European Rotorcraft Forum

# INVESTIGATION OF STRUCTURAL DESIGN PARAMETERS FOR INTERCHANGEABLE COMPOSITE ROTOR BLADE TIPS

Ilyas Toprak, Betul Pelin Ergul, Gokhan Tursun
itoprak@tai.com.tr, bpmaradit@tai.com.tr, gtursun@tai.com.tr

Turkish Aerospace Industries, Inc. (TAI),
Integrated Helicopter Systems
Ankara, Turkey

**Abstract**

It is a well-known fact that rotor blades have significant influence on the vibration and noise performance of helicopters. These characteristics of blades are mostly altered by tip geometry, since it affects tip vortices and tip vibration. When a special blade tip is designed, extensive tests should be carried out. Yet; tips are inseparable parts in the most of the rotor blade designs, and testing a new blade tip necessitates manufacturing of whole blade -and hence a whole set of dies- which are extremely expensive and time consuming. Therefore, usage of interchangeable tips with different geometry significantly reduces tip test costs, since specimen manufacturing process is restricted to the tip part only. In the scope of this study, a guideline for replaceable rotor blade tips is established in three stages which are design, analysis and test. In the first stage, interchangeable blade tips and their attachment interfaces are designed in order to reduce specimen preparation expenses and time in blade tip performance tests. Subsequently, designed tip and its attachment points are simplified and in this manner test specimens are manufactured, then they are tested with a simplified test setup. Later, finite element analyses of the test specimens are performed and the composite model is optimized in the light of the strength results from the tests. By this means, structural design parameters are determined and a guideline on blade tip design, test and analysis is built.

## 1. INTRODUCTION

Vibration and noise are the crucial issues for rotorcrafts and rotor blades play important role in the generation of noise and vibration. Tip geometry of the rotor blades have influence on tip vortices and tip vibration. In addition, blade tips are appropriate locations for installing extra weights that are utilized for improving dynamic stability and reducing stresses.

For a new rotor blade design, several tests should be performed in order to verify its structural and aeromechanical performance. Similarly when a special blade tip is designed, extensive tests should be carried out. On the other hand, usage of interchangeable tips with different geometry significantly reduces tip test costs, since specimen manufacturing process is restricted to the tip part only. Within this context; an investigation on interchangeable composite tip design and analysis is carried out. Composite blade tips are preferable against their metal alternatives as they have low density and high fatigue resistance. However, their design and modeling necessitates paying extra attention on material model due to composites complex failure behavior and highly anisotropic nature [1, 5]. For instance; composites stress-strain behavior is highly depend on their degradation during loading. Therefore, successful prediction of mechanical behavior depends on appropriate choice of failure criteria (ex. Tsai-Wu, Puck, Hashin, maximum stress, maximum strain) and degradation method (ex. gradual or immediate failure). As a result, primary knowledge on failure behavior is necessary to have an understanding about design and analysis of composite blade tips.

In the scope of this study, a guideline for replaceable rotor blade tips is established in three stages which are design, analysis and test. Rotor blades are subjected to very high and complex aerodynamic loading, yet these are negligible in the tip region. The main loading in the tip region is centrifugal force which is significantly high due to tip weights. As a first step, three different blade tips for a rotorcraft having 3m rotor radius, are designed in the consideration of the primary design criteria that is centrifugal forces due to the tip weight. The design of the attachment of the blade tips is similar in all cases and in this study the Anhedral-Tapered blade tip design is evaluated. In the second part of the study, various finite element analyses of the blade tips are conducted.

In the last stage, static tests are conducted to gather information about bearing stresses and ultimate strength of the mechanism. During the test, tensile load was applied in order to simulate centrifugal forces. Instant load, bearing stress and ultimate failure data are obtained. Besides, simulation results are compared with the tests in order to optimize material model.

To sum up; in this study interchangeable blade tips and their attachment interface are designed in order to reduce specimen preparation expenses and time in blade tip performance tests. Subsequently, a designed tip and its attachment points are simplified and in this manner test specimens are manufactured, then they are tested with a basic test setup.

Furthermore finite element analyses of the test specimens are conducted and the composite model is optimized in the light of the strength results from the tests. By this means, structural design parameters are determined and a guideline on blade tip design, test and analysis is built.

