New embrittled shell element for airframe crash simulation

L. Patronelli¹, B. Langrand¹, E. Deletombe¹, E. Markiewicz² & P. Drazetic²
¹Department of solid and damage mechanics, Structural resistance and design section, ONERA-Lille, France
²Industrial and human automatic control and mechanical engineering laboratory, university of Valenciennes, France

Abstract

The paper deals with problem linked to finite element analysis of airframe crash study. Today the limit of Finite Element analysis imposes a minimum shell element size which does not represent local failure due to riveted joints. These local failures are generally critical and it leads the global ruin mode of airframe during crash. It is difficult to find a compromise between computation time (relative to size of model, size of elements, numerical power of tools) and prediction of simulation linked to thin description of geometry which influence generally initiation of crack propagation.

In this context a new embrittled shell element is developed. The main topic of this work is to set up a methodology to extract parameters for the characterisation of this new equivalent element. The feasibility of the concept is shown through an example of a punched plate in aluminium alloy. Then, it is proposed under hypothesis to extend this concept to a plate with six holes. The obtained results show interest of the method and ability to improve the representation of a model conserving reasonable computation time.

1 Introduction

The passengers safety in air transport is now considered to be a design argument which leads to the analysis of crash behaviours of airframes [1]. One of the aims of crash studies is to optimise the ratio “absorption energy versus cabin
deformation”. An airframe is composed of many parts (eg. clippers, beams, plates) assembled using riveting process. During crash test of airframe two complex failure modes are generally observed [2]. In one hand rivets are broken, in the other hand cracks propagate along lines of punching in plates [3]. These local phenomena influence the global ruin mode of the airframe. The initiation of crack propagation in plate is essentially triggered by stress/strain localisation around rivets and punchings. To improve the crash behaviour of airframes manufacturers more and more use numerical tools which are less expensive than experimental investigations [4]. Nevertheless, the considered numerical tools are still limited by computing costs which are directly dependent on the element size (explicit crash codes). For this reason it is impossible to represent the very local geometry (eg. punching of 4 mm diameter) where damage initiates during crash scenario [5]. The literature have highlighted geometrical phenomena linked to holes in sheet metal plates (principally $K_t$, $K_{t_0}$, in linear range).

Today explicit FE codes are able to represent rivet failure during airframe crash simulations. For this kind of application they propose macroscopic model (eg. non linear beam, spotweld) which do not increase the computing costs [6]. Nevertheless the use of classical shell elements is inappropriate to take a punched plates into account (no localisation in these elements due to geometric simplification). Usually users introduce very general numerical simplifications in the shell elements to cope with this inaccuracy (eg. calibration of shell thickness or material law).

In this context it is proposed to develop a new methodology the final aim of which is to create an equivalent ‘punched’ element. The constraints linked to airframe crash simulations imply that the equivalent element had rather be created on the basis of a standard shell element (4 nodes, and 1 integration point). Moreover, the choice is made not to calibrate the material law to modify the local non linear behaviour.

Today two points are considered as being essential and they constitute the main subject of the current research. The equivalent element must be able to count for the disrupted strain field around punching. These punching lead to strain localisation which necessarily trigger the cracks propagation. Then the aim is also to model the complex crack propagation mechanism which leads to the failure of the equivalent element.

A commercial FE Code Pam-Solid™ is used to undertake academic case of punching plate under loading mode. The local failure mechanism is analysed and decomposed in a sum of simple local mechanism. A methodology is presented to introduce embrittlement in equivalent element for the considered configuration. Consecutively to the whole of obtained results an algorithm of explicit FE code is presented and modified in the aim to take into consideration the presented methodology.

