Journal of Aeronautics & Aerospace Engineering

Gangadharan et al., J Aeronaut Aerospace Eng 2016, 5:4
DOI: 10.4172/2168-9792.1000173

Research Article

Open Access

Impact Analysis of Composite Repair Patches of Different Shapes at Low Velocities for Aircraft Composite Structures

Gangadharan S[1]*, Baliga SV[2], Sonawane NH[2], Sathyanarayan P[2] and Shruti Kamdar[3]

[1]*Department of Mechanical Engineering, Embry-Riddle Aeronautical University, 600 South Clyde Morris Blvd, Daytona Beach, Florida 32114, USA*
[2]*Department of Aerospace Engineering, Embry-Riddle Aeronautical University, 600 South Clyde Morris Blvd, Daytona Beach, Florida 32114, USA*
[3]*Department of Design Engineering, EGA, Honda Engineering North America, 640 Colemans Crossing Blvd, Marysville, OH 43040, USA*

Abstract

The area under crack for various aircraft composite structures can be effectively repaired using composite materials. Low velocity impact can cause barely visible damage to the interior structure of laminated composite. These impacts can cause delamination in composite materials. In this study, a finite element analysis was conducted using Abaqus/Explicit and the results of the analysis were compared to the experimental data from literature. E-glass/epoxy composite laminate was subjected to a low velocity impact test. To study the effect of patch repair, a composite patch was applied on a cracked laminate and a low velocity impact was then conducted on this model. The FEA results were validated with the experimental data and an approach to model an ideal composite patch shape was conducted. Different patch shapes like square, rectangle, circle and ellipse were designed and analysed on the crack by keeping the surface area of the patch common. All these patches were compared and an ideal patch shape was found for the model based on stress concentration on the patch. Finally, a parametric study was performed considering the change in impactor speed and impactor material on impact damage. The effectiveness of finite element analysis of low velocity impact on aircraft composite structures is demonstrated.

Keywords: Aircraft; Composite structures; Laminate

Nomenclature

E = Modulus of Elasticity

F = Force

G = Shear Modulus

ν = Poisson's Ratio

ρ = Density

σ = Normal Stress

σM = Von Mises Stress

τ_{max} = Maximum Shear Stress

σ 1 = Principal Stress

V_f = Fraction by Volume

V_m = Fraction by Weight

Introduction

High strength and high stiffness fiber-reinforced materials like glass/epoxy and carbon/epoxy are significantly used in the aerospace industry and material industry. They are highly flexible and have low elastic modulus. Due to low weight and low coefficient of thermal expansion these composite materials are used substantially. However, one of the biggest concern is that such structures are prone to impact loading while handling loads or when the loads are dropped. Serious damages may be caused by failure as a result of impact in composite structures in a variety of ways. It may cause delamination, matrix cracking or fiber breakage of the material. Low to moderate energies caused typically by impact forms delamination, cracking and fiber breakage. Penetration and shear damage at an excessive amount is caused by high impact energies Abrate [1]. The strength and stiffness of the damaged object, the stress state on the damage and the response of the damaged structure makes the problem complex.

It is a known fact that composite structures after impact can endure a major decrease in tensile strength and compressive strength Sierakowski Robert and Chaturvedi [2]. To study and analyse the damage on a composite structure, several experiments have been conducted. Such experiments are conducted by replicating the real-life situations in controlled environment. For instance, drop weight test is conducted to simulate the dropping of hard tools on composites. This test is generally low-velocity impact test. Damage because of low velocity impact on fiber reinforced composites is thought to be very risky for the most part, in light of the fact that the damage is not detectable to the exposed eye; this kind of damage is called as Barely Visible Impact Damage. A composite's compressive strength can undergo a loss of about 60% with this type of damage.

