FINITE ELEMENT SIMULATION OF THE DELAMINATION PROPAGATION OF L-SHAPED CFRTP SPECIMEN

M. Akamatsu [a*], T. Matsuo [a], T. Ohori [a], X. Zhang [a] and J. Takahashi [a]

[a] *Department of Systems Innovation, School of Engineering, The University of Tokyo ,7-3-1 Hongo, Bunkyo-ku, Tokyo 113-8656, Japan*
* *akamatsu-mikio@cfrtp.t.u-tokyo.ac.jp*

Keywords: CFRTP, automobile, delamination, finite element simulation

Abstract

FE simulation of CFRP sometimes show different results from the experimental ones. One of the reasons lies in delamination. This research shows a novel FE simulation technique to express the fracture behavior of CFRTP including delamination. Tensile test of L-shaped CFRTP specimen is performed to observe the fracture behavior of CFRTP. FEM model composed of solid elements is able to express delamination. Plastic deformation parameter enables to express the fracture behavior of CFRTP. By verification of mismatch between the experiments and FE simulation, it is considered that defects in specimens are one of the factors of the mismatch as well as the condition of FEM model.

1. Introduction

These days people are interested in eco-vehicles which have low impact on the environment, such as electric vehicles or hybrid vehicles. Along with them, CFRP (carbon fiber reinforced plastics) vehicles are in the spotlight since CFRP has higher specific rigidity and specific strength than steel and aluminum. It is possible to reduce the weight of automobile efficiently using CFRP for the body structure, which leads to an improvement of fuel efficiency and reduction of the impact on the environment. CFRTP (carbon fiber reinforced thermoplastics) are better to use for mass production automobile than CFRTS (carbon fiber reinforced thermosetting resins) by reason of their superior recyclability and shorter molding time.

In the innovative research and design of new automobiles, FEM (finite element method) is often used to investigate the structural characters. However, in the case of CFRP, the results of FE simulation sometimes show differences from experimental results. One of the reasons lies in delamination. As is different from metallic materials, CFRP are likely to delaminate in a curved section and this causes a drastic reduction in flexural stiffness. Hence FE simulation sometimes shows overestimation for flexural stiffness and hence strength and energy absorption capacity as well. There have been many research about delamination phenomenon [1][2], however few researches mentioned how to express delamination in FEM.

This research shows a novel FE simulation technique to express the fracture behavior of CFRTP including delamination. L-shaped tensile specimen, on which delamination can be observed through choosing a proper radius for curved section [3], is used to verify the developed simulation technique.

2. Tensile test of L-shaped CFRTP specimen

2.1. Materials and specimens

This research deals with UD (uni-directional) materials. L-shaped specimen is made of UD pre-preg sheet produced by Mitsubishi Rayon Co., LTD. using carbon fiber TR50S. Polypropylene is applied as matrix resin and the fiber volume fraction of the UD pre-preg is 45%.

The dimension of the specimen is shown in Figure 1. Width (15 mm) and thickness are constant and inner radius in the curved section is varied as 10, 15, 20 mm.

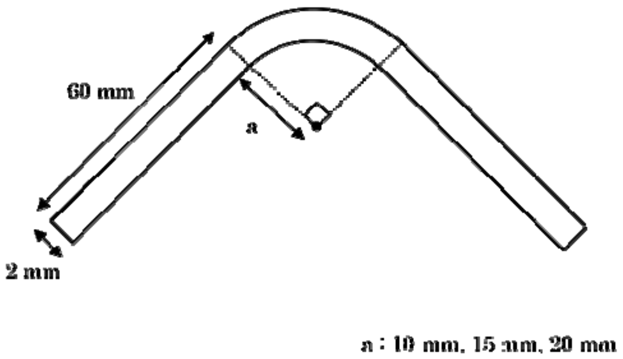

Figure 1. Schematic of L-shaped CFRTP specimen.

2.2. Methodology of L-shaped tensile test

L-shaped specimen is fixed and pulled by using special jigs as shown in Figure 2. The stroke speed is 1 mm / min. The fracture behavior and the load-stroke curves are observed.

