VALIDATION OF THE DAMSTRAT TOOL FOR PROGRESSIVE FAILURE ANALYSIS IN COMPOSITE LAUNCHER STRUCTURES

Martin Côté[1], Marie Jacquesson[1], Hicham Khalouf[2] and Alexis Salzman[2]

[1]CNES – Launchers Directorate
Rond-Point de l'Espace, 91023 Evry CEDEX, France
Martin.cote.26@ulaval.ca, Marie.Jacquesson@cnes.fr
[2]Medysys Air Espace
11, rue Marius Terce, 31300 Toulouse, France

ABSTRACT

Most of the failure criteria used for composite materials do not consider progressive failure of the material. These criteria are often not sufficiently realistic and are unable to predict real failure of a laminate. CNES (the French Space Agency) has access to DAMSTRAT, a finite element software specialized in modelling progressive failure in composites. The following article describes a validation procedure which was achieved on the DAMSTRAT software. Six different FEA models were analysed using DAMSTRAT, each one having an increasing complexity. The results were compared to experimental, analytical and numerical results found in litterature. Comparison was also achieved by comparing the results of a NASTRAN non progressive failure analysis to these of DAMSTRAT. The paper ends with a series of recommendation regarding the use of DAMSTRAT for dimensioning simple structures and underlines the benefits this software may have for the space industry.

1. INTRODUCTION

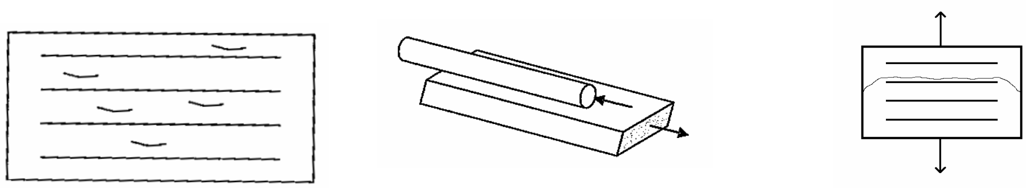
The space launchers industry is always looking for ways to decrease launchers structural mass without diminishing their reliability or resistance. Lighter than steel and often more resistant that most metals, composites materials are a good solution to this problem and composite structures are more and more taking the place of their metallic counterparts. Composite structures designed using first ply failure criteria are generally over designed and more massive than they actually need to be. Progressive failure analysis often achieves a more realistic modelisation of a composite behaviour under a progressive load, increasing the maximum loads the structure can withstand and thus diminishing even more its mass. The DAMSTRAT software is a finite element tool specialized in modelisation of progressive failure analysis in composite material structures.

SLEIGHT [1] has achieved a progressive failure analysis of five composite structures using an numerical method. His results were compared to experimental data. An incremental load was applied to each structure element. At each iteration, strains and stresses were computed in every element of the objects and three different failure criteria were verified. As soon as a failure was detected in an element, the elastic moduli of the ply containing this element were reduced, strains and stresses were computed again and if final failure was not reached, the load was increased on the structure. SLEIGHT's method was in accordance with experimental data. GOYAL et al. [2] developed an analysis method modelising intralaminar and interlaminar progressive failure. Interlaminar failure can occur with three mechanisms: matrix tensile failure, fibre tensile failure and matrix/fibre shear failure. Interlaminar failure is modelised using surface elements between each ply. Interlaminar cracking occurs when the interface traction reaches a critical limit, and this crack will grow when the crack work reaches the material resistance to crack propagation. KASHTALYAN and SOUTIS [3] have modelise intralaminar damage being mostly composed of cracks parallel to the fibres in plies different from the material main axis. Interlaminar damage will occur when these cracks reach another ply.

2. DAMSTRAT SOFTWARE

DAMSTRAT is a finite element analysis tool specifically designed for modelising progressive failure analysis in unidirectional ply composite structures. The DAMSTRAT version used for this paper (v6.0) is used to intralaminar progressive failure modeling. The software makes non linear analysis. DAMSTRAT v6.0 uses classical finite elements such as beams (2D elements) and bricks (3D elements). Version 4.0 and beyond are developped by Medysys Air Espace (Toulouse, France).

