Stress Analysis of Optical Fiber Embedded Composite with Lenticular Resin Pocket

Shiuh-Chuan Her*, Bo-Ren Yao* [Shiuh-Chuan Her #]: mesch@saturn.yzu.edu.tw
***Department of Mechanical Engineering Yuan-Ze University**

Keywords: *embedded optical fiber, smart structure, lenticular resin pocket, tri-axial loading*
#: Name of presenter

1. Introduction

Structures integrated with optical fiber sensors are considered as primary candidates for smart structures to monitor the environmental condition and structural health. Fiber optic sensor with small size, light weight and immunity to electromagnetic interference can be embedded into the host material to form an ideally smart structure system. It is known that stress concentration arises from embedded optical sensor due to the mismatch of material property between the host material and optical fiber. These high stresses will not only perturb the values of the field variables being measured, but also induce local damage in the region of the optical fiber. If an optical fiber is embedded in a composite laminate, cylindrical geometry is the most plausible shape to utilize when the optical fiber is parallel to the reinforced fibers of the composite in adjacent layers [1]. However, when the optical fiber is not aligned with reinforced fibers of adjacent layers, the embedded optical fiber is surrounded by a lenticular resin pocket [2]. Dasgupta et al. [3] used Rayleigh-Ritz variational method to obtain the shape and size of the lenticular resin zone. Case and Carman [4] employed the principle of minimum of potential energy to estimate the shape of the lenticular resin zone. The geometry of the resin pocket affects the stress transfer mechanism between the host composite and the fiber optic sensor. In this investigation, the stress distribution in the resin pocket and composite laminate of the optical fiber embedded composite subjected to axial and transverse tensile loading is investigated by finite element method. The geometry of the lenticular resin obtained by Dasgupta et al. [3] is adopted as follows.

2. **Geometric Model of Lenticular Resin**

The lenticular resin pocket surrounded the embedded optical fiber in the composite laminate is shown in Fig.1. Dasgupta et al. [3] used polynomial function to model the vertical displacement as follow

$$v = (c_1\Phi_1 + c_2\Phi_2)(1-\frac{y}{h}) - \frac{h_c}{h}y$$

$$\Phi_1 = \frac{\left(\frac{a}{\sin\theta}\right)^4}{EI}\left[-\frac{1}{6}\left(\frac{x}{a}\right)^2 + \frac{1}{6}\left(\frac{x}{a}\right)^3 - \frac{1}{24}\left(\frac{x}{a}\right)^4\right]$$

$$\Phi_2 = \frac{\left(\frac{a}{\sin\theta}\right)^4}{EI}\left[-\frac{1}{16}\left(\frac{x}{a}\right)^2 + \frac{1}{12}\left(\frac{x}{a}\right)^3 - \frac{1}{24}\left(\frac{x}{a}\right)^4 + \frac{1}{120}\left(\frac{x}{a}\right)^5\right]$$

Where c_1, c_2 are undeterminate constants; a is the width of the resin pocket; h is the half of the thickness of the composite laminate; h_c is the displacement of the top surface. The potential energy of the whole structure is given by

$$\Pi = U_b + U_c - W$$

$$= \frac{1}{2}\int_V E_x\left(\frac{\partial^2 v}{\partial x^2}\right)^2 y^2 dV + \frac{1}{2}E_y\int_V\left(\frac{\partial v}{\partial y}\right)^2 dV - q\int_S v_{(y=h)}dA$$

Where q is the uniform external loads on the top surface during the manufacturing process. Utilizing the principle of minimum of potential energy

$$\frac{\partial \Pi}{\partial c_i} = 0 \quad , \quad \frac{\partial \Pi}{\partial a} = 0 \quad , \quad \frac{\partial \Pi}{\partial h_c} = 0$$

We are able to solve the unknowns of c_1, c_2, a and h_c, therefore determine the geometry of the lanticular resin pocket.

3. Finite Element Analysis

The stress distribution in the resin pocket and composite laminate is calculated by the finite element method. Finite element software ANSYS is used to perform the stress analysis. A 2-D plane strain mesh is used and fine mesh around the tip of the lenticular resin and coarse mesh elsewhere in the composite laminate as shown in Fig.2. The material properties of the graphite/epoxy composite and optical fiber is shown in table 1. The layup of the composite laminate is $[30_2/90_2/F/90_2/30_2]$ and subjected to tensile loads in the x direction. The Von Mises's stress distribution along the y-axis is shown in Fig.3.