When an L-shaped specimen is pulled as shown in Figure 3, stresses in circumferential direction and in out-of-plane direction are applied to the specimen. Circumferential tensile stress can cause tensile fracture in the inner side of curved section while circumferential compressive stress can do compressive fracture in the outer side. Tensile stress in Out-of-plane direction can cause delamination in the central part of curved section. In this test, the fracture mode can be observed, and the first fracture mode can be triggered by adjusting specimen’s dimension. Otherwise, load and stroke are measured at the crossheads so that the load-stroke curves can be obtained.

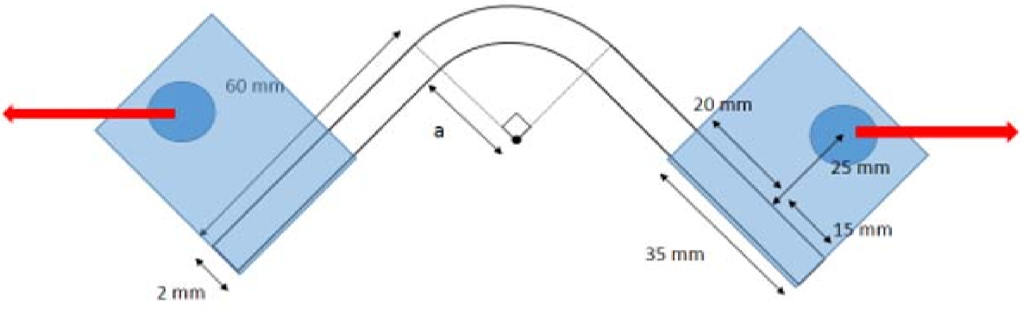

Figure 2. Schematic of the jigs.

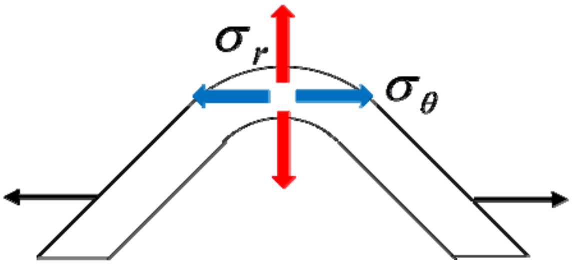

Figure 3. Stresses applied in L-shaped specimen.

2.3. Results

In the case of specimens of which inner radius are 10 and 15 mm, first fracture mode were delamination. Appearance of the side of the specimens are shown in Figures 4-5. Following the first fracture, delamination propagation and compressive fracture occurred at random.

In the case of specimen of which inner radius is 20 mm, first fracture mode was compressive fracture in the outer curved section. The appearance of the surface of a specimen is shown in Figure 6.

Load-stroke curves of each radius are shown in Figures 7-9. What is interesting is that the curves rise after the first fracture. This behavior is due to the ductility and the efficiency of energy absorption of polypropylene, and is not observed in CFRTS. In addition, in the case of specimens of which inner radius are 15 and 20 mm, the gradient of load-stroke curves decreased before the first fracture.

These results show not only the tendency that the smaller the inner radius is, the more delamination is likely to occur, but also the ductility and the yielding phenomenon of CFRTP.

Figure 4. Delamination in a specimen with an inner radius of 10 mm.

Figure 5. Delamination in a specimen with an inner radius of 15 mm.

Figure 6. Compressive fracture on a specimen with an inner radius of 20 mm.

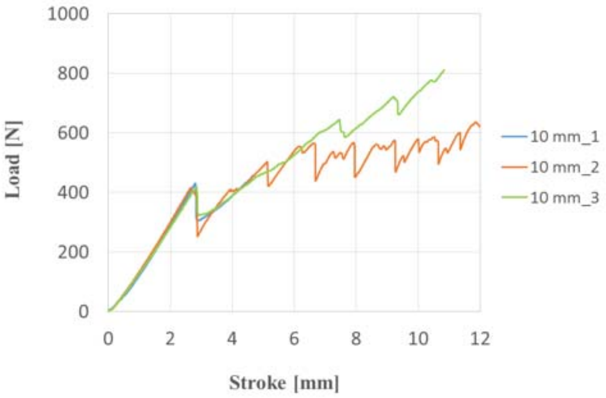

Figure 7. Load-stroke curves with an inner radius of 10 mm.

