Acta Technica 62 No. 3A/2017, 335–346

Analysis on modeling and numerical simulation for badminton racket of braiding composite material based on ANSYS

Liyan Zhang[1]

Abstract. Braiding composite material featured good mechanical property such as high strength and high modulus is widely used in badminton racket manufacturing. Parameterized finite element models for badminton racket of braiding composite material is built by ANSYS software based on Domain Superposition Technique of finite element, such as filament reinforcement phase model and matrix phrase model. Then, statics analysis, elastic properties prediction, invalidation discrimination and modal kinetic analysis are made for badminton racket model, which provides theory for design racket. Finally, elastic property of racket shaft is predicted and its result proves that Domain Superposition Technique of finite element is feasible.

Key words. Braiding composite material, badminton racket, ANSYS, Domain Superposition Technique of finite element, parameterized modeling.

1. Introduction

Composite material, a new material with excellence performance, is made up of no less than two materials of different properties by chemical or physical treatment, such as organic polymer, inorganic nonmetal and metal. Braiding composite material with well mechanical property is made up of pre-form and matrix phase material [1]. And pre-form with overall structure is woven by different methods. Nowadays, studies on braiding composited material are generally focused on unit cell model. Then, proper periodic boundary condition and load was used for mechanical analysis. However, few studies were focused on building overall model with braiding composite material. Badminton racket made up of braiding composite material was taken as object of study and its material model was built to study the overall mechanical property was studied.

[1]Shenyang Normal University, Liaoning, 110034, China; E-mail: `liyanzhang00@aliyun.com`

2. Literature review

In the early 1980, Ko [2] identified "fibrage" in his thesis and elaborated cubic unit cell model with rectangle surface for the first time. A finite element model with multi-scale was built by Wang et al. [3] using fiber, unit cell and laminate in three dimensions. Multi-scale progressive damage analysis was made on composite material of 2×2 braiding structure. However, finite element analysis can only focus on a certain parameters of the structure because of diverse model parameter. Theory and experiment was combined by Masters [4] to analyze mechanical properties of braiding composite material. Among those four methods, laminate model was the easiest one to be achieved. For even laminated board, direction of fiber bundle was the same and a bending correction factor was introduced to represent bending property of the fiber.

Thee badminton racket designed here is manufactured by resin transfer moulding (RTM), a technology developed from wet molding process and shooting technique and the most widely used and mature liquid modeling technology. Products of RTM molding is featuring smooth surface, stable quality, high fiber volume content, minor pollution, low cost and high efficiency. Finite element method can be used to simulate lay-up of fiber material and resin impregnation-reinforced fiber material and to study technological parameter, temperature and pressure those influence molding. Based on previous study, modeling and numerical simulation for badminton racket of braiding composite material was analyzed using ANSYS.

3. Research method

3.1. Domain Superposition Technique of finite element

At present, study on braiding composite material modeling was almost all focused on unit cell model, a representative elementary volume. Unit cell model built by traditional braiding composite material was aimed to reflect fiber reinforced phase model and matrix phase model as well as the overall construction of component as true as possible. True braiding composited material model was obtained after fiber reinforcement phase model was subtracted from matrix phase model with Boolean operation based on ANSYS. Thus, two phases finite element model with no crossover and overlap in space were obtained. Besides, special unit should be added at the junction of the two phases, thus related mechanical relation between them was produced. Within the final model, periodic boundary condition was used to finish pretreatment.

Braiding composite material was generally identified as an interconnection of a large number of representative periodic unit cells. Thus, variation of boundary surface of neighboring unit cell should be the same when under load. Variation of any point on surface and its corresponding node should be the same. Displacement field of any pair of parallel surface can be described as follows [5]

$$\mu_i^{j+} = \varepsilon_{ik} X_k^{j+} + \mu_i^*,\ (i,j,k = x,y,z)\,, \tag{1}$$

$$\mu_i^{j-} = \varepsilon_{ik} X_k^{j-} + \mu_i^* \,. \tag{2}$$

In above equation, μ denotes denoted displacement of any point during a unit, $j+$ denotes in the positive direction, $j-$ denotes in the negative direction, ε_{ik} is the average full-field strain and X_k^{j+} denotes the coordinates of corresponding node of two interfaces of X_{j+} direction in k direction. The below equation can be obtained by combining equation (1) and (2):

$$\mu_i^{j+} - \mu_i^{j-} = \varepsilon_{ik}\left(X_k^{j+} - X_k^{j-}\right) = \varepsilon_{ik}\Delta X_k^j \,. \tag{3}$$

APDL program was used to matching coupling and node. Corresponding coupling and node relationship was identified through equation (3), thus periodic boundary condition was identified.