## 2. BLADE TIP DESIGN

A composite blade is designed for an unmanned rotary air vehicle having 3 m. rotor radius. Tip geometries are designed in order to assess the effect of blade tips on the vibration and noise performance of the rotorcraft. Full scale tests will be conducted in Whirl Tower test facility after freezing structural design of the blade and interchangeable tips. Three different tip geometries are studied by aerodynamically and all tips are connected to the blade with same way. In this work, Anhedral-Tapered blade tips are concentrated on structural design point of view.

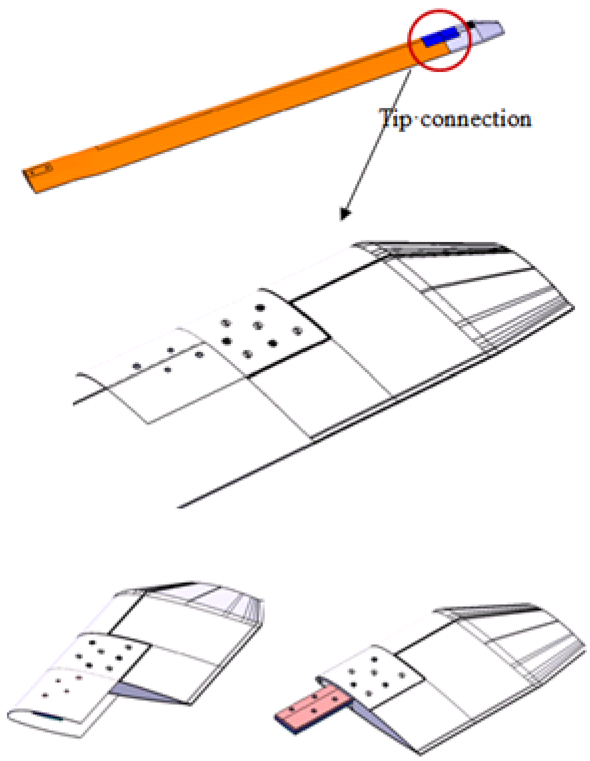


**Figure 1: Design of Anhedral-Tapered tip section**

The composite blade and its Anhedral-Tapered tip section are shown in Figure 1. Tip and blade are connected together with blind nut and screw [1]. Screw and blind nut bring together composite skin, erosion shield and tungsten weight.

In order to simplify the tension test, the test specimen is considered as a rectangular cross-section simulating the complex geometry of the blade tip. Test specimen assembly is shown in Figure 2.

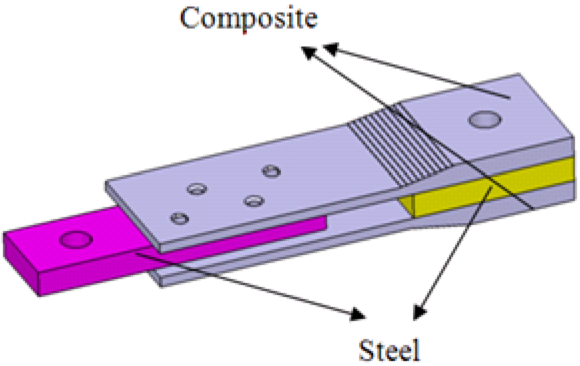


**Figure 2: Test Specimen**

There is an erosion shield in the leading edge of the blade however it’s neglected in this test. Composite skin layer is same with original blade tip. Layup consists of carbon and glass layers and total thickness of the composite layup is 4,46mm. In order to prevent a failure in the vicinity of Loading Pin 1 of composite part, this section is reinforced with additional glass layers (see drop-off region in Figure 2 and 6)

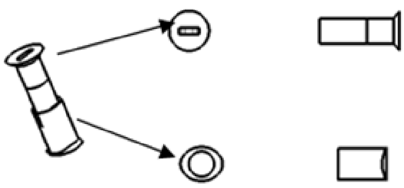

**Figure 3: Standard parts for attachment of tips**

The standard parts are utilized while attaching the tip to the blade (see Figure 3).

## 3. EXPERIMENTS

### 3.1 Instrumentation

The static tension test is conducted with MTS test system which has a 100 kN capacity. The tension test is a force controlled test via test speed 20 kg/s. In Figure 4, test-set up for tension static test of the specimen is shown.