The damage model used by DAMSTRAT gradually decreases the elastic properties of a laminate when the laminate is loaded. The damage level in the material is a function of two thermodynamic forces. These forces are themselves derived from the strain energy in the structure. Three types of damage are accounted for in the modelisation : matrix microcracking, fiber/matrix shear decohesion and brittle matrix tensile cracking.



a) b) c)

Figure 1 : Types of damage modelised by DAMSTRAT. a) Matrix microcracking. b) Fiber/Matrix shear decohesion. c) Brittle matrix tensile cracking. [4]

The elastic properties of each element decrease when load is applied and damage appears. For a local tensile load, the following formulas modelise the decrease :

$$E_{11} = E_{11}^0 \qquad E_{22} = E_{22}^0\left(1-d'\right) \qquad G_{12} = G_{12}^0\left(1-d\right)$$

where E_{11}^0, E_{22}^0 and G_{12}^0 are the initial non damaged elastic moduli and parameters d and d' are functions of the structural strain energy mentioned above. When a compressive local load is applied to an element, we have :

$$E_{11} = E_{11}^0\left(1+\gamma\langle\varepsilon_{11}\rangle_-\right) \qquad \text{if} \qquad \varepsilon_{11}^c \le \varepsilon_{11} \le \varepsilon_{11}^t$$

where $\langle\varepsilon_{11}\rangle_- = \varepsilon_{11}$ if $\varepsilon_{11} < 0$, else $\langle\varepsilon_{11}\rangle_- = 0$, and γ is also a parameter function of the strain energy. ε_{11}^c and ε_{11}^t are respectively the critical compressive and tensile strains in fiber direction. Analysis with DAMSTRAT require characterisation of about 20 material parameters which can be determined using tension/compression tests [5]. Many of these parameters are not conventional but can be obtained with usual load/unload test campaigns. The parameters are extracted with a methodology developed on prupose and explain in [4]. The required parameters are listed below.

Parameter category	**Parameters**	
	Name	Symbol
Elastic properties	Initial elastic modulus (L axis)	E_{11}^{0}
	Initial elastic modulus (T axis)	E_{22}^{0}
	Initial Poisson coefficient (L-T axes)	ν_{12}^{0}
	Initial shear modulus (L-T axes)	G_{12}^{0}
Damage parameters	Initial matrix microcracking threshold	Y_0
	Matrix microcracking resistance	Y_c
	Initial shear threshold for fiber/matrix	Y_0'
	Shear decohesion resistance for fiber/matrix	Y_c'
	Ultimate value for brittle failure of matrix/fiber interface	Y_r'
	Coupling factor between damages	b
	Damage ratio (= d'/d)	b'
	Loss in compressive rigidity coefficient	γ
Plasticity parameters	Initial plasticity threshold (Von Mises Criterion)	R_0
	Coupling coefficient between T-axis tensile stress / shear stress (Von Mises Criterion)	a^2
	Parameters for modeling the increase in plasticity threshold R (Von Mises Criterion) $R(p) = \beta p^{\alpha}$ or $R(p) = R_c(1-e^{-p/pc})$	α
		β
		p_c
		R_c
Failure parameters	Ultimate tensile strain – L axis	ε_{11}^{t}
	Ultimate compressive strain – L axis	ε_{11}^{c}
	Ultimate tensile stress – T axis	σ_{22}^{c}

Table 1 : DAMSTRAT material entry parameters.

DAMSTRAT uses ASCII text files as entry files. Any NASTRAN entry file can be converted into a DAMSTRAT entry file using a component of DAMSTRAT. DAMSTRAT results can be exported as op2 or pch files. DAMSTRAT calculations end when linearisation errors occur within the program.

