Progressive failure analysis of compression-loaded composite flat panel with cutout

Gauspira S. Makandar, N.K. Chhapkhane, S. M. Sawant
PG Scholar¹, Associate Professor, Mechanical Dept², Professor, Mechanical Dept³,
RIT, Islampur.
Email: gausmakandar@gmail.com¹, narendra.chhapkhane@ritindia.edu², suresh.sawant@ritindia.edu³

Abstract—A progressive failure analysis approach is developed for fiber reinforced composite panel using Hashin’s criteria. The initiation of intralaminar failures are identified using Hashin’s criteria. The use of failure criteria will allow the identification of failure mode. As a result of failure the appropriate material stiffness properties will be reduced at a specific integration point according to failure types. These features are applied through built in damage model for fiber reinforced composites in a commercial finite element package ABAQUS. Hashin criteria consider four different damage initiation mechanisms: fiber tension, fiber compression, matrix tension, and matrix compression. By putting material properties and strength allowables in Hashin damage model get damage initiation failure indices and damage variables which indicates mode dependent and complete failure.

Index Terms—Progressive Failure Analysis, Hashin criteria, Damage Initiation, Damage Evolution etc.

I. INTRODUCTION

In some applications, the use of composite materials for structure can result in significant benefits on structural cost and performance. Such applications of composite materials are expected to result in additional advantages of using composites to tailor the stiffness and strength to specific 30-40% weight savings and 10-30% cost reduction compared to conventional metallic structures. But still failure prediction is large concern due to lack of availability of accepted progressive failure methodology. Consequently overweight and usually over stiff structures are repeatedly designed. Alternatively, many composite structures are fabricated using ‘design and test’ process that is extremely costly and time consuming. However unlike the conventional metallic materials, composite structures fail under several modes such as matrix cracking, fiber matrix shear failure, matrix cracking and delamination. The initiation of damage in a composite laminate occurs when a single ply or part of ply in laminate fails in any of these failure modes over a certain region of structure. Accurate determination of failure modes of failure mode and their propagation while the structure is loaded helps to devise structural features for damage containment or to define fail-safe criteria. It is therefore important to understand the damage progression in composite structures subjected to different single and multi-loading conditions.

Since most composite materials exhibit brittle failure with little or no margin safety through ductility as offered by many metals. The propagation of brittle failure mechanism in composite structures must be understood and reliable prediction analysis method need to be available. This need is full filled by the methodology called ‘Progressive failure Analysis (PFA)’. The main objective of PFA is to predict the onset and progression of damage modes such as matrix cracks, fiber-matrix shear failure, fiber failure and delamination in composite laminate under loading.

The objective of this research is to simulate the initiation and progression of laminate panel due to intralaminar and interlaminar failure and to understand the physical response of composite panel under compressive loading. A third order finite element analysis method will be utilized to model the composite panel. Progressive damage will be analyzed through the inclusion of Hashin’s criteria. The inclusion of failure criteria will identify the failure mode as well as location of failure. As a result of failure, the appropriate material stiffness properties are reduced. At the end of degradation process the applied load is incremented according to the general procedure.

II. PROGRESSIVE FAILURE ANALYSIS:

At each load step, a non-linear analysis is performed until converged solution is obtained assuming no changes in material model. Then using this equilibrium state, the stresses within each lamina are determined from the non-linear analysis solution. These stresses are then compared with material allowables and used to determine failure according to Hashin’s criteria. If lamina failure is detected as indicated by a failure criteria, the lamina properties are degraded according to particular degradation
model. Since the initial non-linear solution no longer corresponds to an equilibrium state, equilibrium of structure needs to be re-established utilizing the modified lamina properties for the failed lamina while maintaining the current load level. This iterative process of obtaining non-linear equilibrium solutions each time a local material model change is continued until no additional lamina failures are detected. The load step is then incremented until catastrophic failure of structure is detected as determine by progressive failure methodology.

Therefore, typical progressive failure analysis methods involve five key features:

1) Non-linear analysis capability is used to establish equilibrium.
2) An accurate stress recovery procedure is needed in order to establish the local lamina stress state.
3) Failure criteria is needed in order to detect local lamina failure and determine mode of failure.
4) Material degradation or damage models are needed in order to propagate the failure and establish new estimates for the local material properties.
5) A procedure to re-establish equilibrium after modifying local lamina properties is needed [5].