All in all, there are numerous parameters which characterize the way of the damage in composite structures, for example, delamination in composite structure, caused due to pressure loads. Different parameters which characterize the morphology of the impact incorporates impactor speed, geometric imperatives connected to the framework, impactor shape, and design of the affected structure. In this manner

---

***Corresponding author:** Gangadharan S, Professor, Department of Mechanical Engineering, Embry-Riddle Aeronautical University, 600 South Clyde Morris Blvd, Daytona Beach, FL 32114,USA, Tel: +1 386-226-6100; E-mail: sathya@erau.edu

Received July 25, 2016; **Accepted** October 26, 2016; **Published** October 31, 2016

Citation: Gangadharan S, Baliga SV, Sonawane NH, Sathyanarayan P, Kamdar S (2016) Impact Analysis of Composite Repair Patches of Different Shapes at Low Velocities for Aircraft Composite Structures. J Aeronaut Aerospace Eng 5: 173. doi: 10.4172/2168-9792.1000173

investigations of these parameters are critical in comprehension to the effect procedure and the damage brought on by them in the composite structures.

The damage caused by low velocity impact is inevitable. Hence, a repair or fortification of the damaged portion of the structure to restore the basic structural strength and efficiency is required. Applying composite patch repair is one of the latest solutions. Little research into the combined low-velocity impact damage resistance of the patch is available in published literature. The potential for an outwardly unnoticeable mix of the composite damage with likely adhesive damage recommends that low-velocity effect damage in composite repair is ought to be studied about and considered amid design configuration. It is costly and quite complex to conduct and perform physical experiments to evaluate impact damage on composite patches considering the quantity of distinctive parameters to be viewed and internal damages to be examined. Finite Element Analysis (FEA) gives a more financially savvy approach to foresee and survey damage in composite patches, and giving a road to investigate numerous material mixes and designs. FEA can then show the areas where constrained trial testing may be important for acceptance of the damage behaviour as reported by Goodmiller [3]. Patch shape, properties of materials, thickness, orientation, and number of plies in the composite structure, quality of the bond surface, and damage tolerance properties of materials are some of the parameters that are important to feed in for the impact performance of the structure. The mechanics of the damage of the patch is also imperative to study the analysis of the patch performance. To have an appropriate and optimized patch design, it is important to understand the effects of input parameters, damage mechanics, and their interactions.

The aim of this research was to conduct a FEA which studied the damage mechanisms related to a composite patch performance on E-glass/epoxy material under low-velocity impact loading. The results from this analysis and simulation was compared to available experimental data in quantitative terms of stress, energy, displacement and contact force. Abaqus 6.13 was used for this research, which provided modules for composite structures and adhesive properties. Composite patch performance has limited availability of experimental data. Due to that and also because of a few obscure properties of materials needed for damage models, several assumptions were made. These includes assumptions of material strength, adhesive thickness and its properties. In addition to analysis of the patch, the parameters were studied to obtain an optimum composite patch shape for impact damage resistance based upon the stress carrying capacity. Other potential factors such as number of plies and its orientation, patch size, adhesive type, and thermal expansion mismatch were not examined in this study, but should be investigated in future work.

Procedure

Experimental setup

Geofrey and Yatin [4] conducted low velocity impact experimentally. Experimental data from this study was used as reference for FEA analysis in this research. Geofrey et al. conducted a drop weight test to simulate low velocity impact on an E-glass/epoxy composite laminate. The experimental setup had nine layers of E-glass/epoxy laminates with alternating 0° and 90° plies. The dimension of the laminate was 100 mm × 100 mm and its total thickness was 4.04 mm. This plate was subjected to an impact of 20 J under the velocity of 4.472 m/s. E glass fabric, type C of IS: 11273 were used to fabricate composite laminates. An epoxy matrix based on Lapox L-12 resin and K-5 hardener was selected for making composite panels.

In the next step of the experiment, a cracked laminate was applied a composite patch upon it. The crack was deep up to the third layer of the composite ply while the crack dimensions were varied. The crack dimension for the first case was 5 mm × 5 mm and for the second case it was 5 mm × 7.5 mm. This composite patch had an orientation of 90°. The dimension of the patch used was 10 mm × 10 mm and the thickness of the patch was 1 mm.