3. MODELS STUDIED AND ANALYSIS METHODOLOGY

Validation of the DAMSTRAT software was achieved by studying six different models, each one having an increasing complexity in comparison to the preceding. The first model was a simply supported square plate. Model two was rail panel under shear load. Model three was a tension-loaded laminate with hole. The fourth model was a compression-loaded composite panel and introduced the analysis of buckling. The fifth model was a compression-loaded composite panel with hole. The last model was a blade-stiffened panel. Models 2 through 6 were taken from previous analyses realised by SLEIGHT [1]. SLEIGHT's results were both numerical and experimental.

The mechanical properties required by DAMSTRAT for a progressive failure analysis have been characterised for two materials available at CNES: M55J/M18 and T300/M18 which are both carbon fibers/epoxy matrix composites.

Each model was first modelised with PATRAN/NASTRAN and analysed using a 2D non-linear elastic modelisation. The model was built using the CNES material which had the elastic moduli and critical stresses better matching those used by SLEIGHT for the model. Afterwards, the same model was modelised with DAMSTRAT using a non-linear progressive failure modelisation. DAMSTRAT results were finally compared to those of NASTRAN and SLEIGHT (both numerical and experimental).

Since the SLEIGHT and DAMSTRAT analysis did not use the same materials, SLEIGHT's results are a qualitative element of comparison for the DAMSTRAT analysis. NASTRAN and DAMSTRAT models are built using the same CNES materials, however NASTRAN does not modelise progressive failure. Therefore, NASTRAN results were used as a quantitive comparison with DAMSTRAT for a small external load (when internal damaging remains negligible).

DAMSTRAT calculations end when linearisation errors occur within the program or if loading conditions are not reached before complete failure. Therefore, a failure criteron was chosen to detect final failure on the NASTRAN and DAMSTRAT load curves. Failure was indicated on these load curves using the Hashin failure criterion. This criterion can detect the following failure modes: fiber tension, fibre compression, matrix tension and matrix compression. Indicating these failure modes illustrates the interest of using progressive analysis failure instead of a first-ply failure model.

4. RESULTS AND DISCUSSIONS

Below are presented results for the third and fourth models. For each model, a load graph is presented and contains three curves: the curve obtained by the SLEIGHT's method (using Hashin criterion), the one obtained by NASTRAN and the one obtained using DAMSTRAT. Only a few particular points are shown on the SLEIGHT curves.

4.1 Results for a tension-loaded laminate with hole

The laminate dimensions are 203,2 mm x 25,4 mm x 2,6162 mm (8 po x 1 po x 0,103 po) with 20 plies $[\,0^o / (\pm 45^o)_3 / 90^o{}_3\,]_S$. The panel is made of T300/1034. A 6,35 mm in diameter hole (0,25 po) is located in the center of the plate. One end of the plate is clamped and all other edges are free. The NASTRAN and DAMSTRAT models were built using T300/M18 since the elastic moduli of this material were close to those of T300/1034. NASTRAN used CQUAD8 elements and DAMSTRAT used BRIQUE-16.