Topological structure with internal complex, high fiber volume fraction and interlaced fiber bundle are hard to be solved by traditional modeling method. Thus, Domain Superposition Technique (DST) of finite element [6] is put forward based on finite element. Finite element model for matrix phase and fiber reinforcement phase should be built for implement Domain Superposition Technique (DST) of finite element. And entity unit is adopted by those two phases model. There is an overlap of junction of the built matrix phase and fiber reinforcement phase. Two problems can be tackled by Domain Superposition Technique (DST) of finite element; the first one is stiffness matching of the two phases and the second is coupling the two phase modes properly. Therefore, displacement of the same point in overlapping region is the same when under stress. Model built by DST can simulate the structure of composite material only when stiffness matching and coupling are properly tackled.

A “negative” matrix is needed to match material stiffness by Domain Superposition Technique. Under the same stress state, stress state of “negative” matrix material is opposite to that of matrix material. For linear elastic material, “negative” nature of matrix material is the negative value of its stress-strain stiffness matrix. For isotropic linear elastic matrix material, its stress-strain stiffness matrix may be written in the form

$$D_{\text{host}} = \frac{E(1-v)}{(1+v)(1-2v)} \cdot \begin{bmatrix} 1 & \frac{v}{1-v} & \frac{v}{1-v} & 0 & 0 & 0 \\ \frac{v}{1-v} & 1 & \frac{v}{1-v} & 0 & 0 & 0 \\ \frac{v}{1-v} & \frac{v}{1-v} & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2v}{2(1-v)} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2v}{2(1-v)} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2v}{2(1-v)} \end{bmatrix} . \tag{4}$$

In the above equation, E denotes the elasticity modulus of material and v denotes Poisson’s ratio of matrix material. “Negative” stress-strain matrix is given as

$$D_{\text{Negative material}} = D_{\text{Actual matrix}} \,. \tag{5}$$

Here, $D_{\text{Negative material}}$ is the stress-strain matrix of negative material and $D_{\text{Actual matrix}}$

is the stress-strain matrix of actual matrix.

When implementing the Domain Superposition Technique (DST), it should match material nature to fiber reinforcement phase. The final filamentary phase material after matching equals the subtract matrix material nature from filament phase material nature, which can be obtained by subtracting the actual matrix phase stress-strain stiffness matrix from actual fiber reinforcement phase tress-strain stiffness matrix:

$$D_{\mathrm{cor}} = D_{\mathrm{en}} + D_{\mathrm{ho}}\,. \tag{6}$$

In the above equation, D_{cor} denotes the matrix of modified reinforced stress-strain matrix, D_{en} denotes the matrix of actual reinforced stress-strain matrix and D_{ho} denotes the matrix of actual stress-strain matrix (substrate). In a word, the nature of matrix material was identified by grid cell of overall model and modified material constitutive model was identified by unit of fiber bundle region. Thus, material characteristics of overall superposition domain and true fiber bundle material was the same. For any designated node in any unit, its coordinate of the coupling of model fiber phase and matrix phase node is identified as

$$x = \sum_{i=1}^{m} N_i x_i, \quad y = \sum_{i=1}^{m} N_i y_i, \quad z = \sum_{i=1}^{m} N_i z_i\,. \tag{7}$$

In the above equation, x, y and z denote the coordinates of the overall coordinate system for a node in the unit. This is an interpolation function of the unit which is identified under natural system of coordinates. Symbol N denotes the shape function that is given as $N = \mu/\delta^e$, μ being the distance of any point during the unit and δ^e the displacement of the element node. Also, the displacement of the designated node is identified as

$$u = \sum_{i=1}^{m} N_i u_i, \quad v = \sum_{i=1}^{m} N_i v_i, \quad w = \sum_{i=1}^{m} N_i w_i\,. \tag{8}$$

Here, u, v and w denote the displacement components of a node in the unit.