2 Literature

The elementary formulas in design are based on members having a constant section or a section with gradual change of contour. Such conditions, however,
are hardly ever attained throughout the highly stressed region of actual machine parts or structural members. The presence of holes, rivets, etc, in airframe results in modification of the simple stress distribution and in localisation of high stresses levels. This localisation of high stresses is known as stress concentration, measured by the stress concentration factor. In the same way it is possible to define the strain concentration factor:

$$Kt_\varepsilon = \frac{\varepsilon_{\text{max}}}{\varepsilon_{\text{nom}}} \quad \text{and} \quad Kt_{\sigma} = \frac{\sigma_{\text{max}}}{\sigma_{\text{nom}}}$$  \hspace{1cm} (1)

The subscript t, indicates that stress concentration factor is theoretical factor (i.e., based on assumptions in theory of elasticity Hooke’s law, homogeneity, etc). Stress concentration factor are obtained mathematically or experimentally by such means as photoelasticity, precision strain gage. When the experimental work is conducted with sufficient precision, agreement is obtained with well-established mathematical stress concentration factors.

When local plasticity is reached (i.e. $\sigma_{\text{max}} > \sigma_y$ or $\varepsilon_{\text{max}} > \varepsilon_y$ where $\sigma_y$ and $\varepsilon_y$ represent yield stress and yield strain characteristic of the considered material) the linear elasticity theory is no more applicable [8]. Nevertheless the values $Kt_\varepsilon^*$ and $Kt_{\sigma}^*$ can be always calculated (* for value $Kt_\varepsilon$ and $Kt_{\sigma}$ calculated in plastic range). The non-linear relations between plastic strain and plastic stress implies variations of these both factors. Classically it is observed that, $Kt_{\sigma}^*$ decreases from the $Kt_{\sigma}$ value to 1, and $Kt_\varepsilon^*$ increases from $Kt_\varepsilon$ to a finished value. Here the aim is to find a solution to represent all these phenomenon in finite element crash simulation of framework. The wholes of constraints presented in the introduction and the need to represent correctly failure phenomenon in crash tests simulations show that it is necessary to represent embrittlement in equivalent finite element.

3 Preliminary study

3.1 Case of a single punching plate

Some previous works lead to the ONERA department in Lille on riveted joint behaviour have been undertaken with plate made of 2024-T351 aluminium alloy and 7050 aluminium alloy rivet [9]. In this context, numerical analysis have been performed and Gurson damage parameters have been determined for the 2024-T351 aluminium alloy [10-18]. The using of FE code Pam-Solid™ enables to simulate case of punching plate configuration under loading, up to failure. To agree with previous works lead on Gurson damage, Finite element shell size is about 1.25 mm and crack path is described with nine elements which enable to measured significant effects of a hole on stress and strain distribution. Finally the studied configuration consist in square plate (20*20 mm) with hole (4 mm diameter) at the centre of the plate. The nodal boundaries conditions and the loading case are chosen to qualify the effects of a hole on a specific strain distribution in a plate (Figure 1). The simulation consist in a tension test, transversal displacement of the edges of plate
are forced to zero. The velocities boundaries conditions are imposed on the tip of test specimen. ‘Section force’ part defined on all the edges of the plate enables to extract normal and transversal load. The strains are measured on the cracking path in the same way than applied load, that enables to define relations between global and local behaviour of test specimen during loading mode.

The model is composed of 272 shells elements. Constitutive material laws are described with hardening law coupled to Gurson damage model. Gurson damage parameters are given by the open literature Computation time is about 142 s on HP workstation.

3.2 Results

The imposed velocity on the tip of test specimen imply a transversal reaction due to the boundaries conditions. The ratio $\alpha$ between normal and transversal load is shown on Figure 2. In the elastic range $\alpha$ is about 0.3 (in the same order than Poisson ratio). Then $\alpha$ increases to 0.5 (less than plastic contraction ratio of this material; because of hole presence). Finally, the value is about 1 during damage mechanism. Figure 3 shows the strains gradient measured on cracking path and his evolution during loading mode. Figure 4 shows the different local strain level (strain gradient) versus distance from the edge of the perforation. On this figure, it is possible to determined the value of $K_{t,\varepsilon}$ ($=K_{t,e}$ in the elastic range). For $F=14000N$, $\varepsilon_{nom} = 0.52\%$ (in first approach $\varepsilon_{nom}$ is compared to global strain level, then $\varepsilon_{nom}$ is assimilated to global strain). The using of finite element method does not enable to measured exactly $\varepsilon_{max}$ at the edge of the hole (for evident reasons of domain modelling). Nevertheless, it is possible to extrapolated the value of $\varepsilon_{max}$ on the Figure 4 ($\varepsilon_{max}=1.5\%$). The strain concentration factor is directly calculated : $K_{t,\varepsilon}=1.5/0.52 = 2.8$. The handbook of Peterson gives directly the strain concentration factor value for the studied configuration, in our case : $K_{t,\varepsilon}=2.7$ (Figure 5).