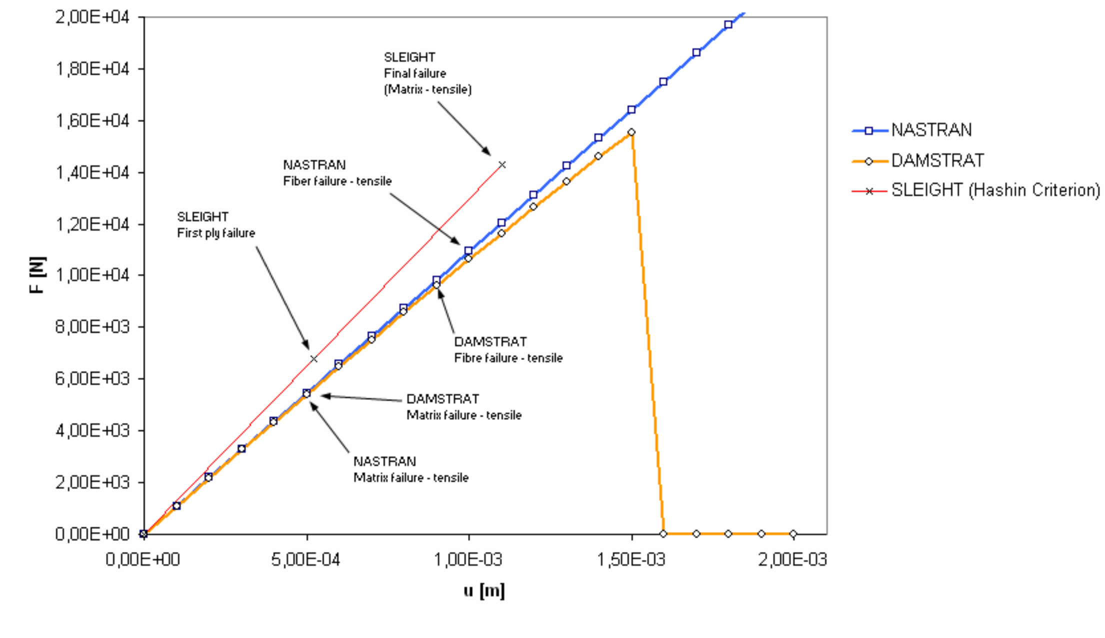


Figure 2: Tension load as a function of impose displacement for a tension-loaded laminate with hole.

Comparison with NASTRAN elastic modelisation

NASTRAN and DAMSTRAT curves are linear and equal up to an imposed displacement of about 0,5 mm. Beyond this point, the DAMSTRAT curve has a slightly decreasing slope, which corresponds well to a progressive failure damaging. The Hashin criterion detects matrix tensile failure for a 0,5 mm displacement in both the DAMSTRAT and NASTRAN models. However, Hashin detects fiber tensile failure on the DAMSTRAT before NASTRAN. This means that the DAMSTRAT progressive failure analysis can lead to lower margin of safety than NASTRAN.

Comparison with SLEIGHT progressive failure modelisation

The load curves for SLEIGHT and DAMSTRAT both exhibit the same behavior. Both curves are almost linear and their slope gradually decreases after first damage is detected.

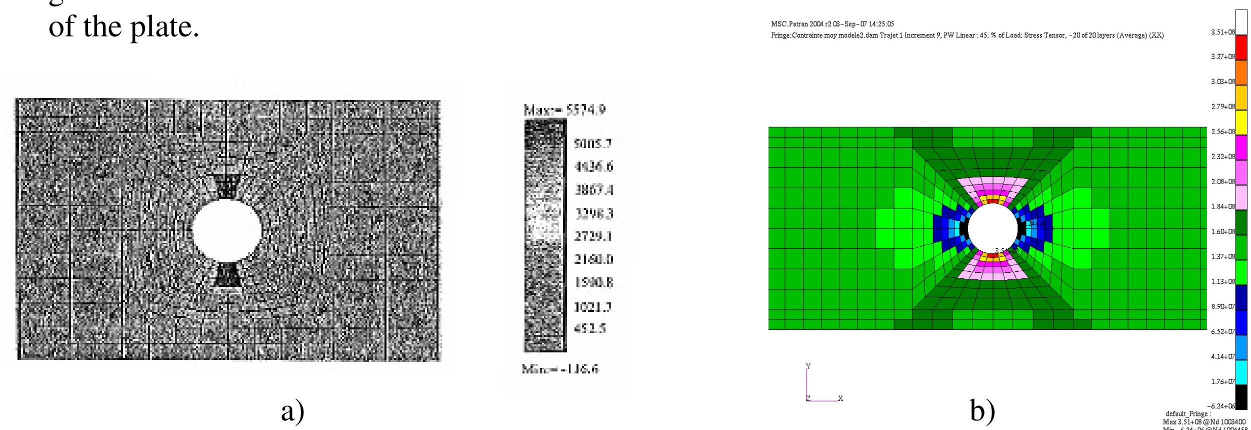
First damage occurred in the SLEIGHT model when first ply failure was detected. In the DAMSTRAT modelisation, this first damage happened when the DAMSTRAT curve stopped being confounded with the NASTRAN curve (damage then becomes non-negligible). Final failure was supposed to be reached on the DAMSTRAT load curve when fiber tensile failure was detected by the Hashin criterion.