**Figure 4: Test Set-up**

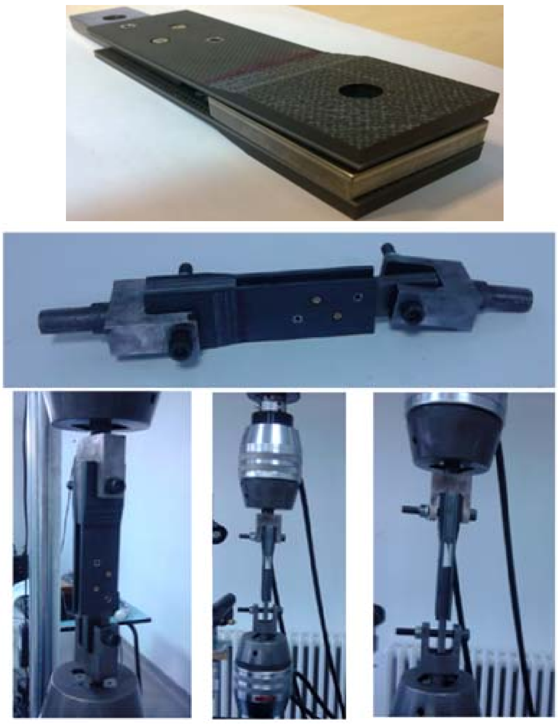

**Figure 5: Manufactured test specimen**

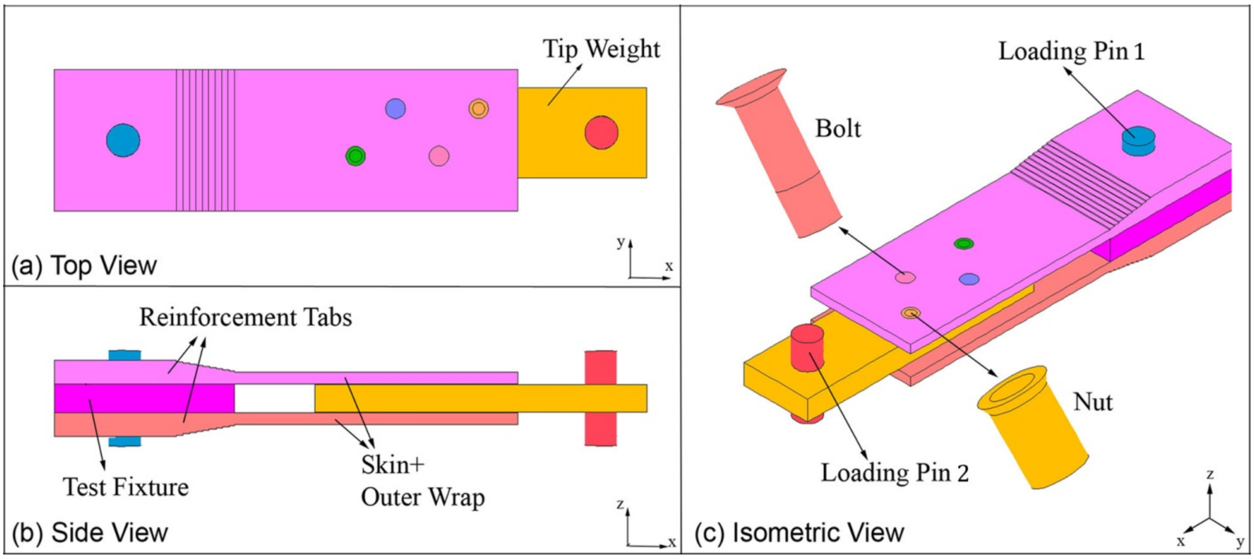


**Figure 6: Components and contact conditions of the finite element model**

## 4. NUMERICAL PROCEDURE

Quasi-static numerical simulations are performed using commercial finite element analysis software MSC. Marc® with contact and boundary conditions based on the experimental study (Figure 6) [2, 3]. All components are modeled as deformable bodies. Tip-composite-layup, tip-weight, loading pins, text fixture, bolts and nuts are considered as interacting contact bodies. Tabs are perfectly bonded to the skin of the tip layup. Also; each bolt-nut couple is perfectly bonded to each other. In addition to contact constraints, there are two boundary conditions; first one constraint upper and lower nodes of loading pin1 in all directions, whereas; other loads the loading pin 2 with constant velocity. (20N/sec). This loading condition simulates centrifugal force on the tip weight.