III. DAMAGE INITIATION

Damage initiation refers to the onset of degradation at a material point. In Abaqus the damage initiation criteria for fiber-reinforced composites are based on Hashin’s theory. These criteria consider four different damage initiation mechanisms: fiber tension, fiber compression, matrix tension, and matrix compression.

In order to predict ultimate failure of the composite part, ABAQUS must first calculate the point at which the onset of damage begins. ABAQUS uses damage initiation criteria proposed by Hashin and Rotem in 1973 and by Hashin in 1980 [12]. These are interacting failure criteria where more than one stress components have been used to evaluate the different failure modes. Failure indices for Hashin criteria are related to fiber and matrix failures and involve four failure modes. The criteria are extended to three dimensional problems where the maximum stress criteria are used for transverse normal stress component. The equations for these damage initiation criteria are listed in the following equations [12]. Tensile or compressive damage can be initiated in the fiber or the matrix when the respective F equals one.

Fiber tension:

\[ F_f^t = \left( \frac{\tilde{\sigma}_{11}}{X_f} \right)^2 + \alpha \left( \frac{\tilde{\sigma}_{22}}{X_f} \right)^2 \quad (\tilde{\sigma}_{11} < 0) \]

Matrix tension:

\[ F_m^t = \left( \frac{\tilde{\sigma}_{22}}{Y_m} \right)^2 + \left( \frac{\tilde{\sigma}_{33}}{Z_m} \right)^2 \quad (\tilde{\sigma}_{22} \geq 0) \]

Matrix compression:

\[ F_m^c = \left( \frac{\tilde{\sigma}_{22}}{Z_m} \right)^2 + \left( \frac{\tilde{\sigma}_{33}}{Y_m} \right)^2 - 1 \left( \frac{\tilde{\sigma}_{22}}{Z_m} \right)^2 \quad (\tilde{\sigma}_{22} < 0) \]

Components of an effective stress tensor \( \tilde{\sigma} \), are used to evaluate the damage initiation. The true stress, \( \sigma \), is converted into the effective stress through a tensor operation. The damage tensor, \( M \), will be modified each step of the FEA simulation based on the fiber, matrix, and shear damage variables. The equations that show these relationships are shown below. Prior to any damage initiation and evolution the damage operator, \( M \), is equal to the identity matrix, so \( \sigma' = \sigma \). Once damage initiation and evolution has occurred for at least one mode, the damage operator becomes significant in the criteria for damage initiation of other modes. The effective stress, \( \sigma' \), is intended to represent the stress acting over the damaged area that effectively resists the internal forces.

\[ \tilde{\sigma} = M \sigma \]

where \( \sigma \) is the true stress and \( M \) is the damage operator:

\[
M = \begin{bmatrix}
1 & 0 & 0 \\
0 & \frac{1}{1-d_f} & 0 \\
0 & 0 & \frac{1}{1-d_m}
\end{bmatrix} 
\]

\( d_f, d_m \) and \( d_\theta \) are internal (damage) variables that characterize fiber, matrix, and shear damage [3].

After damage has been initiated, ABAQUS will calculate the material response using a modified stiffness matrix for an orthotropic plane stress material. The modified tensor and its relationship to stress and strain can be seen in the following relationships. In the plane stress tensor, \( C_d \), it can be seen that the stiffness matrix used to calculate the stress is becoming less stiff as the damage variables increase. Also, if any of the damage variables become one, stress can no longer be supported in the respective direction because the stiffness goes to zero.

\[ \sigma = C_d \epsilon \]

\[
C_d = \begin{bmatrix}
(1-d_f)E_1 & (1-d_f)(1-d_m)d_{21}E_1 & 0 \\
(1-d_f)(1-d_m)d_{21}E_1 & (1-d_m)E_2 & 0 \\
0 & 0 & (1-d_\theta)GD
\end{bmatrix}
\]
A damage evolution model is needed to predict the material response as loading increases beyond the point of initial damage. The damage progression is characterized by diminishing the entries of the stiffness matrix. The plot shown in Figure 2, will aid in describing how the damage evolution model works. Leg OA is shows the material response prior to any damage, which shows a linear increase in stress up to point A. Point A represents the point where damage initiates. At the point of initiation the damage evolution model will begin to calculate the drop in stress as the displacement of the element increases. The stress will drop until it reaches zero, the amount of displacement required to get to this point is defined by the fracture energy, \( G^c \). The fracture energy respect the area under curve OAC. Therefore, if the strength properties remain constant as fracture energy is increased, the element will experience a larger displacement before it reaches the point of ultimate failure.

