FrontCOMP FOR STRENGTH EVALUATION OF CARBON FIBER REINFORCED PLASTIC

N. Yoshikawa[a*], K. Hariya[a], T. Ogasawara[a], S.-W. Kim[a]

[a]*Institute of Industrial Science, The University of Tokyo, Komaba 4-6-1, Meguro-ku, Tokyo, JAPAN*
**yoshi@telu.iis.u-tokyo.ac.jp*

Keywords: Cure Process, Damage Mechanics, Finite Element Method, Pressure Vessel

Abstract

A simulation system, named FrontCOMP, has been developed for a reasonable strength evaluation of carbon fiber reinforced plastic (CFRP) members. The fiber bundle and resin are definitely distinguished for numerical modeling in meso-scale. A finite element data of meso-scale model is automatically constituted by the system, and transferred to residual stress analysis after cure. The ultimate strength of a CFRP member is estimated by explicitly taking account of meso-scopic structural parameters, such as material non-linearity of resin, size of fiber bundle and so on. The validity of the system is exemplified through analyses of high pressure vessel for adequate design of hydrogen tanks equipped on fuel cell vehicles.

1. Introduction

Almost all the CFRP members are produced by trial and error design consuming laborious destructive examinations using actual CFRP products. Ordinary design process for steel members, that is, the design by stress criterion defined by material test using test piece cannot be applied for the CFRP members. The main reason comes from meso-scopic stress concentration caused by structure of carbon fiber bundle and resin. Macro-scopic stress evaluated by rule of mixture [1] is not able to indicate the real stress state of carbon fiber and resin system, which is decisive for failure of the members.

Various trials have been attempted to establish design methodology supported by rational material tests [2]. Simulation technology represented by finite element method may be indispensable to elucidate meso-scopic stress concentration. We have developed a simulation system named FrontCOMP to support meso-scopic stress evaluation in CFRP members on the basis of fiber bundle/resin separation modeling. Merits of FrontCOMP are described with example simulations concerning filament wound CFRP pressure vessel.

2. FrontCOMP system

FrontCOMP is a simulation system to promote optimization of CFRP member strength in terms of meso-scopic material parameters. It consists of fundamental simulation softwares for the finite element analyses with meso-scopic separation of resin and fiber bundle. A meso-scale model is constituted so as to carry out optimum design of CFRP in terms of geometrical and material parameters in fiber bundle/resin system. 1) molding, 2) cure and 3) damage processes are independently simulated by separated engines.

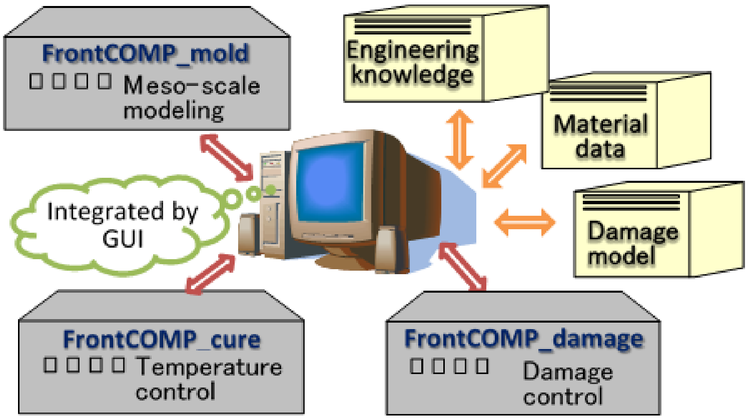


Figure 1. FrontCOMP system constitution.

2.1. FrontCOMP_mold for meso-model construction

Molding process simulator FrontCOMP_mold is an engine for meso-scopic finite element model of Filament Winding (FW) and Resin Transfer Molding (RTM) members. Fiber bundle path generator generates surface mesh and decides fiber bundle path. Topology and structure generator generates three-dimensional fiber bundle model. Solid mesh generator generates structured fiber bundle mesh and solid mesh for resin part.