Composite material is modeled by using 3D eight node composite brick elements in which each composite layer has four integration points (Figure 8). This element type is suitable for 3D composite analysis, since each integration point is located in the middle of the thickness of each layer [6]. Other structural components (tip weight, loading pins, test fixtures, bolts and nuts) are modeled by using 8-node hexagonal elements. After successive refinement of the mesh, it is observed that 28884 elements with constant time step algorithm produce stable results. Distribution and topology of elements are presented in Figure 7.

Number and sequence of carbon layers are modeled same as designed skin and outer wrap. The material properties and allowable of carbon fabric and glass UD layers are determined with numerous coupon tests [4] and these values are used in the FEA analysis (Table 1). Furthermore, to estimate failure behavior of the composite replaceable blade tip, distribution of different failure criteria such as maximum stress, Hill, Tsai-Wu, Puck, Hashin are calculated along the tip geometry. After evaluating the test data a proper failure criteria for thick composite lug is investigated.

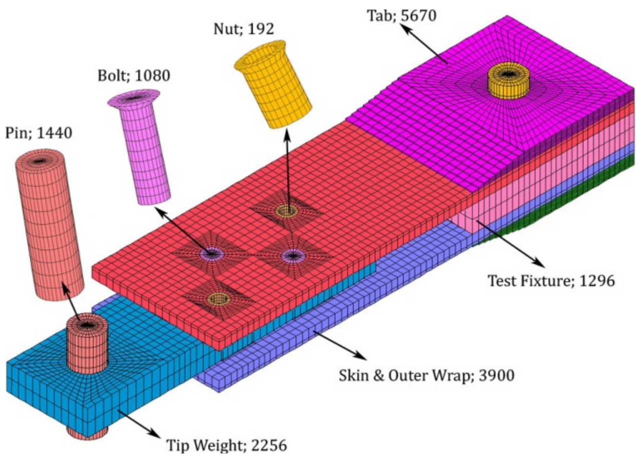


**Figure 7: Finite element mesh and corresponding number of elements**

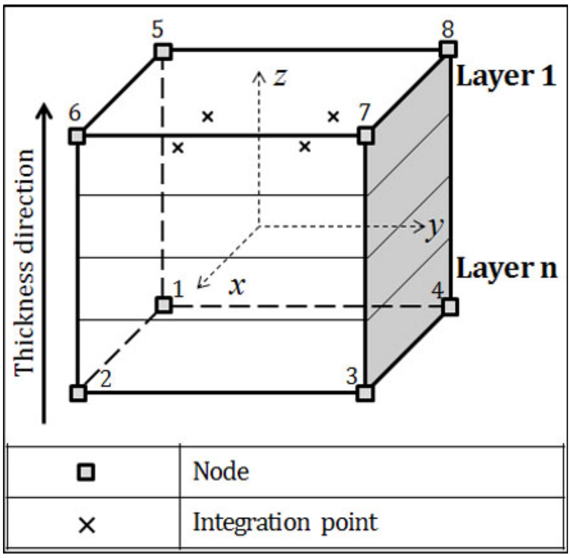


**Figure 8: Typical 8-node 3-D composite element, element type 149 in MSC.Marc.**

**Table 1: Material data of GFRP UD and CFRP Fabric layers**

| Properties | GFRP UD Material | CFRP Fabric Material |
|---|---|---|
| $E_1/E_1$ | 1 | 1 |
| $E_2/E_1$ | 0.054 | 1 |
| $E_3/E_1$ | 0.054 | 0.114 |
| $G_{12}/E_1$ | 0.025 | 1 |
| $G_{23}/E_1$ | 0.020 | 0,75 |
| $G_{31}/E_1$ | 0.025 | 1 |
| $\nu_{12}/\nu_{12}$ | 1 | 1 |
| $\nu_{23}/\nu_{12}$ | 0.6 | 0.6 |
| $\nu_{31}/\nu_{12}$ | 0.02 | 0.02 |
| Thickness | 0.185 | 0.28 mm |

## 5. RESULTS

### 5.1 Experimental Results

Five test specimens are manufactured and to acquire initial understanding only one specimen is tested in tension test- set up. The other specimens will be instrumented with strain gauges and tested in the light of the first test results.

Until the 12000 N no failure was shown in test specimen. According to the Figure 11 bearing failure start at 12000N and it continue until 14000 N.

Moreover second bearing failure is shown in Figure 11 at 19000 N and it continues until 23000 N.