When local plasticity is reached at the edge of hole others effects linked to non linear behaviour must be considered in the calculation of concentration factor. It is proposed to determined relations $\eta_{\varepsilon} = \varepsilon_{i}/\varepsilon_{nom}$ (Figure 6, this ratio can be calculated in term of stress $\eta_{\sigma}$). These relations are calculated for the nine shells elements which describe the cracking path. Figure 7 shows the evolution of $\eta_{\varepsilon}$ (i=1) and $\eta_{\sigma}$ (i=1) versus $\varepsilon_{nom}$. The parameters $\eta_{\varepsilon}$ will be called strain gradient descriptor. In first approximation, the Figure 6 shows that functions $\eta_{\varepsilon}$ can be considerate approximately constant in elastic and plastic domain.
Figure 1: Numerical specimen

Figure 2: Evolution of ratio $\alpha$

Figure 3: Strain gradient versus time

Figure 4: Strain gradient/distance

Figure 5: Determination of concentration factor with Peterson Abacus

\[ K_{ta} = \frac{\sigma_A}{\sigma_I} = 1 + \frac{2a}{b} - \frac{\sigma_2}{\sigma_I} \]

\[ K_{tb} = \frac{\sigma_B}{\sigma_I} = \frac{\sigma_2}{\sigma_I} \left[ 1 + \frac{2}{a/b} \right] - 1 \]

Figure 6: Strain gradient descriptor

Figure 7: $\eta_i^e$ et $\eta_i^\sigma$ for $\lambda_i$=0.84 mm
3.3 Conclusion

Numerical tools can be used to measure stress and strain concentration factors. The obtained values in the elastic range confirmed numerical capabilities to well predict behaviours. Mesh sensitivity or experimental validation will be studied in future investigation.

The Local measures presented enable to extend the concentration factor in terms of stress and strain to the plastic domain. It is proposed to simplify the problem in first approximation, $\eta^p$ is considerate as being constant in elastic and plastic domain. The value obtained in plastic domain is different than first one because of material effect.

The local measures presented show a gradient decreasing from edge of the hole to the edge of the plate. It has been proposed to describe this gradient with gradient descriptor $\eta^g$. With these functions $\eta^p$ and the global strain $\varepsilon_{\text{nom}}$, the strain gradient of punching plate is completely know.

In the aim to develop an equivalent finite element it is the consideration of these local parameters which is going to lead to the introduction of embrittlement in modified shell element.

4 Methodology

4.1 Ruin mechanism description

The aim is to use the analysis of strain gradient to derive a finite element shell and introduce embrittlement. Material law is considerate as being an intrinsic data of the problem (it is not proposed to modify material law at integration point of shell element).

In the finite element analysis proposed in preliminary study the precision of strain and strain rate measurement depends on the elementary element size. These measures are extracted from nine finite elements shell along cracking path. Indeed, the local behaviour in this area influences and control global behaviour of punching plate during ruin mode. Local working sections are coupled to strain and strain rate measurement. These local sections depend of gradient shape and to the position (far or near hole) of the measure (or shell element). A determination of the local section is proposed on Figure 8.

The general idea consist in a sum of local behaviour, $F = \Sigma F_i = \Sigma k.S_i \varepsilon_i$. The Global load $F$ is interpreted has being sum of local load $F_i$ due to a working section $S_i$ (associated to the considered shell element) of a material (stiffness $k$) with imposed strain noted $\varepsilon_i$.

