear User’s Guide (MSC Nastran 2013)
[3] MSC Nastran Composite Seminar Notes – NAS113_Sec06_Advanced_failure_theory_and_prediction.
[4] Nonlinear Structural Analysis with MSC Nastran (NAS400)
[5] Contact Analysis using MSC Nastran and MSC Patran (NAS133)
[6] Advanced Contact Analysis Using MSC Nastran Seminar Notes (NAS134)