2.2. FrontCOMP_cure for stress and strain evaluation during and after cure process

Cure process simulator FrontCOMP_cure is an engine for finite element analysis of residual stress and strain after cure process. Temperature T is analyzed with respect to time t by considering heat generation $\dot{Q}$ in Cartesian coordinate x-y-z,

$$\rho c \frac{\partial T}{\partial t} = \kappa_{xx} \frac{\partial T}{\partial x^2} + \kappa_{yy} \frac{\partial T}{\partial y^2} + \kappa_{zz} \frac{\partial T}{\partial z^2} + \rho \dot{Q} \tag{1}$$

where ρ, c and κ_{ii} are mass density, specific heat and heat conductivity, respectively. The heat generation $\dot{Q}$ is given with degree of cure a by following Kamal model [3],

$$\dot{Q} = H_r \frac{da}{dt} = H_r \left(K_1 + K_2 a^m \right) \left(1 - a \right)^n \tag{2}$$

where H_r is reaction heat, m and n are reaction orders, and reaction rate constant K_2 is set by

$$K_2 = A \exp\left(-E / RT \right) \tag{3}$$

with frequency factor A, activation energy E and gas constant R, neglecting K_1. Time integration is carried out by means of the generalized Crank-Nicolson method in the framework of finite element method.

Strain is estimated in consideration of non-uniform temperature and resin conversion by phase change [4]. Volume-change ratio is given as

$$\left(\frac{1}{V_0}\frac{dV}{dt}\right)=\left\{\beta_{gel}\left(1-a\right)+\beta_{cured}a\right\}\frac{dT}{dt}-\lambda_{chem}\frac{da}{dt} \tag{4}$$

with thermal expansion coefficient β_{gel} for gel state, β_{cured} for cured state and the maximum polymerization shrinkage factor λ_{chem}. The thermal expansion coefficients are set as linear function with respect to temperature T. The dependency in Young's modulus of resin E_{res} is also taken into account as double exponential function with respect to temperature and degree of cure a [4].

$$E_{\mathrm{res}}\left(T,a\right)=c_1\exp\left(-d_1T\right)c_2\exp\left(-d_2a\right) \tag{5}$$

Relaxation process is simulated by visco-elastic analysis using the Maxwell model for resin under cure progress. The viscosity coefficient is given as linearly proportional to the Young's modulus with coefficient of relaxation time T_g [5].

2.3. FrontCOMP_damage for stress concentration analysis caused by meso-structure

Damage propagation simulator FrontCOMP_damage is an engine for strength analysis by primitive damage laws of resin and fiber bundle. The fiber bundle is handled as linear elastic body with orthogonal anisotropy. The damage propagation in resin is simulated in terms of decreasing stiffness ruled by damage parameter D. The increment of damage parameter ΔD is related to equivalent strain increment $\Delta\bar{\varepsilon}$ normalized by tensile strength σ_{b} and initial Young's modulus E_0.

$$\left(\frac{E_0\Delta\bar{\varepsilon}}{\sigma_{\mathrm{b}}}\right)^2-\left(\Delta D\right)^2=0 \tag{6}$$

The Young's modulus of resin E_{res} starts to decrease when the damage parameter reaches to 1.0. Its decrease rule from initial value E_0 is given by eq.(7) with reduction constant r.

$$E_{\mathrm{res}}=\exp\left(\frac{1-D^r}{r}\right)E_0 \tag{7}$$

Nonlinear behavior of resin is characterized by the constant r, which is set to 0.2. Defects in resin are directly handled in the simulation. Meso-mechanics from initiation to final fracture can be revealed through the simulation.

3. Application for pressure vessel

3.1. Finite element modeling

A feasibility study is demonstrated in problem of fiber bundle/resin system in helical wound CFRP vessels, which is promising for hydrogen tank of fuel cell vehicles or accumulator equipped in hydrogen station. The meso-scopic finite element model is constituted by following helical filament winding process of sixty times reciprocations on aluminum liner of 504 mm total length and 160 mm cylindrical outer diameter. Geodesic bundle passes are searched for by iterative searches of minimum pass length on liner surface, which is descretized by triangle patches. Outer diameter of boss is 56.0 mm. Cross-sectional shape of the bundle is set as ellipsoid with major axis length of 5.0 mm and sub 1.25 mm. Three-dimensional structure of fiber bundle/resin system is embodied with undulation of fiber bundle caused by crossover as shown in Figure 2.