For all the local response $F_i$, it is proposed to create an equivalent behaviour in a layer noted $\mu_i$. The geometric characteristics of the different layers are imposed by studied configuration: $a^4a = 20^420$. A layer $\mu_i$ is characterised by his thickness $ep_i$ calculated has being: $ep_i = S_i/a$. In the preliminary study, shell elements are eliminated with Gurson damage model. Failure appears at the edge of the hole and propagated to the edge of plate. Shells elements are eliminated in
the same order. The last one to be eliminate (at the edge of plate) can be globally considerate as being loaded with $\varepsilon_{\text{nom}}$ (ie. with strain equal to $\varepsilon_{\text{glob}}$ in first approach). The strain level of the others layers can be obtained with $\varepsilon_{\text{nom}}$ and gradient descriptor $\eta^i_\varepsilon$ presented in the analysis of failure mechanism. The strain level of the layer $\mu_i$ is calculated as: $\varepsilon_i = \eta^i_\varepsilon \times \varepsilon_{\text{nom}}$.

An application of the presented description of failure mechanism can be undertaken with FE code Pam-soldT$^\text{TM}$. Here, the aim is to show feasibility of the concept. The different layers are modelled with shell element. Imposed velocities, boundaries conditions, and constitutive material law are the same than ones presented in the preliminary study. The different equivalent element parameters are given in the Table 1. As the equivalent modelling proposed is not developed in the FE code, the strain and strain rate of the different layers are imposed with velocities boundaries conditions different:

$$\varepsilon_{\text{glob}} = \frac{d(l_{\text{glob}} - l_0)}{dt} = \frac{l}{l_0} \times \frac{dl_{\text{glob}}}{dt} = \frac{V_{\text{imp}}^{\text{glob}}}{l_0}$$

$$\varepsilon_i = \frac{d(l_i - l_0)}{dt} = \frac{l}{l_0} \times \frac{dl_i}{dt} = \frac{V_{\text{imp}}^i}{l_0}$$

$$\varepsilon_i = \eta^i_\varepsilon \times \varepsilon_{\text{glob}} \quad \text{imply that} \quad \frac{V_{\text{imp}}^i}{l_0} = \eta^i_\varepsilon \times \frac{V_{\text{imp}}^{\text{glob}}}{l_0}$$

where $l_0$ is the initial length of the layers $\mu_i$. It is shown that layer $\mu_i$ can be loaded with $V_{\text{imp}}^i = \eta^i_\varepsilon \times V_{\text{imp}}^{\text{glob}}$. The transition between elastic and plastic local behaviour is a complex mechanism because these local phenomenon appear at different time during loading mode (considering the whole of layer). It is proposed to describe this mechanism considering only gradient descriptor $\eta_{\text{plast}}^i$ (the plastic strain range of this material is greater than elastic strain range). The development of this equivalent model in FE code will enable to simulate and consider the complete mechanism in the next.

Table 1. Equivalent element parameters

<table>
<thead>
<tr>
<th>$\mu_i$</th>
<th>0</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S_i$ (mm$^2$)</td>
<td>1.28</td>
<td>2.72</td>
<td>3.248</td>
<td>4.032</td>
<td>4.256</td>
<td>4.48</td>
<td>4.502</td>
<td>4.524</td>
<td>6.72</td>
</tr>
<tr>
<td>$e_p_i$ (mm)</td>
<td>0.056</td>
<td>0.120</td>
<td>0.144</td>
<td>0.178</td>
<td>0.188</td>
<td>0.198</td>
<td>0.199</td>
<td>0.2</td>
<td>0.297</td>
</tr>
<tr>
<td>$\eta_{\text{elas}}^i$</td>
<td>2.7</td>
<td>2</td>
<td>1.2</td>
<td>1.1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>$\eta_{\text{plast}}^i$</td>
<td>5.9</td>
<td>4.9</td>
<td>4</td>
<td>2.7</td>
<td>1.9</td>
<td>1.5</td>
<td>1.3</td>
<td>1.1</td>
<td>1</td>
</tr>
<tr>
<td>a (mm)</td>
<td>20</td>
<td>20</td>
<td>20</td>
<td>20</td>
<td>20</td>
<td>20</td>
<td>20</td>
<td>20</td>
<td>20</td>
</tr>
</tbody>
</table>

