Simulation of ball impact on composite plate with PP+30% LGF

Dr. T. Sakakibara[1], Dr. R. Akita[1], Dr. Y. Ohnishi[1]
S. Kijima[2], Y. Kanki[2]
Dr. M. Seto[3]
K. Suda[4], K. Yamakawa[4], Y. Ayano[4]

[1]*ITOCHU Techno-Solutions Corporation, 3-2-5, Kasumigaseki, Chiyoda-ku, Tokyo, Japan*
[2] *UES Software Asia Inc., 1-6-9, Furou-cho, Yokohama, Kanagawa, Japan*
[3] *Kanazawa Institute of Technology, 3-1, Yatsukaho, Hakusan, Ishikawa, Japan*
[4]*Toray Engineering Co., Ltd, 1-1-1, Sonoyama, Shiga, Japan*

Abstract

The failure prediction of the long glass-fiber (LGF) reinforced resin is difficult because of the complicated fiber orientation compare to the short fiber. In order to establish a simulation procedure to represent the failure behavior of the LGF reinforced resin, we have carried out simulations of ball impact on the composite plate with polypropylene and 30% LGF using LS-DYNA® coupled to DIGIMAT. The fiber orientation was calculated by the mold flow code 3D TIMON and the composite material properties and failure criterion were estimated and verified by DIGIMAT based on mean-field homogenization. The stiffness reduction and fracture progress in the simulation are represented by the First Pseudo Grain Failure model (FPGF). The numerical results of the impact simulation are discussed through comparing the experimental results.

Introduction

The glass-fiber (GF) reinforced resin is recently popular in the manufacturing industry. The demands of the numerical simulation for the GF reinforced resin or the composite materials are gradually increasing. However, it is difficult to analyze the failure behavior of the resin with fiber accurately. Because, the fibers in matrix tangle so that it is necessary to take into account the fiber orientation in the simulation. In order to establish a computation technique for resin materials, we performed experiments and simulations of ball impact on composite plate with PP+30% long glass-fiber. Firstly, test specimens of the dumbbell and product specimens are made for the tension test and the ball impact experiment. Secondary, resin flow analyses are performed to determine the orientation of the fiber by injection molding simulation code 3D TIMON. After that, the properties of the specimens are identified by DIGIMAT-MF which compares to the experimental results. Finally the coupling analyses by LS-DYNA and DIGIMAT are executed and compare to the experimental results.

Experiment

Figure 1 shows the experimental piece geometry for tensile tests. Two types of experimental pieces are prepared. One is the MD which means that the tensile direction of the tension is parallel to the flow direction. The other is the TD which means that the tensile direction is perpendicular to the flow direction. Figure 2 shows the SEM photograph of the thick cross section expanded by 60 times. It is visible to three-layer structure in the cross section. The orientations of the fiber in the skin layer turn the flow direction of the resin. On the other hand, that of the core layer turns perpendicular to the flow direction. Figure 3 shows a series of tensile experimental results.