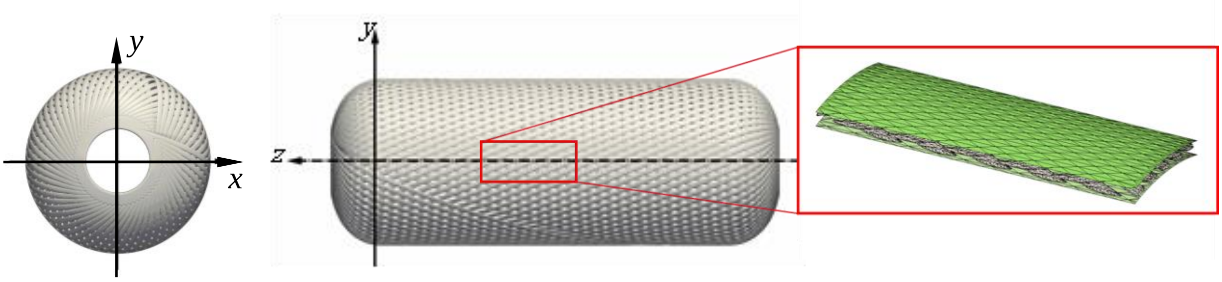
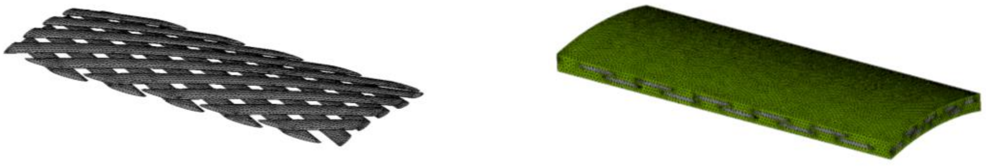


Figure 2. Rectangular analysis region punched out from helical winding bundle model of pressure vessel.

A rectangular part of cylinder is set as analysis region, in which the three-dimensional solid model is automatically discretized by tetrahedral finite elements in the framework of meso-scopic modeling. The thickness of the region is 4.78 mm. Total number of nodes and elements are 262,837 and 180,384, respectively. Discretization of fiber bundle inside and region surface are illustrated in Figure 3.



(a) (b)

Figure 3. Finite element model of analysis region of cylinder part. (a) Inside fiber bundle. (b) Surface.

3.2. Residual stress after cure

Curing process of resin is simulated with material parameters of refs. [4] and [6]. Time-history of ambient temperature is set for thermal boundary condition on upper and lower surface. The temperature rises from $20^{\circ}C$ to $100^{\circ}\mathrm{C}$ by 1.4 hours, and holds three hours. Four edges are handled as thermal insulating surface. Rigid geometrical boundary condition is also set on the four edges in residual stress analysis.

The minimum principal stress after cure is indicated in Figure 4. In the curing process of thick CFRP vessels, excessive temperature rise due to heat generation of the resin appears. The temperature rise seems accurately evaluated for avoiding trial and error approach of temperature control for quality guarantee by means of FrontCOMP. Complicated residual stress distribution raised by fiber bundle crossing is reasonably evaluated by virtue of the meso-scopic modeling.