Model	First Damage Load [N]	Final Failure Load [N]
SLEIGHT	6760	14 300
DAMSTRAT	5420	9600

Table 2: First damage and final failure loads for a tension-loaded laminate with hole using the Hashin criterion.

Both SLEIGHT's numerical results and DAMSTRAT results for first damage and final failure loads are of the same order of magnitude. Furthermore, it can graphically observed that first damage load occur in both models at about the same imposed displacement. However, SLEIGHT's analysis did not contain experimental data on first ply failure and on the dominant failure mode type.

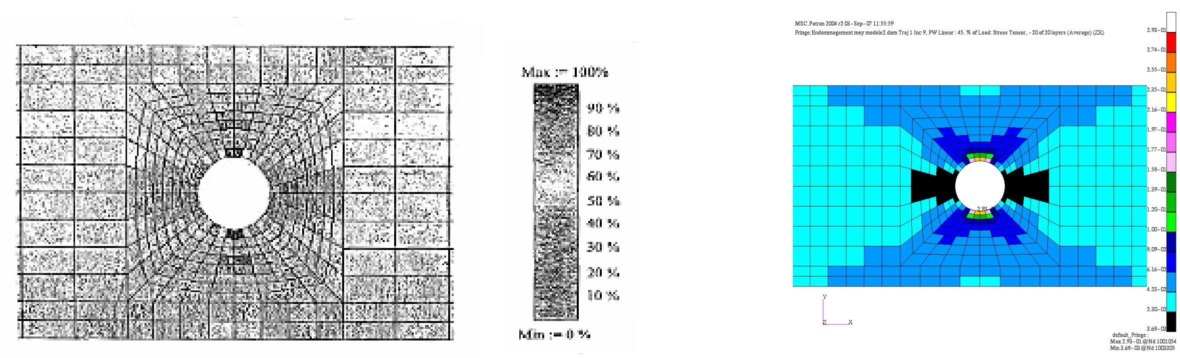
The longitudinal tensile stresses at final failure in both models also exhibit the same general behavior. The stresses are maximum on each side of the hole and uniform in most of the plate.



a) b)

Figure 3: Tensile stresses at final failure for a tension-loaded laminate with hole. a) Sleight [1]. b) DAMSTRAT

As for damage distribution, it is also maximal on both sides of the hole and uniform in most of the plate.



a) b)

Figure 4: Damage at final failure for a tension-loaded laminate with hole. a) Sleight [1]. b) d parameter - DAMSTRAT

4.2 Buckling results for a compression-loaded composite panel

This model was analysed using a method similar to the one described above. However, since this model introduced a plate buckling behavior, we shall limit our observations to this buckling behavior.

The laminate dimensions used for this analysis are 508,0 mm x 171,45 mm x 3,26 mm (20 po x 6,75 po x 0,1284 po) with 24 plies [$\pm 45^o$ / $(0^o)_2$ / $\pm 45^o$ / $(0^o)_2$ / $\pm 45^o$ / 0^o / $90^o]_S$ of T300/5208. The lateral edges are simply supported. The other edges are clamped and a longitudinal compression displacement is gradually imposed to one of these clamped edges. The NASTRAN and DAMSTRAT models were built using T300/M18, once again since the elastic moduli of this material and T300/5208 were similar. Since buckling is to be observed, NASTRAN used CQUAD4 elements and DAMSTRAT used BRIQUE-8.

An initial geometric imperfection of 0,163 mm (5 % of plate thickness) is introduced in the NASTRAN and DAMSTRAT mesh in the center of the first buckling mode in order to create specific buckling.