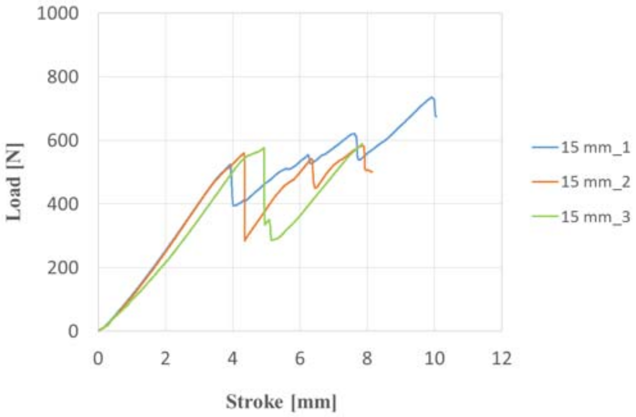

Figure 8. Load-stroke curves with an inner radius of 15 mm.

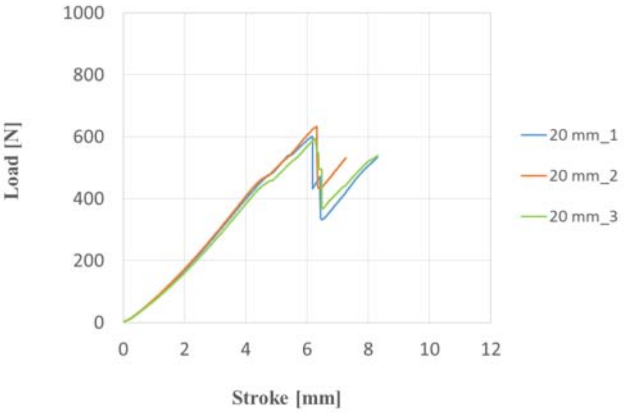

Figure 9. Load-stroke curves with an inner radius of 20 mm.

3. FE simulation

3.1. FEM model setting

By establishing FEM model of the tensile test of L-shaped specimen and comparing the results of FE simulation with those of experiments, validity of the model and requirements for simulation of CFRTP materials are verified. HyperMesh was used as pre-processor and Hyper View as post-processor.

This research deals with solid elements instead of shell elements. Because solid elements calculate out-of-plane stress which contributes to delamination while shell elements don't. Linear element was used to calculate. The number of mesh in out-of plane direction was 20 in order to improve the accuracy of calculation of out-of-plane direction stress which contributes to delamination.

An L-shaped specimen has 2 planes of symmetry. Therefore, the FEM model used in this study was 1/4 part of the actual shape in order to shorten the calculation time.

LS-DYNA and RADIOSS Block were used as solvers. MAT COMPOSITE FAILURE SOLID MODEL (MAT59) and COMPOSITE SOLID MATERIAL (law14) were selected as material card in LS-DYNA and RADIOSS block, respectively. Both cards are adequate for simulation of composite materials.

Main parameters of material properties set in each material card are shown in Table 1. These parameters are given by Mitsubishi Rayon Co., LTD. or from a previous research by Wan et. al. [3]. In the case of composite materials, anisotropy should be concerned. RADIOSS block (law14) provides a parameter about plastic deformation and also compressive yield stress instead of compressive strength.

Degree of freedom in the planes of symmetry was constrained. Part of the specimen fixed by jigs was constrained by rigid body and set velocity of 10 mm / min at the tip of the rigid body. Whole image and the size of the FEM model are shown in Figure 10. Load and stroke are measured at the tip of the rigid body.