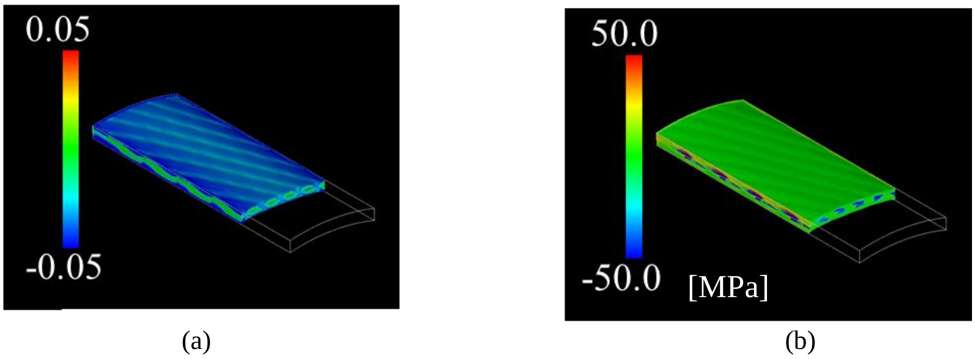


Figure 4. Curing process results. (a) Residual principal strain ε_3. (b) Residual principal stress σ_3.

3.3. Damage propagation by internal pressure

The internal pressure is translated by biaxial stress loading of σ and 2σ for axial and hoop direction as illustrated in Figure 5. The damage analysis is carried out until σ reaches to 45 MPa. Decreased Young's modulus for resin is indicated in Figure 6. Damage area develops between fiber bundle by local stress concentration caused by meso-scopic structure of resin/fiber bundle system.

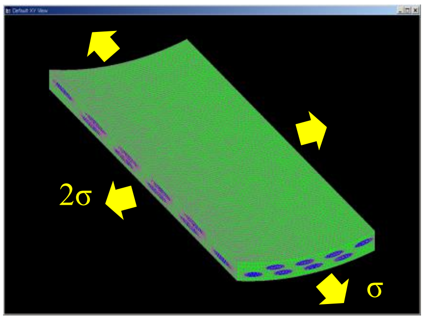


Figure 5. Biaxial loading condition imitating internal pressure loading of vessel.

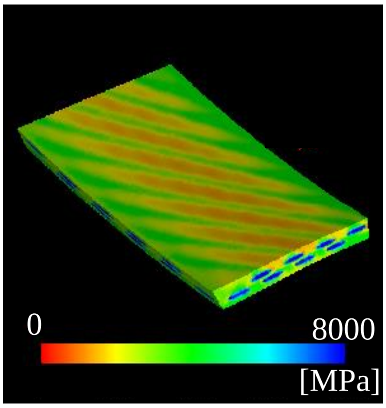


Figure 6. Stiffness reduction by damage propagation.

Legitimate damage models seem given independently for carbon fiber bundle and resin through conventional material tests owing to the meso-scopic modeling. Optimum material design can be carried out in terms of meso-scopic parameters both of carbon fiber bundle and resin for the purpose of strength reliability enhancement. An advantage of high stiffness resin or carbon fiber bundle may be clarified through the damage simulations.

4. Concluding remarks

We have developed a simulation system named FrontCOMP, expecting a reasonable strength evaluation of CFRP members. We can directly set material models for resin and fiber bundle from the result of primitive material tests, by virtue of meso-scopic modeling in terms of fiber bundle and resin. Thermal stress and deformation are correctly evaluated in curing process for optimum temperature control to reduce initial defect. The mechanics of damage propagation can be elucidated through meso-scopic simulation using reasonable material models. Meso-scopic optimum design seems to enhance ultimate strength of a CFRP member by means of meso-scopic structural parameters, such as material non-linearity of resin, size of fiber bundle and so on. The validity of the meso-scopic analysis and design methodology is demonstrated through an analysis of helical wound pressure vessel.

References

[1] J. M. Whitney and M. B. Riley. AIAA Journal, 4(9): 1537-1542, 1966.
[2] Z. Hashin. Journal of Applied Mechanics, 47: 329-334, 1980.
[3] M. R. Kamal and S. Sourour. Polymer Engineering & Science, 13(1): 59-64, 1973.
[4] E. Ruiz and F. Trochu. Composites Part A, 36: 806-826, 2005.
[5] H. Matsui and K. Ikegami. Transactions of the Japan Society of Mechanical Engineers, Series A, 62(596): 993-1000, 1996.
[6] A. Cheung, Y. Yu and K. Pochiraju. Finite Elements in Analysis and Design, 40: 895-912, 2004.