4. Conclusion

The shape and size of the lenticular resin region surrounded the embedded optical fiber is obtained by using the method developed by Dasgupta et al. [3]. Numerical calculation of the stress distribution in the optical fiber embedded composite show that there is stress concentration at the interface between the resin pocket and optical fiber. The stress increases rapidly in the vicinity of the optical fiber. Thus, this resin pocket may form the site of interlaminate damage initiation under mechanical loads, resulting the degradation of the accuracy and long term reliability of the smart structure.

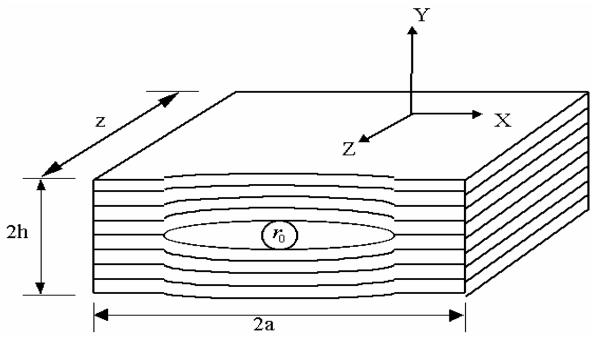

Fig.1 : geometric model of the resin pocket

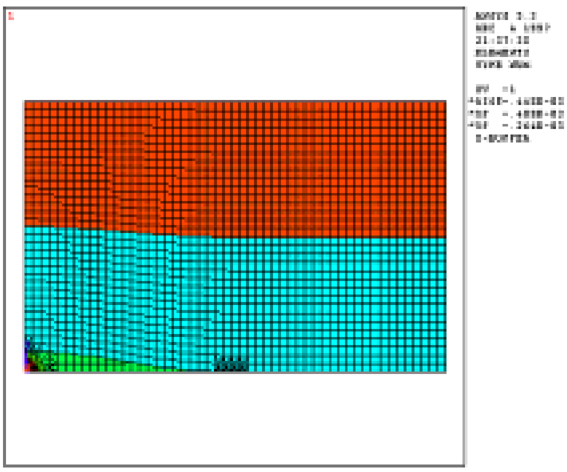

Fig.2: finite element mesh for optical fiber embedded composite

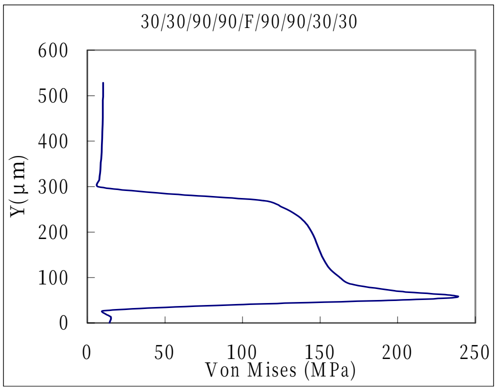

Fig.3: Von Mises stress distribution along the y-axis

Table 1: material properties of Graphite/Epoxy and optical fiber[3]

longitudinal modulus E_L	149 GPa
transverse modulus E_T	11.4 GPa
transverse shear modulus G_{LT}	4.5 GPa
Optical fiber Young's modulus	71 GPa

References

[1] Dasgupta A.and Sirkis J. "*Importance of coatings to optical fiber sensors embedded in smart structures*". *AIAA journal*. Vol. 30, No.5, pp 1337-1343, 1992.

[2] Tay A., Wilson D. A. and Wood L. "Strain analysis of optical fibers embedded in composite materials using finite element modelling". *Proceedings of* SPIE, Boston U.S.A, Vol. 1170, pp 521-533, 1989.

[3] Dasgupta A., Wan Y., Sirkis J. and Singh H. "Micro-mechanical investigation of an optical fiber embedded in a laminated composite". *Proceedings of* SPIE, Vol. 1370, pp 119-128, 1990.

[4] Case S. W. and Carman G. P. "Compression strength of composites containing embedded sensors or actuators". *Journal of Intelligent Material Systems and Structures*, Vol. 5, pp 4-11, 1994.