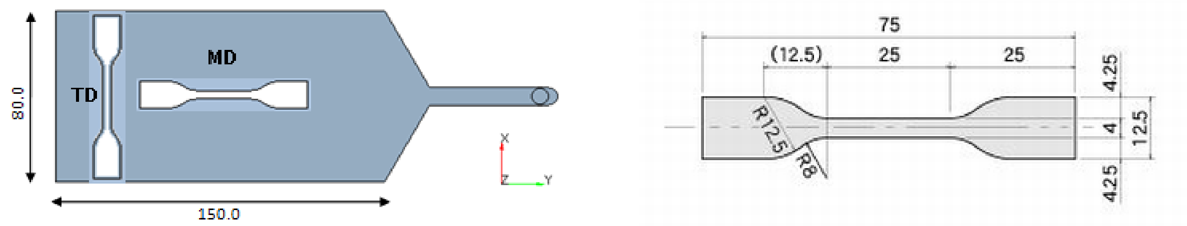


Figure 1 Dimension of the dumbbell specimen

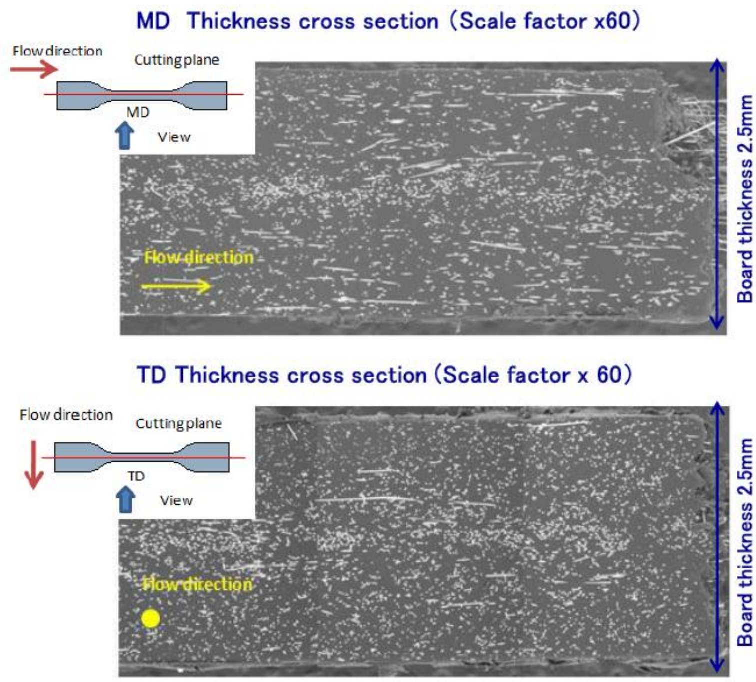
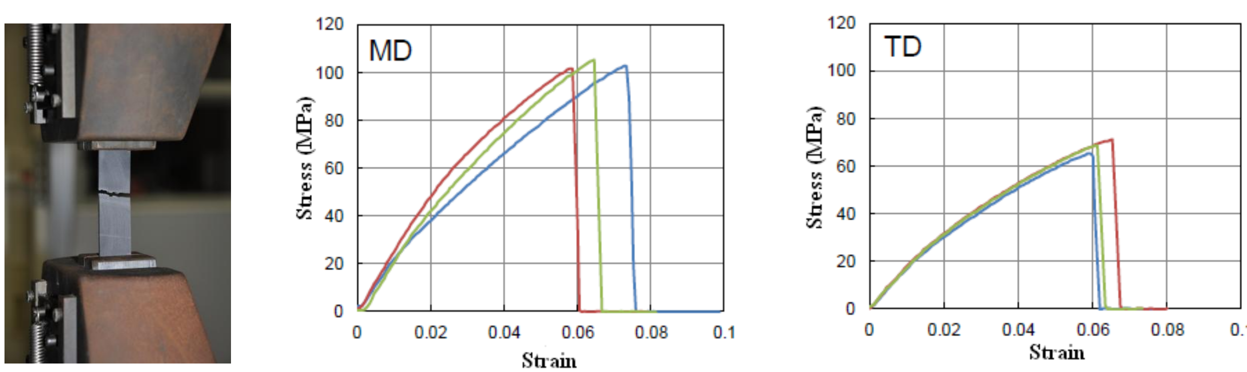


Figure 2 Fiber orientation by SEM



(a) Photograph of test **(b) Stress-strain curves of MD type** **(c) Stress-strain curves of TD type**

Figure 3 Results of tensile tests

The geometry of the experimental specimen for a ball impact test is shown in Figure 4 and the photograph of the test equipment and specimen are shown in Figure 5. A weld line can be seen in photograph clearly. In the experiment, we prepare two types of specimens, one is a specimen with weld and the other is that without weld. Figure 6 shows experimental results of the force history due to a ball impact on each type of specimens. The fracture was occurred at 1.82 sec in case of specimen with weld shown in Figure 6(a). The fracture was not observed in the specimen without weld.