It can be clearly observed that all bearing failure occurred in the middle of the test and all bearing are shown at Figure 9.

After the 23000 N specimen again starts to carry load until 47000 N. Complete failure are observed at this point. Shear failure occurred at screw which is shown Figure 10

In this test, main target was to investigate ultimate strength of the blade tip system. In the outlook of the study Load-strain behavior of the system will be investigated.

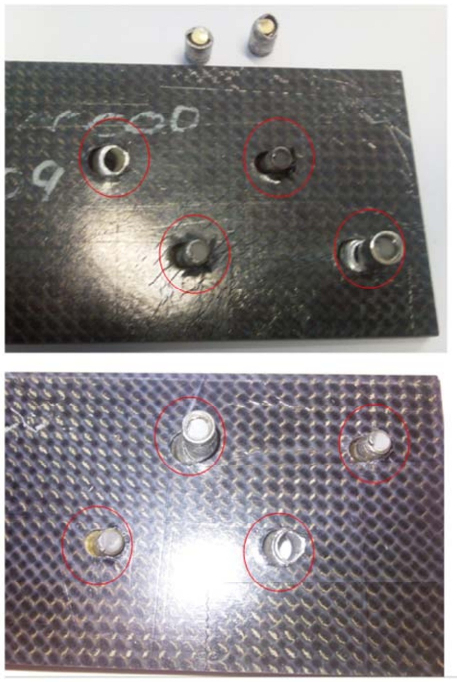

**Figure 9 Bearing failure of tip specimen**

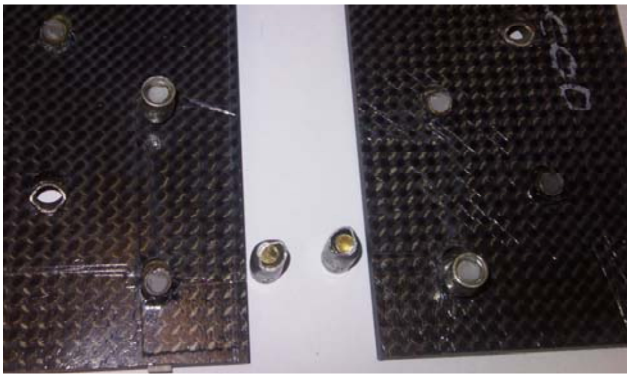

**Figure 10 Shear failure of tip specimen**

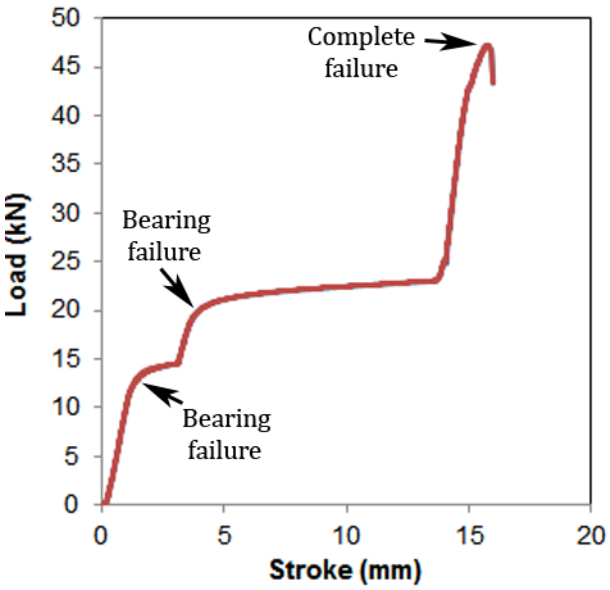


**Figure 11  Test "Load- Stroke" results**

### 5.2 Simulation Results

Deformation predicted by simulations is similar to the deformation observed in the test (Figure 12a). Upper and lower skins are warped, whereas; bolts and nuts are bent. Also, distribution of component 11 of stress in the most critical layer (outermost carbon fabric layer) is presented in Figure 12b. As it is expected, stresses are concentrated around the holes.

During simulations; Tsai-Wu, Hill maximum stress, Puck and Hashin failure criteria are calculated. Analysis showed that all of the failure indices provide similar qualitative distribution along the test specimen, and they well-predict the location of failure. All of the indices reach their maximum at the edge of the bolt-nut with dominated bearing failure at tests. Distribution of failure indices is illustrated with Tsai-Wu results in Figure 13a.