	LS-DYNA (MAT59)	**RADIOSS block (law14)**
Fiber direction elastic modulus [GPa]	101	101
Out-of-plane direction elastic modulus [GPa]	4	4
In-plane direction Poisson's ratio	0.34	0.34
In-plane direction shear modulus [GPa]	1.63	1.63
Fiber direction compressive strength [MPa]	700	
Fiber direction compressive yield stress [MPa]		700
Out-of-plane direction tensile strength [MPa]	23	23
Plastic deformation parameter		1.0

Table 1. Material property parameters set in FEM models (Fiber direction elastic modulus, out-of-plane direction elastic modulus, in-plane direction Poisson's ratio and in-plane direction shear modulus are given by Mitsubishi Rayon Co., LTD. Fiber direction compressive strength and out-of-plane direction tensile strength are from the research by Wan et.al.).

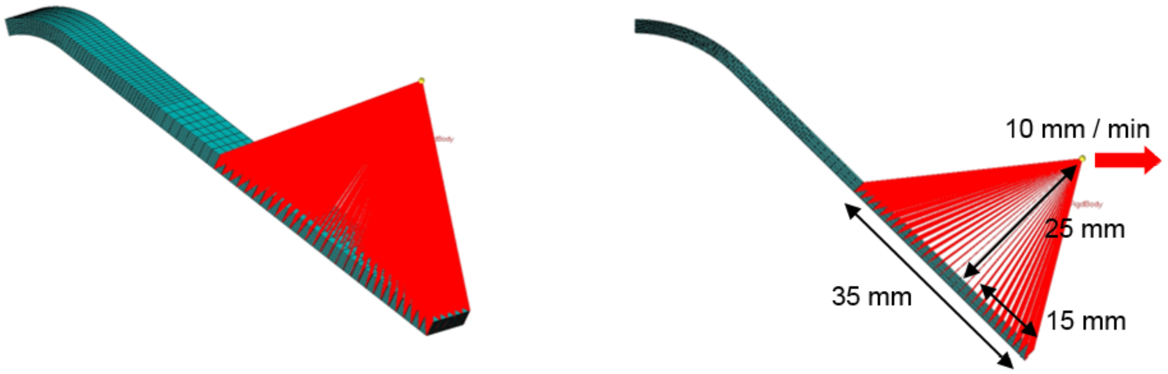

Figure 10. Whole image (left) and the size (right) of the FEM model.

3.2. Results

Contour diagrams of stresses are shown in Figure 11. Red color shows tensile stress and blue color shows compressive stress. Delamination is expressed as out-of plane tensile fracture in FE simulation. Delamination mode in FEM model is shown in Figure 12.

First fracture modes of FE simulations were compared with those of experiments for each inner radius. For every inner radius and for every solver, the first fracture mode agree with the experiments as shown in Table 2.

Load-stroke curves calculated by FE simulation are shown in Figure 13 with the results of the experiments. RADIOSS block (law14) calculates the load after the first fracture since it can deal with plastic deformation. But the gradient and the peak of the curves didn’t agree with the result of the experiments. Some parameters set in the FEM model should be reconsidered.