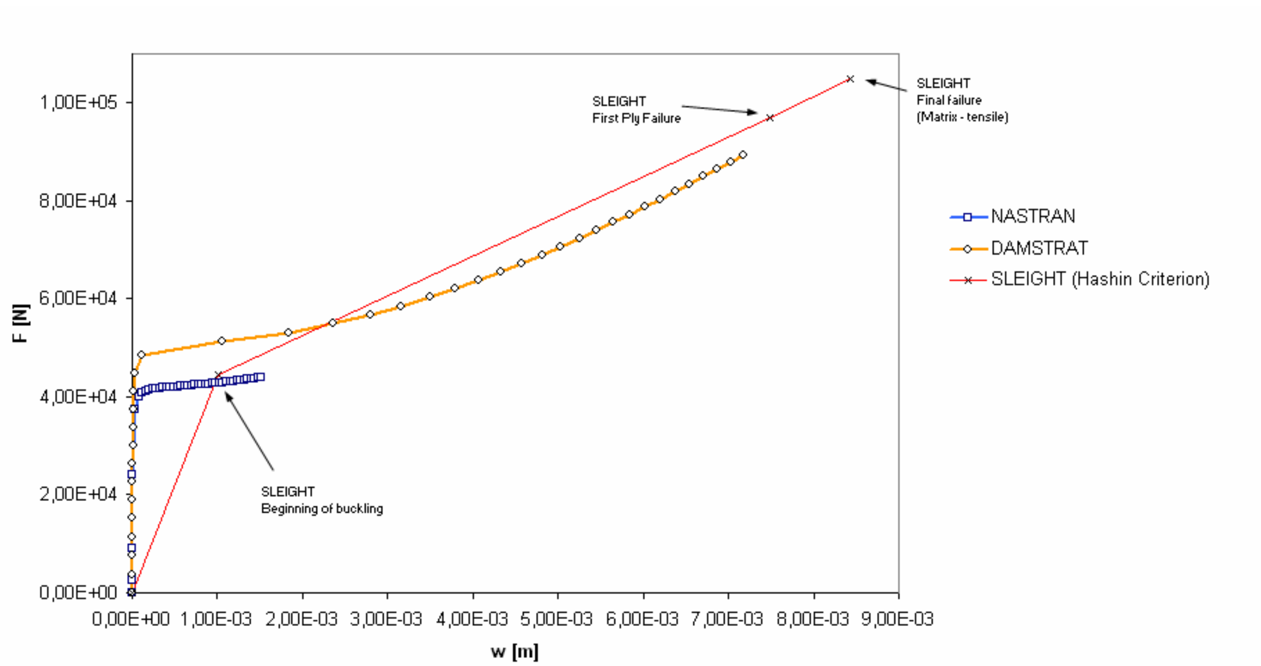


Figure 5: Out-of-plane deflection as a function of impose displacement for a compression-loaded composite panel.

Model	Buckling Load [N]
SLEIGHT	44 500
NASTRAN	40 090
DAMSTRAT	48 509

Table 3: Buckling loads for all three modelisations.

Comparison with NASTRAN elastic modelisation

Both NASTRAN and DAMSTRAT load curve exhibit the same buckling behaviour. The DAMSTRAT buckling load is however about 20 % higher than that of NASTRAN. It can also be observed that DAMSTRAT is able to continue the analysis way beyond the bifurcation point and way farther than NASTRAN.

Comparison with SLEIGHT progressive failure modelisation

The buckling loads in SLEIGHT and DAMSTRAT models have the same order of magnitude. Furthermore, both load curves have about the same slope in the post-buckling regions.

5. RECOMMENDATIONS

Twenty recommendations relative to the use of DAMSTRAT as a dimensionning tool were given following the construction and analysis of the six models. The complete list of recommendations can be found in reference [6].