As it is shown in Figure 13b, when failure indices are compared in terms of their success in prediction of first ply failure, there are considerable differences. First of all, Tsai-Wu and Hill criteria predict bearing failure significantly earlier than test results which is not preferable in design stage. In the case of maximum stress, if criterion is considered in 22 direction of stress, it perfectly predicts the bearing failure. However, in 33 direction it expects earlier failure. This might be due to material and numerical uncertainties in 33 direction. When Puck and Hashin are considered, it is seen that, they predict failure to be in fiber compression mode and they predict first ply failure close but in slightly later stages of the test [7]. Therefore, these two criteria are not in the conservative side in the current case. As it is expected, these two criteria give equal results when dominant mode is fiber compression.

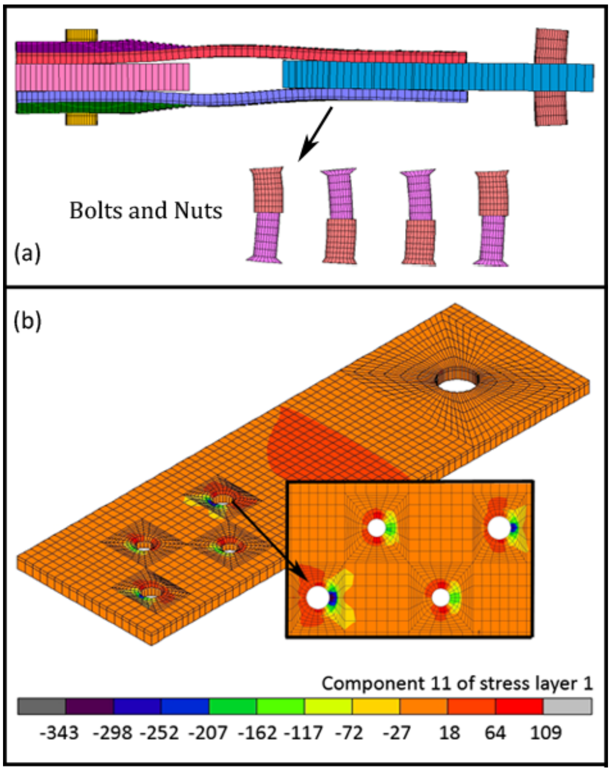


**Figure 12: (a) Deformed body (deformation exaggerated 4 times). (b) Component 11 of stress layer 1 in MPa.**

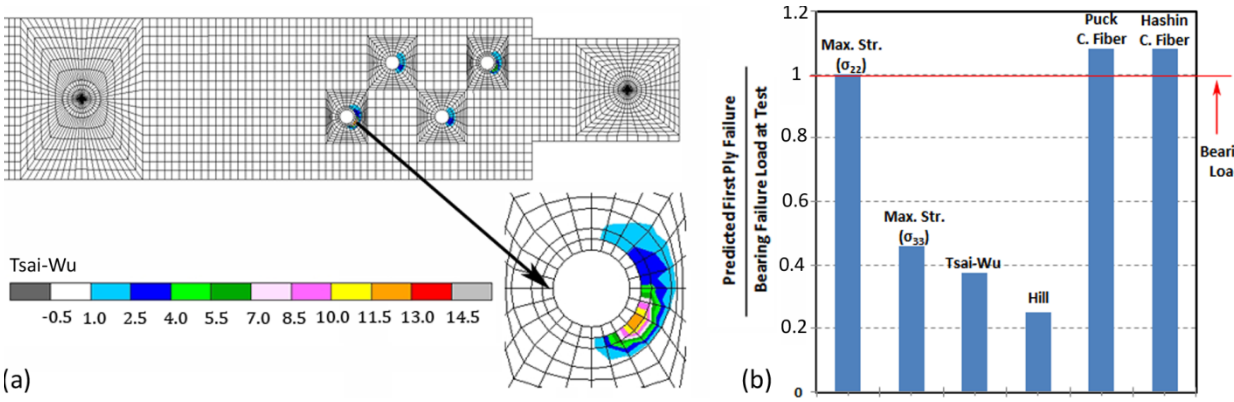


**Figure 13: (a) Tsai-Wu failure index distribution in the outermost carbon fabric layer at experimental bearing failure load. (b) Comparison of first ply failure estimations of failure criteria. Predicted first ply failure is divided by bearing failure load at the test.**