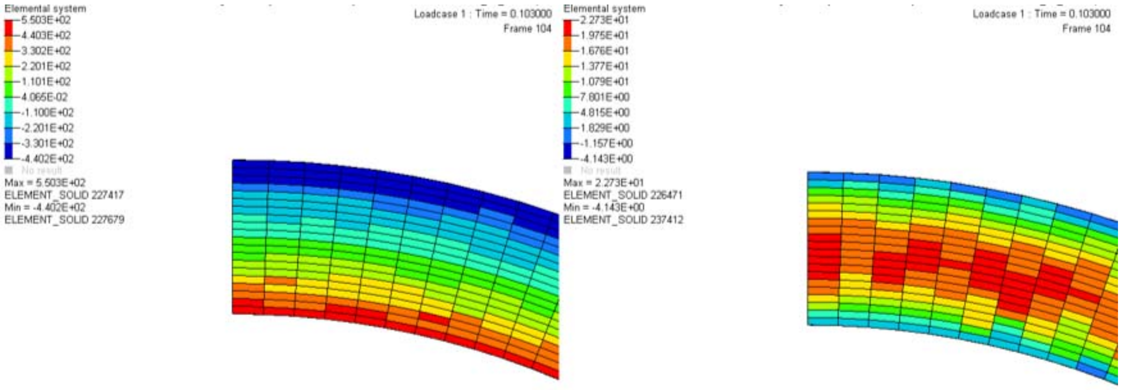

Figure 11. Circumferential (left) and out-of-plane direction (right) stress of L-shaped specimen calculated by FEM (inner radius: 10 mm / LS-DYNA).

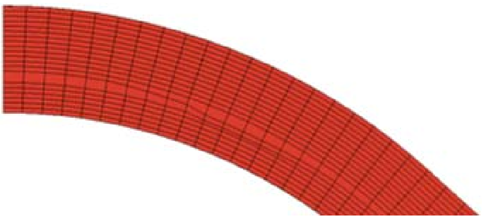

Figure 12. Delamination mode in FEM model (inner radius: 10 mm / LS-DYNA).

		Experiment	**LS-DYNA**	**RADIOSS block**
Inner radius	**10 mm**	Delamination	Delamination	Delamination
	15 mm	Delamination	Delamination	Delamination
	20 mm	Compressive fracture	Compressive fracture	Compressive fracture

Table 2. The first fracture mode comparison between the experiments and FEM models.

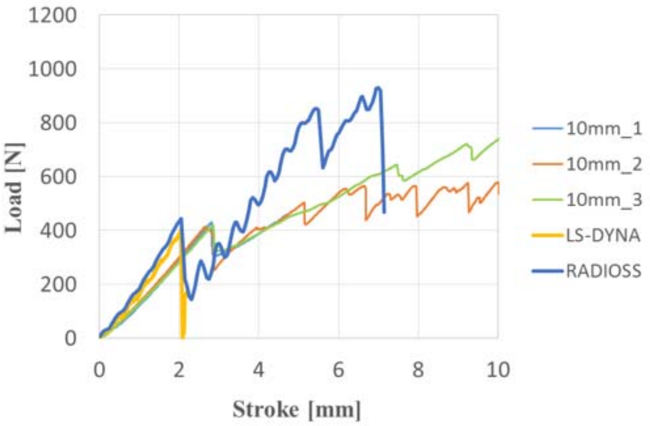

Figure 13. Load-stroke curves of the experiments and FEM (inner radius: 10 mm).

4. Discussions of mismatch

The mismatch in load-stroke curves between FE simulation and experiments should be solved.

First, the FEM model was verified by the reformed FEM model in which constraint was changed. Components of steel jigs and condition of contact between the specimen and the jigs were considered in the reformed FEM model. The image and the result is shown in Figure 14. The gradient of the load-stroke curve was more similar with experimental results than the original FEM model.

Secondly, L-shaped specimen itself was verified. The inside of a specimen was observed by X-rays CT scans and many voids were observed in the curved section (Figure 15), which may affects the result of the experiments. It is difficult to make CFRTP materials which have no void in the curved section. It is necessary to specify the level of defects in specimens which don’t affect the result of experiments and to establish methods to observe the defects as well.