The principle of Domain Superposition Technique is coupling of the designated node and freedom degree of the node on the same unit with the designated node. Thus, node coupling method is used to couple node of the fiber reinforcement phase model to the matrix model.

3.2. Parametric modeling and mechanical analysis

For the designed racket handle, its diameter was 7.3 mm, thickness was 1.5 mm and length was 450 mm. Given racket frame was an oval, whose major semi-axis was 140 mm, minor semi-axis was 110 mm, thickness was 14 mm and its diameter was set as 0.7 mm. Composite material of carbon fiber-epoxy resin was used, whose fiber reinforcement phase material was M40 carbon fiber and matrix phase and bracing wire was bisphenol A (BPA) epoxy resin. Parameters of material used for fiber and resin are shown in Table 1.

Table 1. Thermophysical properties of regular fluid and nanoparticles

Material property	E_1 (GPa)	E_2 (GPa)	G_{12} (GPa)	G_{23} (GPa)	v_{12}	v_{23}
Carbon fiber	393	25.53	25.53	11.94	0.20	0.25
Resin	3.62	3.62	-	-	0.35	0.35

(1) Analysis on finite element model

Solid64 entity unit was used to build fiber reinforcement phase and matrix phase of racket of two-dimension and three-direction braiding. There were three axial yarn and four braiding yarn included in the unit cell model of braiding composite material [7]. Fiber bundle material with filament dip-dyed part of resin and elastic constant of fiber bundle material was calculated by weighted average method. Through calculation below we obtained: $E_1 = 141.90\,\text{GPa}$, $E_2 = E_3 = 7.13\,\text{GPa}$, $G_{12} = G_{13} = 4.65\,\text{GPa}$, $G_{23} = 3.93\,\text{GPa}$, $v_{12} = v_{13} = 0.23$, $v_{23} = 0.36$. For simulating pre-stress of positions that bracing line connected with racket frame, PSMESH command was used to build pre-load stretching section of each positions. Then, SLOAD command was used to exert preload and poundage of bracing wire can be parameterized. Two-dimension and two-direction braiding and two-dimension and three-direction braiding were adopted by fiber reinforcement phase and both of their knitting angles are 45 degree. Fiber volume fraction of racket of two-dimension and three-direction braiding was 21.56 %. Finite element model of fiber phase and matrix phase for badminton racket handle of two-dimension and three-direction braiding are shown in Figs. 1 and 2. Other specified model are not listed one by one here.

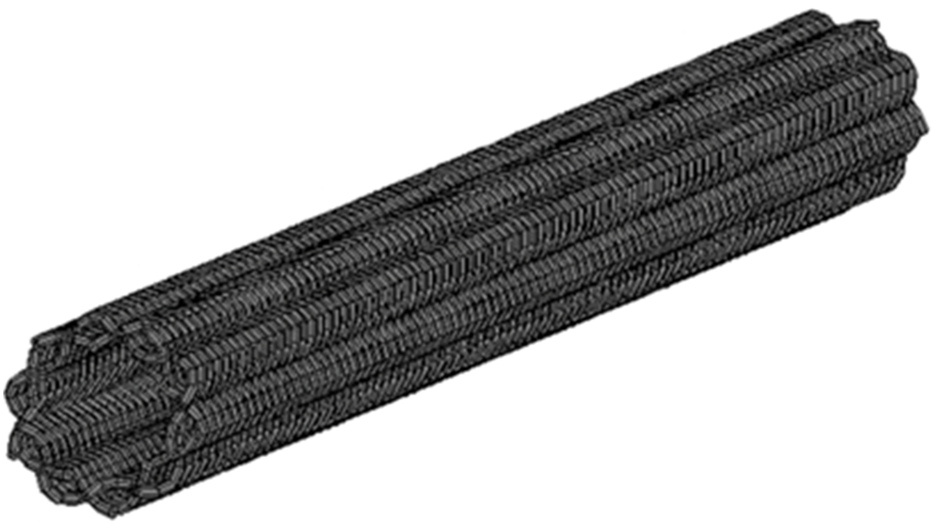

Fig. 1. Finite element model for fiber phase of racket handle of two-dimension and three-direction braiding

According to the practical situation of badminton racket under stress, two different forces were put on the two end faces of ANSYS model of racket handle using

Fig. 2. Finite element model for matrix of racket handle of two-dimension and three-direction braiding

DST- a full constraint and a vertical force. Diameter of badminton cork was 20 mm–25 mm, thus a round area was defined to be hit by vertical force (300 N). For the round area, midpoint of bracing wire was identified as a center of a circle and 25 mm was set as the diameter. Domain Superposition Technique was used to process the built finite element parameterized model, thus the final model was identified as shown in Fig. 3.