In the case of bolt and nut, it is observed that analysis predicts very early yielding. In Figure 14, Von Mises stress distribution in bolts and nuts at the instant of experimental bearing failure load are presented. It is clear form figure that, stresses are quite high and bolt-nuts are yielded before this loading. During the tests small amount of bending also observed at the instant of bearing failure; however, with current test results and analysis, it is not possible to compare analysis and tests in terms of extent of deformation. The reason is that, deformation of bolts and nuts have not been measured in tests and plastic behavior of the bolts and nuts has not been included to the simulations. Such high Von Mises stresses do not mean bolts and nuts ultimately fail. These are caused by the absence of plasticity in the material model. The success of analysis in predicting ultimate failure can only be determined by including plasticity in the material model.

During numerical simulations and test-analysis comparison, behavior of the composite layers, bolts and nuts are investigated until the bearing failure, not till the end of the test. Behavior after this point is not scope of this study. This is because; after first ply failure, nonlinearity due to softening of the composite brought in to composite material and this affects both stress-strain behavior and bolt-composite interaction. To be able to model this phenomenon, progressive failure should be included to the model, which is planned in the outlook of this study.

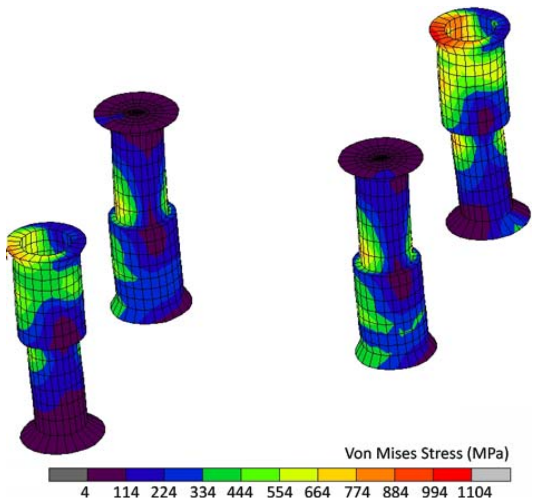


**Figure 14: Von Mises stress distribution in bolts and nuts at the instant of experimental bearing failure loads.**

## 6. CONCLUSION

In this study by the help of conducted static test and finite element analyses, an understanding for prediction strength and failure behavior of replaceable tip design is obtained. Anhedral-Tapered tip design is concentrated and the design details of the attachment of the tip to the blade are determined. Under centrifugal loading the design of blade tip including the tip weight is verified by the conducted static test. However, further sub-component tests are planned for the static confirmation of the blade tip before full-scale Whirl Tower tests.

When the finite element analyses are evaluated, current results show that, maximum stress criterion gives the most reliable results. Inconsistency in 33 direction can be attributed to the material uncertainties. Puck and Hashin are close enough to the tests but they are not on the conservative side. On the other hand, Tsai-Wu and Hill predicts failure in a very early stage which is not preferable. Yet, it should be noted that, these results in failure indices might be because of the numerical and material uncertainties in 33 direction or undetectable local failure in the test specimen during the test. In the outlook of this study, more experiments and numerical investigations about the topic is planned.

## 7. REFERENCES


[1] Bruhn, E. F. Analysis and Design of Flight Vehicle Structures. Jacobs Publishing, Indianapolis, IN, Chap. D1.11, 1973

[2] MSC. Marc 2008 r1 Volume A: Theory and User Information

[3] MSC. Marc 2008 r1 Volume B: Element Library

[4] "Composite Materials Handbook, Volume 1 Polymer Matrix Composites Guidelines for Characterization of Structural Materials", MIL-HDBK-17-1F, Department of Defense Handbook, 17 June 2002

[5] Reddy D.J. "Composites in Rotorcraft Industry & Damage Tolerance Requirements." *FAA Composites Workshop*, 2006.

[6] Kuhlmann G., Rolfes R. "A Hierarchic 3D Finite Element for Laminated Composites." *International Journal for Numerical Methods in Engineering*, Volume 61, Issue 1, pages 96–116, 7 September 2004

[7] Lopez R. H., Luersen M. and de Cursi E. S. "Puck Failure Criterion As Constraint In The Optimization Of Laminated Composites." WCCM8, ECCOMAS 2008, Venice, Italy, June 30 – July 5, 2008