6. INTEREST FOR DIMENSIONING SPACE LAUNCHERS STRUCTURES

Progressive failure analysis for designing composite material parts is of great interest for designing space structures. This type of modelisation better reflects the behavior of a structure. A composite part can see its maximum design load be increased way beyond the first ply failure and thus decrease over dimensioning. Thinner walls in composite structures result in less structural mass and therefore a greater payload can be carried by the launcher.

Progressive failure analysis does not necessarily mean higher load capacity. This type of modelisation can result in lower margins of safety than a first-ply failure analysis.

Below are presented two examples in which the DAMSTRAT tool was used for verifying industrial dimensionning cases in which using first ply failure methodology had resulted in negative or no margin of safety.

DAMSTRAT has already been used to study the progressive failure analysis of the Ariane 5 equipment bay. Failure of this equipment bay had happened during a qualification test on a ground test bench. A NASTRAN modelisation of the bay using the Tsai-Wu criterion had only been able to detect failure after addition of initial damaged in the vicinity of a hole. DAMSTRAT has however been able to detect failure in the vicinity of holes at 15 % of the maximum load in some conditions.

Many launcher structures contain radii of curvature, such as on the Ariane 5 Interstage Skirt (ISS), made of composite. These radii can be observed on square doors. A previous analysis had been conducted on the ISS using a shell modelisation on NASTRAN. This analysis yeld load margins of -20 %. A manual first-ply failure analysis had been able to raise this margin to +16 %. This example of local discontinuity at the origin of low margins of safety, is a good case where DAMSTRAT could be used in order to obtain directly a more reliable margin, closer to reality.

7. CONCLUSIONS

The preceding article presented a general description of validation procedure on the DAMSTRAT software, tool to be included in the tool box for composite structures analysis at the French Space Agency (CNES).

DAMSTRAT has been able to accurately modelise the progressive failure behavior of a laminate with hole under tensile loading. The final distribution at rupture of the longitudinal stresses and damage match those obtained by SLEIGHT [1] on a similar plate. However, the progressive failure modelisation for this model was more conservative than a NASTRAN first ply failure elastic analysis. The software was also able to correctly modelise the buckling behavior of a clamped panel under compressive load. The post-bucking DAMSTRAT analysis was pushed way beyond the results obtained with the NASTRAN software used with non linear behaviours.

A series of recommendations regarding the use of DAMSTRAT for modelizing structures has been established when the analysis were achieved.

DAMSTRAT is a still evolving software. A R&D project is actually underway and involving CNES, Medysys Air Espace, ENSICA and Thales for implementing interlaminar progressive failure in the software. Comparison of DAMSTRAT results with experimental data will also be achieved in order to complete the validation procedure. The long term goal of this evaluation project is to eventually use DAMSTRAT as a dimensioning tool at CNES for space related structures.

ACKNOWLEDGEMENTS

The work presented in this paper was achieved at CNES (Évry, France) as part of a student internship.

REFERENCES

[1] Sleigth, D. W., "Progressive Failure Analysis Methodology for Laminated Composite Structures", NASA/TP-1999-209107, 1999, 92 p.

[2] Goyal, V. K., Jaunky, N. R., Johnson, E. R, Ambur D. R., "Intralaminar and interlaminar progressive failure analyses of composite panels with circular cutouts", *Composite Structures*, 2004;64:91-105

[3] Kashtalyan, M., Soutis C., “Analysis of composite laminates with intra- and interlaminar damage”, *Progress in Aerospace Sciences*, 2005;41:152-173

[4] Macaluso, C., “Identification d’un modèle élasto-plastique endommageable pour les composites stratifiés UD”, CNES, Toulouse, France, 2006, 116 p.

[5] Khallouf, H., Salzmann, A., “Logiciel Damstrat – Manuel Théorique Version 6.4“, Medysys Air Espace, Labège, France, 2005, 177 p.

[6] Côté, M., “ Validation de l’outil DAMSTRAT pour l’étude de l’endommagement progressif dans les structures lanceurs composites“, CNES – Direction des Lanceurs, Évry, France, 2007, 187 p.