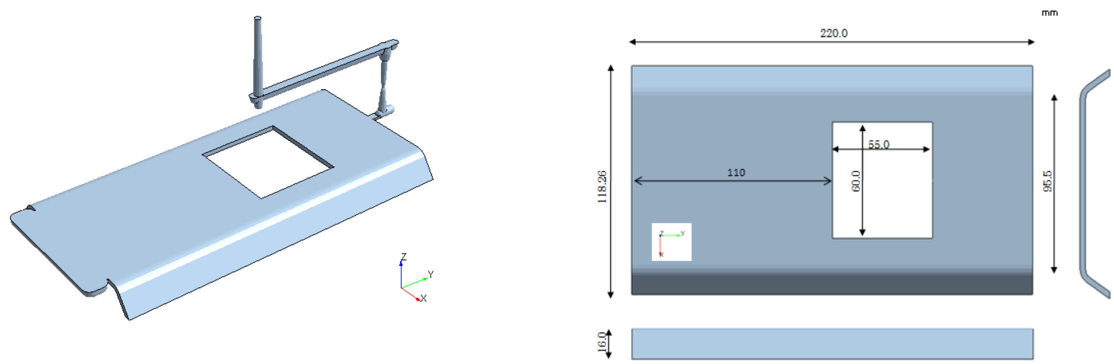
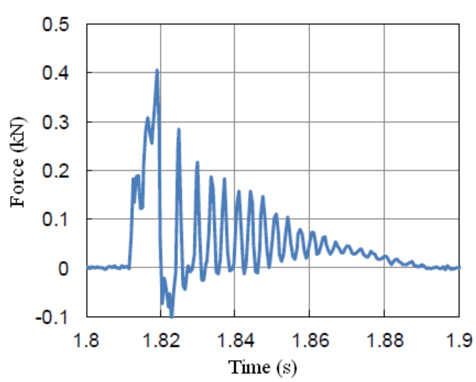
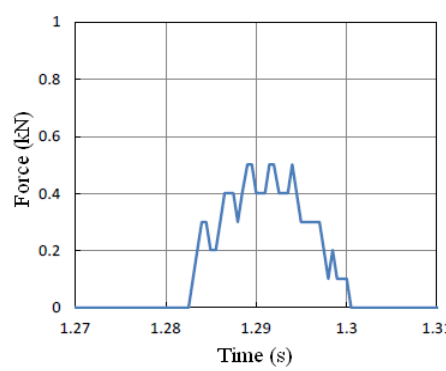


Figure 4 Geometry of the specimen

Figure 5 Impact test equipment and the photograph of the specimen



(a) with weld



(b) without weld

Figure 6 Force history

Identification

The model properties of the specimen required for FE analysis are identified by comparing with the experimental values using DIGIMAT-MF. The number of layers in a cross section is set to 3, they are called as skin layer, core layer, and skin layer respectively. Thickness and intensity of orientation of each layer are described in Table 1.

Table 1 Layer geometry

Multi-layer model	Thickness (2.5mm)	Intensity of orientation [a_{11}-a_{22}-a_{33}]	
		MD	TD
Skin layer	0.85mm×2	0.7 - 0.3 - 0.0	0.3 - 0.7 - 0.0
Core layer	0.8mm	0.3 - 0.7 - 0.0	0.7 - 0.3 - 0.0

Figure 7 shows the comparison of stress-strain curves between calculation results by DIGIMAT-MF and the experimental results. Both types of computed results are almost identical with the experiment.

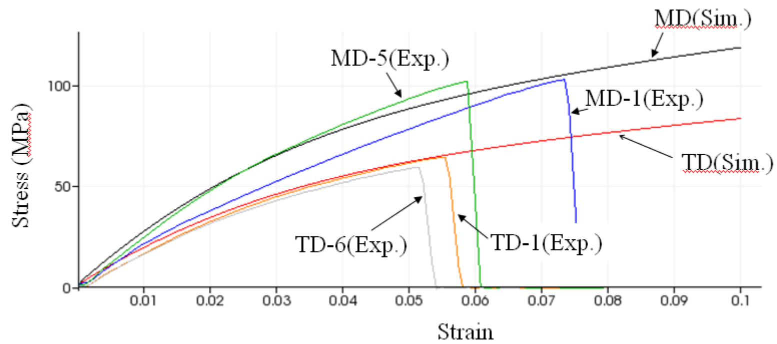


Figure 7 Stress-strain curves

Ball Impact Simulation

Ball impact simulations are carried out by using DIGIMAT and LS-DYNA. The FE model of specimen is shown in Figure 8. A rigid ball due to gravity impacts on a composite plate with initial velocity 2359 mm/sec which is equivalent to 284mm height free fall. The composite plate consists of Polypropylene (PP) and Long Glass-Fiber (LGF). Material properties described in Table 2 are identified by DIGIMAT-MF.

First Pseudo Grain Failure (FPGF) model implemented on DIGIMAT is applied to represent the stiffness reduction after starting failure and the finally occurred fracture. Required parameters of FPGF are obtained after trial and error during the tensile test simulation. They are described in Table 3

In order to take in to account the orientation of the fibers in composite plate, it is necessary to couple LS-DYNA and DIGIMAT. The identified parameters are generated by DIGIMAT-MF as *MAT_USER_DEFINED_MATERIAL_MODELS automatically.