Modeling and analysis

Finite element method is a numerical technique that is used to find solutions to a large level and variety of engineering problems which includes stress analysis in dynamic conditions. The three basic steps to perform finite element analysis are, pre-processing, solving and post-processing. In pre-processing, geometric models are made as per the requirement. The modelled geometry is then applied with appropriate meshing. Material properties are assigned to the elements and boundary constraints are applied to the nodes of the element. The next step involves, solving which is the processing of geometric data. After the data is processed the output file is generated.

The third step is post-processing which involves studying the obtained data in the form of stress, strain and force graphs. In this research Abaqus serves as both, pre-processor and post-processor. Abaqus is an interactive 3D modeling software that can be used to model many complex and simple components in engineering. Since, it has very user friendly tool interface and extensive customizing capacity; it is used on a large scale for modeling. Solving and post-processing both the jobs are done in this software. Abaqus software has explicit and implicit finite element program that is used to analyze the responses that are non-linear and dynamic. It has a fully automatic definition of contact areas and a large library of constitutive material models and failure models.

A finite element model of a symmetric, cross ply, laminated composite and impactors were modeled in Abaqus design module. The finite element model consisted of nine separate layers with each layer being 0.44 mm thick and 100 mm × 100 mm in dimension. The orientation of these layers was [0/90/0/90/0/90/0/90/0]. Every layer was attached to each other with a cohesive layer between them having a thickness of 0.1 mm. The total thickness of the composite structure was 4.04 mm. These plies were modeled with SC8R: 8 nodes, quadrilateral, reduced integration, continuum shell element. It had enhanced hourglass control with Hashin damage viscous stabilization factor of 1×10^{-7}.

The material that was modelled was E-glass/epoxy. The material properties of the E-glass/epoxy used in this test is shown in Table 1. Elements are 0.5 mm × 0.5 mm in the center of the mesh and their size increases with the distance from the impact zone. The adhesive layer between every ply is of 0.1 mm thick.

An impactor was modeled, providing impact energy of 20 J and velocity of 4.472 m/s. A friction penalty of 0.5 was provided for the contact between the impactor and the composite layer (Figure 1).

The second part of the test involved creating crack in the composite layer and a patch for the crack. This involved two tests with crack of thickness 1.34 mm and varying thickness. The first composite was modeled with 5 mm × 5 mm crack dimension and the second composite was modeled with 7.5 mm × 5 mm crack dimension. A patch

Property	Units	Value
X_c	Mpa	800
Y_t	Mpa	40
Y_c	Mpa	145
S_L	Mpa	73
S_T	Mpa	54.8
ε_{1t}	%	2.807
ε_{1c}	%	1.754
ε_{2t}	%	0.246
ε_{2c}	%	1.2
G^t_f	N/mm	17.965
G^c_f	N/mm	7.016
G^t_m	N/mm	0.049
G^c_m	N/mm	0.87

Table 1: Properties of E-glass/epoxy.

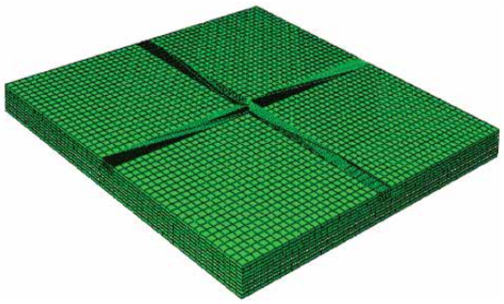

Figure 1: Modeling of the composite laminate.

was modeled for both the conditions. This patch was made of the same E-glass/epoxy element with a single layer having orientation of 90°. The thickness of this patch was modeled to 1 mm and other dimensions were 10 mm × 10 mm. The patch was attached to the composite using the cohesive layer. Both these models were validated comparing with the experimental results and the shape of the patch was changed as per the stress concentration to provide with an ideal shape.

Sensitivity study

A sensitivity study was performed to obtain a good mesh. Meshes that are good enough are ones that produce results with an acceptable level of accuracy, assuming that all other inputs to the model are accurate. Mesh density is a significant metric used to control accuracy (element type and shape also affect accuracy). Assuming no singularities are present, a high-density mesh will produce results with high accuracy (Figure 2).