Damage in the element will increase until it becomes fully damaged, this will occur when the damage variable is equal to one. Once the element is fully damaged it can no longer support any load. The material response of a damaging element can be seen in leg AC of Figure 2. If the model is unloaded before damage reaches 100 percent it will unload linearly to zero stress and displacement. The unloaded element now has a reduced strength and stiffness. Therefore, it will have a different behavior if it is reloaded. This loading cycle is shown as OBC in Figure 2.

**IV. NUMERICAL ANALYSIS:**

To assess the predictive capability of the present failure analysis method, one flat panel with cutout subjected to axial compression have been analyzed. Cutout is located at center with diameter having ratio with width of panel, d/w=0.2. The dimensions are 210mmx210mm and boundary conditions are shown in Figure 2. The panel has a laminate stacking sequence of [±45/0/90]3s, with a measured ply thickness of 0.134mm. The mechanical properties and strength allowables of the panel material used are given in following table. Three integration points through each ply thickness are used in the analysis for the computation of section properties.

<table>
<thead>
<tr>
<th>( E_{11} )</th>
<th>( E_{22} )</th>
<th>( G_{12}, G_{13} )</th>
<th>( G_{23} )</th>
<th>( \gamma_{12} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>105280MPa</td>
<td>6298MPa</td>
<td>2113MPa</td>
<td>2300MPa</td>
<td>0.2</td>
</tr>
<tr>
<td>a</td>
<td>a</td>
<td>a</td>
<td>a</td>
<td>4</td>
</tr>
</tbody>
</table>

Table 1: Mechanical Properties of Graphite- Epoxy.

<table>
<thead>
<tr>
<th>( X_1 )</th>
<th>( X_c )</th>
<th>( Y_1 )</th>
<th>( Y_c )</th>
<th>( S_L )</th>
<th>( S_T )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1315MPa</td>
<td>483MPa</td>
<td>37MPa</td>
<td>145MPa</td>
<td>83.4MPa</td>
<td>83.4MPa</td>
</tr>
</tbody>
</table>

Table 2: Intralaminar Strength Allowables of Graphite-Epoxy.

The panel finite element model consists of 6561 nodes and 6400 shear deformable S4R shell elements [14]. To know how the Hashin model works within FEA, a select a panel as shown in figure. The damage initiation and evolution variables were observed at various displacements from 0.5mm to 4mm. The first mode failure i.e. fiber compressive failure is occurring at displacement 0.5mm, is the point at which the Hashin damage initiation criteria first reaches one and marks the onset of damage evolution. This concept is seen in the contour plots shown by Figure 4. The plot of the initiation damage variable shows that damage should initiate at side edges of cutouts. It is also seen that the element reading one for damage initiation is the first element to hold a value for the damage evolution variable.

The displacement is incremented with 1mm for every run and failure indices and damage evolution variables are noted. The next point of interest occurs at 3mm. This is the point at which all mode failure initiates and give damage variable other than zero. It is seen that much of the laminate has reached an initiation value of one. As expected this corresponds to larger damage evolution variables. As the magnitude of the damage evolution variables increase, the load carrying abilities of the receptive elements decrease. The degradation of material properties on an elemental basis causes the nonlinearity because of the localized weakening of the part. To illustrate how damage progresses through the nonlinear portion of the curve contour plots of the damage variables at 4mm displacement are shown in Figure 5. As expected, the area afflicted by damage initiation increases and the magnitude of the evolution variable become nearly one.
V. CONCLUSION

A built in damage model for fiber reinforced composites is used to simulate failure analysis of laminated composite panel. Results are obtained at various loads and damage initiation variable and damage evolution variables are noted.

Load is increased at every run by 10N. Output field variables of Hashin damage model indicates damage with particular
damage mode along with particular damage evolution variable. When damage initiation variable of one is equal to one, it indicates onset of damage with that particular mode and gives some value of damage evolution variable which is less than one. When an element reaches a damage evolution value to one it can no longer carry any load. At peak load damage evolution variable for matrix compressive damage becomes one, which shows stress can no longer be supported in the respective direction because the stiffness goes to zero.

REFERENCES:


