Research on the Bearing capacity of aircraft metal skin with pit damage

Luchun Zhao*, Zhiqin Zhao
Shanghai Aircraft Design and Research Institute, Shanghai, China

*Corresponding author: luchunzhao@comac.cc

Abstract. Pit is a kind of damage form that often occurs in the daily operation process of aircraft. It is formed by permanent plastic deformation due to the impact of foreign objects. In addition to the stress concentration caused by structural geometric changes, it will also produce local residual stress in the interior. Its impact on fatigue cannot be evaluated only through the stress concentration analysis of structural geometric changes. In this paper, the bearing capacity of the damaged metal skin dent is studied. By establishing the finite element model, the stress of the intact structure and the dent with different width depth ratio is compared and analysed.

Keywords: metal skin, dent, aspect ratio, stress concentration.

1. Introduction
Due to the influence of weather, season and other environmental factors, the plane often encounters the impact of hail and other external forces in the flight process, resulting in pits. Zhou Guangzhou [1], Lu Xiangyong [2] and Z J Cao [3] proposed that the length, width and maximum depth are the important parameters for the pit maintenance of Boeing and other civil aircraft. Once these parameters exceed a certain range, the pit will affect the aerodynamic shape of the aircraft, increase the flight resistance of the aircraft, and affect the performance of the aircraft. Under the loading condition, the stress concentration factor at the pit is very large, which will lead to the fatigue strength greatly reduced and pose a serious threat to the safety of flight. Therefore, it is of great practical significance to study the dent on the surface of aircraft skin and its bearing capacity. At present, in the domestic aircraft maintenance work, the maintenance engineers usually rely on visual inspection, mark, measure and record the diameter and depth of the pit by hand, and judge whether the pit needs to be repaired according to the aircraft maintenance manual standard. Based on the experience of aircraft skin damage repair, this paper studies the relationship between the dent width, depth, width depth ratio and skin stress concentration factor by Patran & Nastran software, so as to promote the optimization of maintenance scheme.

2. Establishment of finite element model
All manuscripts must be in English, also the table and figure texts, otherwise we cannot publish your paper. Please keep a second copy of your manuscript in your office.
When receiving the paper, we assume that the corresponding authors grant us the copyright to use the paper for the book or journal in question. Should authors use tables or figures from other Publications, they must ask the corresponding publishers to grant them the right to publish this material in their paper.

2.1. Introduction of Patran & Nastran software
Patran & Nastran was first developed by NASA. With high reliability and excellent quality, Patran & Nastran is the most famous parallel frame finite element pre- and post-processing and analysis system in industry. Its open and multi-functional architecture can integrate engineering design, engineering analysis, result evaluation, customization and interactive graphics. It is a complete CAE integrated environment. Patran's concurrent CAE design idea breaks the traditional pre-processing and post-processing mode of finite element analysis. Its unique DGA technology provides a perfect integrated environment for the geometric model communication between CAD / CAM software systems and the seamless connection of various analysis models. Using DGA technology, application engineers can directly access the existing CAD / CAM system database in Patran framework, read, transform, modify and operate the geometric model being designed without copying. Patran supports different geometric transmission standards, including Parasolid, ACIS, step, IGES and so on.

2.2. Finite element simulation of impact crater
In the process of impact crater simulation, Patran & NASTRAN software has the advantages of more accurate and fast calculation of nonlinear problems compared with other simulation software, because of the existence of non-linear contact between punch and matrix, non-linear material elastic-plastic deformation and non-linear large geometric deformation. Patran & NASTRAN software was used to simulate the contact process of impact crater. Material constitutive model is a necessary prerequisite for numerical simulation of impact crater and an important basis for predicting large deformation of parts. By establishing the function between stress and strain, hardening parameters and hardening index, the deformation size and the change trend of crater depth can be predicted. This paper uses MSC Nastran. The equation of state of No.06 material matd006 in sol700 module is shown in equation (1), which is used to describe the relationship between internal pressure $P$ and material density.

$$
\begin{align*}
\epsilon & = a_1 \mu + a_2 \mu^2 + a_3 \mu^3 + (b_0 + b_1 \mu + b_2 \mu^2 + b_3 \mu^3) \rho_0 e \\
(\mu > 0) & \\
\epsilon & = a_4 \mu + (b_4 + b_5 \mu) \rho_0 e \\
(\mu > 0) & 
\end{align*}
$$

(1)

Where: $\mu = \rho / \rho_0 - 1$

$\rho$ —— Material density at any time; $\rho_0$ —— material reference density;

$\epsilon$ —— The specific internal energy of the material; $a_1$, $b_i$ —— Material constant.