Validation results

A comparison of experimental results and finite element analysis was done. Both the results showed a good agreement in between the two (Table 2).

The above results were a comparison for nine layer composite laminate without the patch. The experimental tests conducted with the patch also showed good accordance with the finite element test results (Table 3).

Following is the comparison of both the approaches for 5 mm × 5 mm crack on the composite:

Other laminate had a crack of 5 mm × 7.5 mm (Table 4). The results of these laminates are as shown below:

Towards ideal repair patch

The patch shape matters a lot when it comes to repairing of the material. The amount of stress concentration changes with the change of shape of any material. For instance, a shape with more cornered edges may have higher stress concentration when compared to the ones with lesser or no edges. This is good enough to know that the patch shape used in the experimental test may not be an ideal one. To have a better patch shape for the crack, different shapes of nearly same areas were modeled and analyzed. The experimental test which was taken into consideration was the one with the crack length of 5 mm × 7.5 mm (Table 5).

Results and Discussion

Rectangular, circular and elliptical patches were created. Following are the result comparison of all the patches (Table 6).

It is shown in the table above that the maximum displacement is

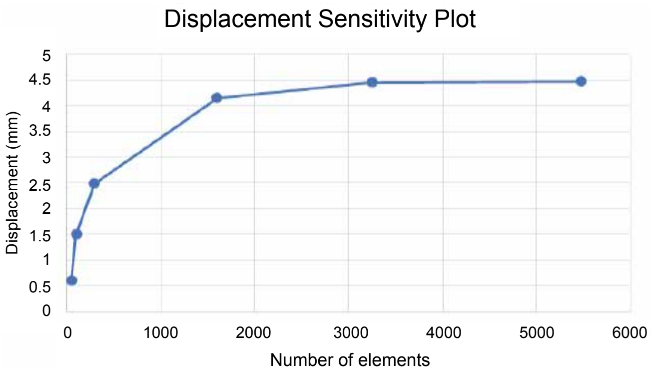

Figure 2: Displacement sensitivity plot.

Parameter	Contact Force	Maximum Displacement	Von Mises Stress
Experimental	5170.4 N	6.283 mm	54.98 MPa
FEA	5468 N	4.472 mm	55.88 MPa
% Difference	5.44	28.82	5.43

Table 2: Comparison of results for composite laminate.

Parameter	Contact Force	Maximum Displacement	Von Mises Stress
Experimental	1097 N	1.42 mm	78.53 MPa
FEA	1579 N	1.20 mm	75.95 MPa
% Difference	30.52	15.49	3.28

Table 3: Comparison of results for the first patch.

Parameter	Contact Force	Maximum Displacement	Von Mises Stress
Experimental	3732 N	0.79 mm	38.42 MPa
FEA	4294 N	0.689 mm	37.75 MPa
% Difference	13.08	12.78	1.74

Table 4: Comparison of results for the second patch.

Shapes	Area (mm²)
Square	100
Rectangle	100.5
Circle	100.1
Ellipse	100.2

Table 5: Areas of different patch shapes.

Patch Shape	Maximum Displacement (mm)
Square	0.689
Rectangle	0.448
Circle	0.447
Ellipse	0.447

Table 6: Maximum displacement comparison.

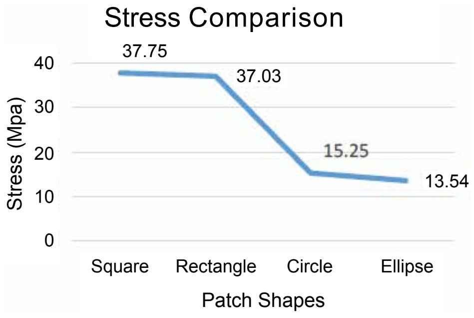

Figure 3: Stress comparison.