Fig. 3. Finite element model of badminton racket modified by Domain Superposition Technique

Under above constraint and stress, it can be calculated by ANSYS that the maximum equivalent stress of the whole racket was 154.28 MPa and the maximum equivalent displacement was 1.25 mm, which was positioned around the top of racket frame. The maximum equivalent stress of racket frame was 72.86 MPa at cross point of fiber and matrix phase. The maximum equivalent stress of racket handle was 154.82 MPa around the contact position of racket handle and racket frame. Position around the connection point of racket frame and racket handle can be easily broken off because figure of the contact position was changed. Thus, it can be inferred that distribution of whole equivalent stress of braiding composite material was uneven and whole equivalent stress of fiber bundle was obviously higher than whole equivalent stress of the matrix. Therefore, it can be known that fiber bundle of braided composite material was bearing the main force. Besides, apart from the stress concentration at top, whole fiber bundle was under even distributed force and whole equivalent displacement of racket was linear distributed. Thus, whole performance of braiding composite material under stress was proved basically the same.

For finite element model for racket of two-dimension and two-direction braiding, its fiber volume fraction was 13.75 %. Above parametric modeling and mechanical analysis was made for racket of two-dimension and two-direction braiding, thus it can be found that under the same constraint and loading, the maximum equivalent stress of model of two-dimension and two-direction braiding and model of two-dimension and three-direction braiding was 154.28 MPa and 198.18 MPa, respectively. The whole equivalent stress of two-dimension and three-direction was much less than that of two-dimension and two-direction. Besides, their whole equivalent displacement was different. The reason that the whole bearing capacity of braided racket of two-dimension and three-direction braiding was much better than that of braided racket of two-dimension and two-direction braiding was that part of the loading was born by the added axial fiber of racket of two-dimension and three-direction braiding. The whole equivalent stress of fiber bundle of racket of two-dimension and two-direction braiding was obviously higher than that of the matrix. The whole force of fiber bundle was even distributed except that there was stress concentration at the end of constraint.

(2) Failure identification

With the wider application scope of braiding composite, it was of more and more practical importance to have a failure criterion. Nowadays, strength criterion of modified traditional material was applied by many researchers to braiding composite material, including the maximum stress criterion, Tsai-Hill criterion and Tsai-Wu multinomial criterion. Among all these mature criterion to explain fracture of composite material, Tsai-Wu multinomial criterion was the most comprehensive one and it would not be described in detailed as the limited to layout. According to failure identification criterion of Tsai-Wu, when failure factor was no less that 1, component was identified as failure. When intensity factor was less than 1, component was identified as safe. For racket handle of two-dimension and two-direction braiding, the maximum overall intensity factor of Tsai-Wu failure criterion was 0.51. For racket handle of two-dimension and three-direction braiding, the maximum overall intensity factor of Tsai-Wu failure criterion was 0.36. Because intensity factor of those two braiding method was less than 1, the overall component was identified as safe and would not fail. Security coefficient of racket handle of two-dimension and three-direction braiding was higher than that of the racket handle of two-dimension and two-direction braiding. The racket design was identified as reasonable and safe because it was measured within security coefficient under above extreme case.

(3) Elastic performance prediction for racket handle

Nowadays, mechanical performance analysis on composite material was mainly focused on elastic performance, specifically numerical modeling on elastic modulus, Poison's ration and shear modulus based on unit cell. Thus, elastic performance of the overall racket handle of composite material was analyzed as bellow based on Domain Superposition Technique (DST) of finite element.

Two different forces were put on the end faces of finite element model of racket handle using DST—a full constraint and a displacement loading. Let $\delta = 0.05 \times L$, which means that the overall strain is 0.5 %. The mean strain ε and mean stress δ_x on the plane $X = L$ was calculated by ANSYS software. Then, elasticity modulus E_y, Poisson's ratio μ_{xy} and shear modulus G_{xy} of the composite can be obtained according to this formation and definition formula. Besides, $E_y = E_z$, $\mu_{xz} = \mu_{yz}$ and G_{xy}, G_{yz} can be obtained using the common approach.