Figure 9 shows results of the specimen with weld. Fracture occurred in weld line is shown in Figure 9(a) and the force history due to the ball impact is shown in Figure 9(b). The solid curve indicates the simulation result and the broken curve indicates experimental result. The time length from impact start to fracture occurrence of the simulation is almost same as the experiment. Although the oscillation is observed in the force history in the simulation, the result of the simulation is almost identical to the experimental result.

Table 2 Material Properties

	Young's modulus (MPa)	Poisson's ratio (-)	Mass density (ton/mm^3)	Material type
PP	1300	0.2	1.21E-9	Elast-viscoplastic
LGF	72000	0.4	2.54E-9	Elastic

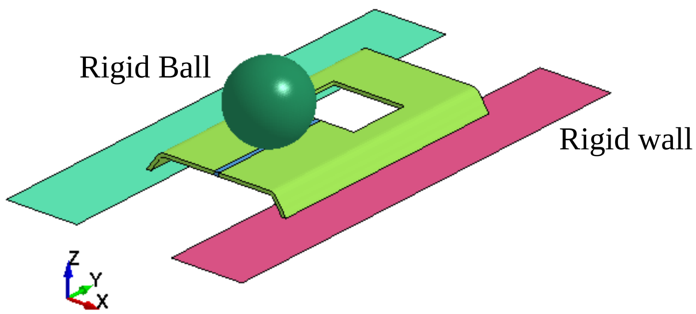
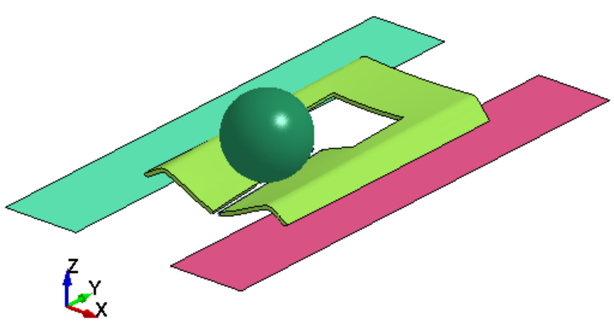
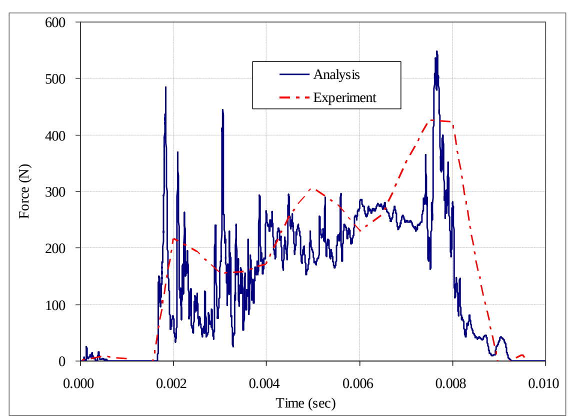


Figure 8 Ball impact model

Table 3 FPGF parameter

	Weld line	Matrix
Fiber direction	0.0315	0.0315
Right angle to fiber direction	0.023	0.05





(a) Fracture **(b) Force history**

Figure 9 Simulation result of weld type

Conclusion

In order to predict the behavior of resin materials with glass fiber, we performed experiments and simulations of tensile tests and ball impact tests. The properties of the specimens were identified from experimental results. FPGF method implemented on DIGIMAT was applied to represent fracture propagation. The ball impact simulations were carried out by using LS-DYNA and DIGIMAT. The results of the simulation were identical to the experimental.

References

[1] DIGIMAT Software, e-Xstream engineering, Louvain-la-Neuve, Belgium. http://www.e-xstream.com
[2] L. Adam, A. Depouhon & R. Assaker, Multi-Scale Modeling of Crash & Failure of Reinforced Plastics Parts with DIGIMAT to LS-DYNA Interface, 7th European LS-DYNA Users' Conference.
[3] J. Seyfarth, M. Hormann & R. Assaker, Taking into Account Fiber Reinforcement in Polymer Materials: the Nonlinear Description of Anistropic Composites via the DIGIMAT to LS-DYNA Interface, 7th European LS-DYNA Users' Conference.