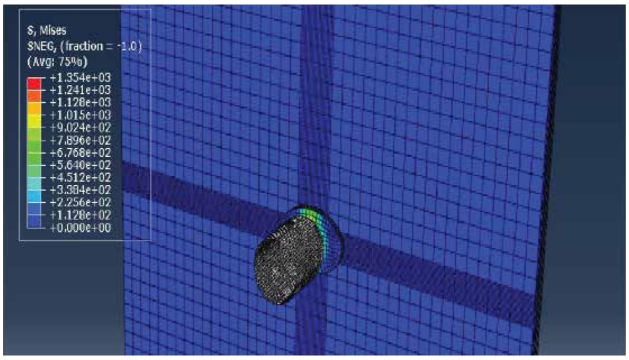

Figure 4: Von Mises stress for ellipical patch.

more in the square patch. The maximum displacement decreases for the remaining patches. The maximum displacement is more or less equal for rectangle, circle and elliptical patches. The figure below shows the stress comparison of all the patches. Elliptical patch shape has the lowest stress of all the patch shapes which is 13.54 MPa (Figures 3 and 4).

Parametric Study

A parametric study was done to understand the effect of uncertain inputs with existing boundary conditions and geometry. Impactor material and impactor velocity are the two topics included in this parametric study. For all the simulations, the thickness of the composite was kept constant throughout the process.

Steel and aluminum projectile were used for parametric study of impactor material. The impactor diameter and velocity was kept the same as that used in the tests. This study was specifically to see the effect of changing material of the impactor on the impact damage. There were differences observed in the impact force with the change in materials. Having the same impact velocity, aluminum and steel had the kinetic energy in a similar ratio. The maximum impact force of steel was found to be 1000.5 N and that of aluminum was 912 N. The figure also shows a plot of impact force vs. time for both aluminum and steel impactor materials.

The next parametric study was conducted for the change in velocity of the impactor. The impactor given in the experimental setup is used the way it is. The velocity is given as 5 m/s and 6 m/s. The maximum damage is high as compared to that of the velocity used in the experiment. Since the velocity is more, the impact damage would be greater too. Impact force of approximately 1500 MPa was calculated for the impactor speed 5 m/s and impact force approximately 1200 MPa was calculated for impactor speed at 6 m/s.

Conclusion

To simulate low velocity impact scenario on a composite material Finite element method can be effectively used. A Finite Element Analysis model of E-glass/epoxy and impactor were successfully modeled and developed to analyze their behaviour during low-velocity impact analysis. The results from the FEM simulations matches and are in good accordance with the experimental data.

The ideal patch shape analysis was done. Keeping the surface area of all the patches as constant. All the different patch shape geometries were compared to each other on the basis of stress concentration. Elliptical patch shape had stress value of 13.54 MPa and displacement 0.447 mm. It was evident from the results that elliptical patch shape is the ideal patch for the model. The stress concentration on the elliptical patch shape was the least as compared to the other patch shape geometries. Also, it is proved that square patch is not the ideal one. After the analysis of the patch, the model was subjected to parametric studies. To understand the difference obtained by change in the material nature of the impactor on the impact damage, two different type materials were used. Aluminum and steel were used as the impact material on the composite for the parametric study. It was found that the impact energy due to aluminum as well as steel impactor increases with time at a similar constant ratio. The impact force was highest for the steel impactor giving 1000.5 N while that for the aluminium impactor was 912 N. The change in velocity of the impactor was also checked in the parametric study. The experimental tests had velocity of the impactor as 4.472 m/s. The increase in velocity of the impactor for the 8 parametric studies gave high values for the maximum impact force. The damage caused by both these velocities gave excessive distortion for the laminate.

Acknowledgement

The authors of this paper would like to thank Embry-Riddle Aeronautical University, for providing the resources for this research.

References

1. Abrate S (1998) Impact on composite structures. Cambridge University Press.
2. Sierakowski-Robert L, Chaturvedi SK (1997) Dynamic loading and characterization of fiber-reinforced composites. John Wiley & Sons Inc., NJ, USA.
3. Goodmiller G (2013) Investigation of composite patch performance under low-velocity impact loading Knoxville.
4. Geofrey R, Yatin B (2010) Low velocity impact analysis on composite patch 16th Conference for Material Conference of Russia.