Two simulations are undertaken to show mesh sensitivity. In the first one the different layers are described with any shell elements (noted macro_fin). In the
second one layers are constituted of only one shell element (noted macro_dege) (Figure 9).

The responses $F_i$ are summed to obtain the global response $F$ (the global transversal responsive can also be obtained). The computation time of the model macro_fin is about 35.29 s compared to 1 s for the model macro_dege (with time step calibration to 0.55 compared to classical default value 0.9). The local response of reference modelling and macroscopic modelling are compared on the Figure 10. The Figure 11 shows comparison between normal and transverse load of the different model proposed. Two phenomenon are observed on global result. The simplification $\eta = \eta_{plast}$ influences global response of the macroscopic model. The plastic state appear more quickly and the obtained behaviour is different. When failure is obtained with the model macro_dege global strength (normal and tangential) is forced to 0. This is not the same mechanism with macro_fin model, as in the reference case the tangential stiffness imply transversal load important during damage mechanism.

Results as a whole proves validity of the presented method. An introduction of the equivalent element proposed can be envisage in a commercial explicit FE code such as Pam-Solid\textsuperscript{TM}.

---

**Figure 8**: determination of local section and methodology

**Figure 9**: local response

**Figure 10**: global response
5 Explicit algorithm modification

FE code such as Pam-SolidTM uses explicit algorithm. Velocities are interpolated in the interval \((-^{n-1/2}t, n^{-1/2}t)\), displacements and accelerations are computed in the interval \((-^{n-1/2}t, n^{-1/2}t)\). Considering the time \(n^{-1/2}t\), the finite element code computes \(n^{1-2}\sigma, F_{int}, F_{ext}, a, n^{-1/2}V\) and \(n^{1}X\) dependant of quantities \(n^{-1/2}V, nX\) and \(n^{-1}\sigma\).

**one increment of loading**

\[
\begin{align*}
F_{ext}^n &= F_{ext}^{n-1} + \Delta F_{ext}^n \\
\sigma_{ij}^n &= \sigma_{ij}^{n-1} + (n^{-1/2} t - n^{-1/2} t) \times C^e \times \varepsilon_{ij}^n \\
F_{int}^n &= \int B^T : \sigma_{ij}^n dV \\
a^n &= M^{-1} (F_{ext}^n - F_{int}^n) \\
V^{n+1/2} &= V^{n-1/2} + (n^{1/2} t - n^{-1/2} t) \times a^n \\
X^{n+1} &= X^n + (n^{1/2} t - n^{-1/2} t) \times V^{n+1/2}
\end{align*}
\]

It is proposed to compute internal forces \(F_{int}\) has being the sum of internal forces developed in the whole of layer:

\[
F_{int}^n = \sum_k F_{int}^k = \sum_k \int B^T : \sigma_{ij}^k dV
\]  
(5)

In the same way, the quantity \(\sigma_{ij}^k\) must be computed for all the layers. The relations obtained with gradient descriptor on local strain and strain rate measured (ie. \(\varepsilon_{ij}^k = \eta^k_r \cdot \varepsilon_{i,j}^{nom}\) or \(\varepsilon_{ij}^k = \eta^k_r \cdot \varepsilon_{i,j}^{nom}\)) enable to compute the value \(\sigma_{ij}^k\):

\[
\begin{align*}
\sigma_{ij}^k &= \sigma_{ij}^{n-1} + (n^{-1/2} t - n^{-1/2} t) \times C^e \times \varepsilon_{i,j}^{n-1/2} \\
&= \sigma_{ij}^{n-1} + (n^{-1} t - n^{-1} t) \times C^e \times \varepsilon_{i,j}^{n-1/2} \\
&= \sigma_{ij}^{n-1} + (n^{-1} t - n^{-1} t) \times C^e \times \varepsilon_{i,j}^{n-1/2}.
\end{align*}
\]  
(6)