| Table 1. Mechanical Properties of 2524-t3 materials |
|---------------------------------------------------|
| E/GPa | $\mu$ | /MPa | /MPa |
|-------|-------|------|------|
| 71.0185 | 0.35 | 310.48 | 420.76 |
### Table 2. Relationship between yield stress and plastic strain

| Yield stress / MPa | Plastic strain/ $\varepsilon$ | Yield stress / MPa | Plastic strain/ $\varepsilon$ | Yield stress / MPa | Plastic strain/ $\varepsilon$ |
|--------------------|-------------------------------|--------------------|-------------------------------|--------------------|-------------------------------|
| 310.48             | 0                             | 326.06             | 0.045329                      | 351.73             | 0.38955                       |
| 312.09             | 0.0034373                     | 331.25             | 0.071431                      | 360.24             | 0.58775                       |
| 314.35             | 0.0085932                     | 336.38             | 0.11058                       | 372.98             | 0.88507                       |
| 317.4              | 0.016327                      | 341.11             | 0.169311                      | 392.09             | 1.331                         |
| 321.33             | 0.027928                      | 345.84             | 0.25741                       | 420.76             | 2                             |
|                    |                               |                    |                               | 420.76             | 2.1                           |

The finite element model established by Nastran explicit software is shown in Figure 1.

![Figure 1](image_url)

**Figure 1.** Impact finite element model of nonlinear contact thin plate

The punch is a spherical rigid body with radius $r$ of 2mm, 3mm and 4mm. In order to simplify the finite element model, the size of the punch is $700 \times 600 \times 3$mm. The mesh is locally refined near the impact position, and the number of elements divided by the punch is. the punch is set as a rigid body, and the aluminum alloy plate is elastic-plastic material.

Impact simulation process: explicit explicit dynamics was used for analysis, and the total simulation time was set as 0.003s to ensure that the punch completely left the surface of the test piece after impact. Contact type: the contact between punch and aluminum alloy surface is set as face to face contact, and the contact type is hard contact.

Boundary conditions: the degree of freedom in Y direction of the bottom constraint of the structural member is 0, the degree of freedom in X direction is 0, the degree of freedom in both sides of Z direction is 0, the punch retains the degree of freedom in Y direction, and the degree of freedom in the other two directions is set to 0.

### 2.3. Related parameters affecting stress concentration factor

Pit is caused by heavy objects falling or colliding with other objects. Generally, it can be assumed that it is a spherical crown, and the form of depression is represented by diameter and depth. As shown in Figure 2, $W$ is the size of depression, $D$ is the depth of depression, and $t$ is the thickness of plate. the types of pits can be defined according to the location of pits.
3. **Text Analysis of finite element calculation results**

The law of impact damage is studied by changing the impact energy of sphere at low speed (1000mm / s), medium speed (1800mm / s) and high speed (2500mm / s), the impact simulation was carried out.

3.1. **Analysis of stress and strain results**

The results show that the residual normal stress increases first and then decreases, which indicates that the distribution of residual stress near the dent is symmetrical along the center of the dent, and the trend of shear stress is the same as that of normal stress, the results are shown as Figure 4-Figure 5.
3.2. Stress concentration factor
The stress concentration factor has nothing to do with the material itself, but has something to do with the shape and loading mode of the structural parts. The stress concentration factors near the pits of thin plates with different pit size factors under axial tension are shown in Table 2. According to the data in Table 1 and figures 3 to 5, it can be seen that the stress concentration factor of the concave increases with the decrease of the width depth ratio of the concave.

Table 3. Stress concentration factors of pits under different parameters

| Serial number | Width W / mm | Depth Y / mm | Plate width / mm | Pit width depth ratio w / Y | Stress concentration factor |
|---------------|--------------|--------------|------------------|---------------------------|----------------------------|
| 1             | 70           | 2.0          | 600              | 35                        | 2.35                       |
| 2             | 72           | 2.1          | 600              | 34.28                     | 2.38                       |
| 3             | 54           | 2.2          | 600              | 24.54                     | 2.66                       |
| 4             | 51           | 2.1          | 600              | 24.28                     | 2.66                       |
| 5             | 33           | 2.0          | 600              | 16.5                      | 3.25                       |
| 6             | 35           | 2.2          | 600              | 15.9                      | 3.44                       |

4. Conclusion
This article mainly through MSC. patran finite element analysis software was used to establish the finite element model of 2524 aluminum alloy impact crater defect generation process, and the numerical simulation analysis of the stress and strain of the crater under various variables was carried out. The stress and strain distribution produced by different impact velocity was studied in detail, and the stress concentration factor of the crater under different parameters was studied, which provided the basis for the crater generation the results of fatigue impact assessment provide reliable data.

References
[1] Zhou Guangzhou. Inspection and evaluation of dimple damage of Boeing civil aircraft [J]. Civil aircraft design and research, 2013 (2): 60 - 63.
[2] Zhan Zhixin, Tong Yang, Li Binkai, et al. Fatigue life prediction of titanium alloy plates with impact defects [J]. Acta Aeronautica Sinica, 2015, 37 (7): 2200 - 2207.
[3] Bouzakis K D, Skordaris G, Mirisidis J, et al. Determination of coating residual stress alterations demonstrated in the case of annealed films and based on a FEM supported continuous simulation of the nanoindentation [J]. Surface & Coatings Technology, 2003, 174 (03): 487 - 492.
[4] Deng D. FEM prediction of welding residual stress and distortion in carbon steel considering phase transformation effects [J]. Materials and Design, 2009, 30 (2): 359 - 366.