ANSYS finite element analysis model for predicting elastic performance is shown in Fig. 4.

Fig. 4. Model for predicting elastic performance of racket handle with full constraint and displacement loading

It can be concluded that with Domain Superposition Technique (DST) of finite element, it is effective to analyze overall component of racket handle, which effectiveness is also proved by accurate result of sparse finite element mesh. Elastic performance of racket handle predicted by DST is shown in Table 2.

Table 2. Predicting result of elastic performance

	E_x (GPa)	E_y (GPa)	E_z (GPa)	μ_{xy}	μ_{xz}	μ_{yz}	G_{xy} (GPa)	G_{xz} (GPa)	G_{yz} (GPa)
Two-dimension and two-directions	40.33	6.35	6.33	0.31	0.31	0.93	13.6	13.6	3.2
Two-dimension and three-directions	61.75	9.31	9.31	0.31	0.35	0.97	15.4	15.1	4.7

In conclusion, because part of the loading was born by the added axial fiber, racket of two-dimension and three-direction braiding was better that racket of two-dimension and two-direction braiding. Weight of those two rackets was almost the same. Besides, hand feeling and overall anti-bending and anti-torsion of racket of two-dimension and three-direction braiding was better than that of racket of two-dimension and two-direction braiding. Thus, racket of two-dimension and three-direction braiding was more and more popular among designers and amateur of badminton.

3.3. Analysis of modality dynamics

Modality is the inherent vibration of object. There are corresponding inherent frequency, damping ratio and modal shape for each order of mode. Those parameters can be used to design and optimize badminton racket to avoid unnecessary loss caused by resonance during hitting through modal analysis on finite element and finite element analysis result. Finite element model for modal analysis is the same model for static analysis and material property used in statics. Fixed constraint was put on one of the end of racket handle. Modal calculation of finite element for racket was calculated by ANSYS software with partitioning Lanczos method when there was not loading on finite element model, which calculation was based on parameterized calculation procedure developed according to static calculation method. For modality of high order, its contribution value to response was low and its rate of decay was relative fast. Generally, only modality of low order was included in modality analysis, thus, inherent frequency of the first-four-order was selected. For racket of two-dimension and three-direction braiding, conclusion of inherent frequency and mode of vibration of the first-four-order are shown in Table 3.

Table 3. Inherent frequency and mode of vibration of the first-four-order of racket of two-dimensional and three-directional braiding

Order	Inherent frequency (Hz)	Mode of vibration
1	34.25	Vibration in $X--Z$ plane
2	56.83	Vibration in $X--Y$ plane
3	75.92	Vibration in $X--Z$ plane
4	120.60	Compound vibration in plane $X--Y$ and $X--Z$

4. Design result and analysis

Overall size of badminton racket and the automatic analysis software were designed based on visual programming software Microsoft Visual Basic6.0 and finite element analysis software ANSYS. The analysis result can be extracted by the pre-programmed procedure and automatic generated a report saved as WORD format.

The software designed has to be logged in with user name and code for security. At the top of operation interface, there were menu item, tool bar and diagram figure of ANSYS finite element for badminton racket. In function menu, there were system, parametric modeling of finite element, analytical calculation and analysis report. There were options for specific function in pull-down menu, such as initial installation path of software, material property setting for matrix phase and fiber phase, boundary conditions and loading, ANSYS solving and exit. Statics analysis, failure identification and modal analysis were included in this software. Click statics analysis, load and boundary constrain interface would popup. User can load badminton racket according to specific requirement. Thus, the earlier stage of designing size of badminton frame and handle, constraint and load were accomplished. After loading, a box of "save data" would popup. Click "confirm", the software would start analysis and solving with background program. When background program finished reading, dialog box would return to operational status and popup "ANSYS calculation finished". Then, information bar would remind user the succeed solving. The final solving result can be checked by clicking "result query". At the left side of interface, result of overall badminton racket, racket frame, racket handle can be checked according to the need. Corresponding equivalent stress and displacement fringe can also be checked. Partial enlarged details of fiber phase equivalent stress at racket handle of two-dimensional and three-directional braiding are shown in Fig. 5.