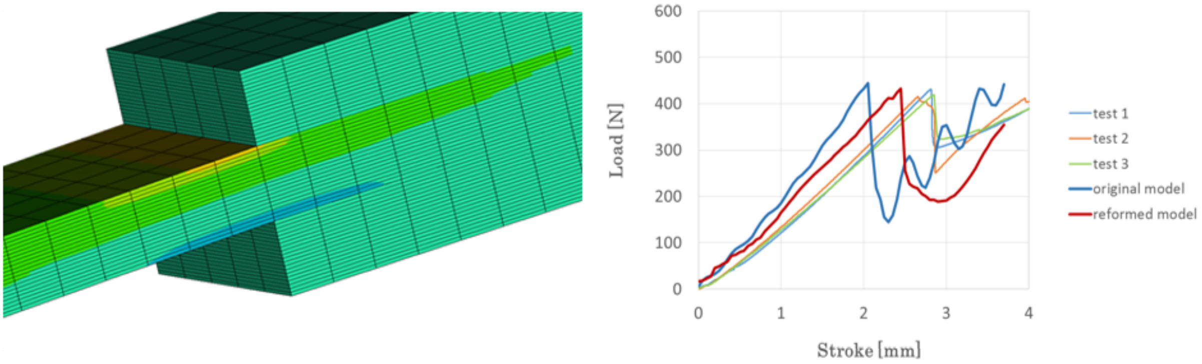

Figure 14. The image (left) and the load-stroke curve (right) of reformed FEM model.

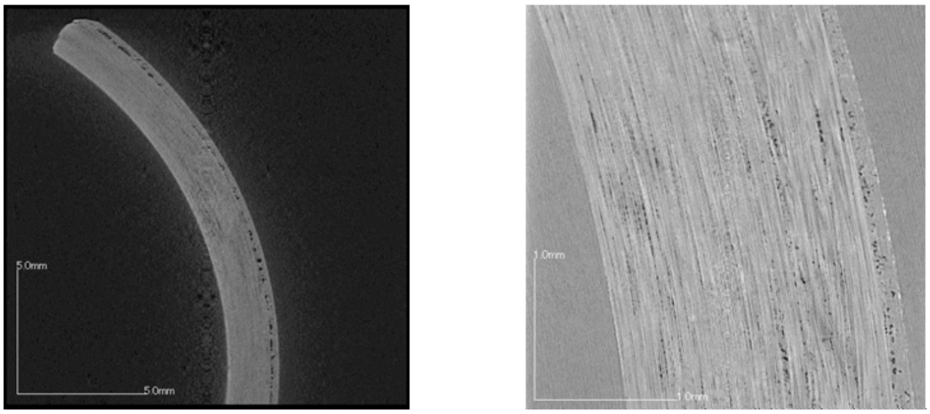

Figure 15. The side view of L-shaped specimen (left) and its zoomed image (right) by X-rays CT scans.

5. Conclusions

Tensile tests of L-shaped CFRTP specimen showed that the gradient of load-stroke curves rises after first fracture, which shows the ductility of thermoplastics. This cannot be observed in the case of CFRTS and a new knowledge about the fracture behavior of CFRTP was obtained. It is suggested that this kind of behavior should be considered in FE simulation.

In FE simulation concerning delamination, out-of-plane direction stress should be considered, so this research approached with solid elements instead of shell elements. Anisotropy, plastic deformation and yielding should be considered as well and this enabled to express delamination and fracture behavior of CFRTP in FE simulation.

Regarding to the mismatch between the results of the experiments and FE simulation, defects in the curved section of L-shaped specimen was likely to be one of the reasons as well as configuration of FEM models. It is necessary to make countermeasures by CAE, such as specification of the defect level which doesn’t affect the results of experiments, or improvement of FEM method to evaluate the effect of defects in materials.

Acknowledgement

This study was conducted as a part of Japanese METI project “the Future Pioneer Projects / Innovative Structural Materials Project” since 2013fy. Authors would like to express sincerely appreciation to the project members who have provided valuable information and useful discussion.

References

[1] W. C. Jackson and R. H. Martin, An Interlaminar Tensile Strength Specimen, *Composite Materials, Testing and Design (11), ASTM STP 1206. E*, 333-354, T. Camponeschi Jr., Ed, American Society for Testing and Materials. Philadelphia, 1993.

[2] I. Tawk, P. Navarro, J.-F. Ferrero, J.-J. Barrau and E. Abdullah, Composite delamination modelling using a multi-layered solid element, *Composites Science and Technology 70(2)*, 207-214, 2010.

[3] Y. Wan, T. Goto, T. Matsuo, J. Takahashi and I. Ohsawa, Investigation about fracture mode and strength in curved section of carbon fiber reinforced polypropylene, in *19th international conference on composite materials*, 6695-6702, 2013.