The proposed equivalent modelling consist in a classical modified. The additional parameters are:

- i - Number of layers, \(k\),
- ii - Layers thickness, \(e_{pl}\),
- iii - Gradient descriptor \(\eta^k_r, \eta^k_s\).
As it was shown in the finite element modelling, the principal advantage is the calculation time.

![Modified Shell](image)

**a - equivalent modelling**

![Thin Modelling](image)

**b - thin modelling**

![Failure with New Element](image)

**c - failure with new element**

![Calibration of Thickness](image)

**d - failure with thin model**

**e - calibration of thickness**

**f - load / time for the 3 simulations**

Figure 11: Validation (numerical/numerical).

### 6 Validation (Numerical/Numerical)

#### 6.1 Modelling description

The goal of this part is to show improvement obtained with the new embrittled element. In this way three approaches are compared:

The first one consist in a thin modelling of a plate with a distribution of six holes (Figure 11-b). The second one is a classical equivalent modelling. The size of shell element do not able to represent the holes. So to take into account of hole a calibration of shell thickness is undertaken (Figure 11-a). In the last model no calibration is realised and the embrittled shell element is used in the area of holes (Figure 11-a).
The loading consist in a tension case. The using of Pam Solid\textsuperscript{TM} enable to simulate behaviour of plate up to failure. As in the part 3.1 Gurson damage model identified for 2024-T351 is used.

6.2 Results

The results are shown on the Figure 11. The non linear behaviour is better described with the new embrittled element formulation (Figure 11-f) compared to classical method (calibration of shell thickness). The comparison of the both equivalent modelling with thin model shows an improvement of the numerical failure mode obtained with the new equivalent element (Figure 11-c-d-e). The computation time decreases with the new model (thin model/new equivalent modelling = 15).

7 Conclusion

The final aim of this research is to simulate full crash scale events in airframe in case of crash. This is why explicit FE code such as Pam-Solid\textsuperscript{TM} have been used in the presented study.

The classical numerical tool do not enable to describe correctly ruin mode of airframe during crash test simulation. Computation cost imposed a minimum finite element size which imply a simplification of local geometry. More particularly, riveted joints lead to assemblies embrittlement (concentration stress factor) and influences local failure mechanism. Today the need of manufacturer is also in well characterisation of ruin mode and global strength in case of crash.

The using of finite element method enables to decrease cost of this kind of studies.

The whole of these constraints and the need of manufacturers lead to a project in which one the aim is to introduce embrittlement due to holes in a finite element shell. In the same way than previous works realised on the dynamic strength of riveted joints (collaboration between ONERA-lille and university of Valenciennes UVHC/LAMIH), academic loading case of punching plate are performed. The using of Gurson damage model enables to simulate loading case up to failure. The relations are established between local significant behaviours and global behaviour (gradient descriptor). Result as a whole are compared with literature data (stress concentration factor obtained) and have been validated in elastic range.

Finally a methodology is proposed to substitute complex local geometry in airframe (single punching plates) by equivalent element. The main proposal of this paper is to describe the methodology. The proposed equivalent element consist in a derived shell to a multi-layered shell. Parameters of the different layer are determined with reference simulation.

A finite element modelling of the proposed mechanism is undertaken with FE code Pam-Solid\textsuperscript{TM}. Some simplification must be realised for evident reasons of FE code limits. Nevertheless, the results as a whole correlate with reference
simulation ones. Today others works are in progress with material sensitive to the strain rate. Moreover some numerical improvement are undertaken to couple embrittled element with equivalent element rivet.

References


[8] Peterson, R. E. Part 3 - Engineering and design aspects, Material research & standards, pp 122-139, February 1963