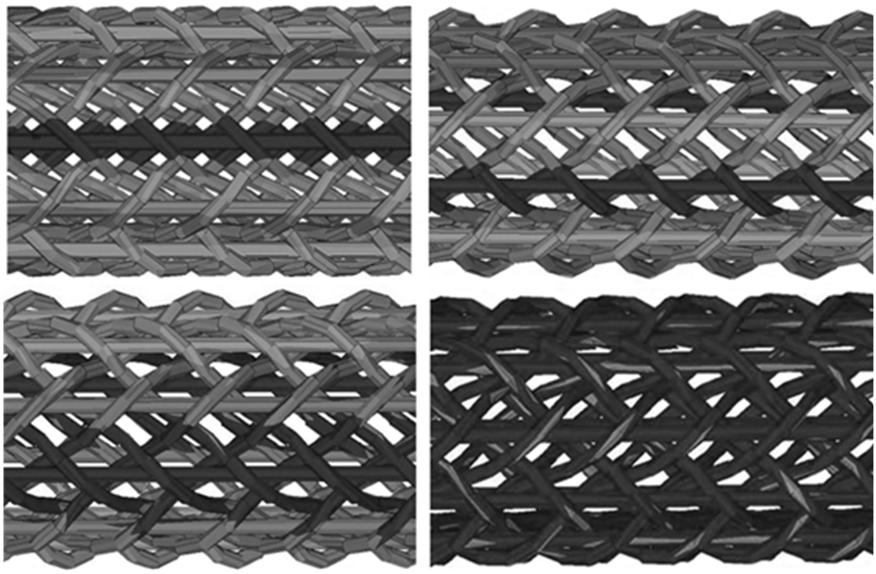

Fig. 5. Partial enlarged detail of fiber phase equivalent stress at racket handle of two-dimension and three-direction braiding

With this software, overall parameterization design of badminton racket and computation requirement of automatic finite element analysis can be satisfied, which is of practical significance because a more convenient guideline for badminton racket design is provided.

5. Conclusion

Main research result:

(1) Finite element model for badminton racket of two-dimension and two-direction braiding and two-dimension and three-direction braiding was built based on Domain Superposition Technique (DST) of finite element. Equivalent stress and equivalent

displacement fringe of racket when hitting can be calculated with ANSYS analytical software. Besides, statics analysis, elastic performance prediction, failure identification and modal mechanical analysis for racket were complete.

(2) Calculation and optimization software for badminton racket structure was developed based on visual programming software VB 6.0 and software of finite element analysis ANSYS, which can be used to design structure of badminton racket and analyze and calculate it with finite element.

References

[1] Z. X. Tang, R. Postle: *Mechanics of three-dimensional braided structures for composite materials – part I: fabric structure and fibre volume fraction.* Composite Structures *49* (2000), No. 4, 451–459.

[2] F. K. Ko: *Three-dimensional fabrics for composites. An introduction to the weave structure.* Proc. International Conference on Composite Materials, ICCM IV, 25–28 October 1982, Tokyo, Japan, Progress in Science and Engineering of Composites, Japan Society for Composite Materials *2* (1982), 1609–1619.

[3] Y. Q. Wang, A. S. D. Wang: *On the topological yarn structure in 3-D rectangular and tubular braided preforms.* Journal Composites Science & Technology *51* (1994), 575–586.

[4] J. E. Masters, R. L. Foye, C. M. Pastore, Y. Gowayed: *Mechanical properties of triaxially braided composites: Experimental and analytical results.* Journal of Composites, Technology and Research *15* (1993), No. 2, 112–122.

[5] Z. Xia, Y. Zhang, F. Ellyin: *A unified periodical boundary conditions for representative volume elements of composites and applications.* International Journal of Solids and Structures *40* (2003), No. 8, 1907–1921.

[6] W. G. Jiang: *Implementation of domain superposition technique for the nonlinear analysis of composite materials.* Journal of Composite Materials *47* (2013), No. 2, 243–249.

[7] C. Zhang, X. Xu, X. Yan: *Three unit-cell structure models of 3-D five-directional and full five-directional braided composites.* Journal of Nanjing University of Aeronautics & Astronautics *45* (2013), No. 02, 170–178.

Received May 7, 2017