Application and CAE Simulation of Over Molded Short and Continuous Fiber Thermoplastic Composites: Part I

Prasanna S. Kondapalli
BASF Corp., Wyandotte U.S.A

Kipp Grumm
BASF Corp., Wyandotte U.S.A

Abstract

Short Fiber Reinforced Thermoplastics (SFRT) such as glass filled Polyamide 6 and 66 have been widely adopted as a metal replacement in a wide range of industries. The main advantage of using these materials is high strength to weight ratio, light weight, parts consolidation, easy manufacturability etc. Continuous Fiber Reinforced Thermoplastics (CFRT) are also gaining popularity because of its ability to achieve high directional stiffness/strength by tailoring the number of layers and angles. Applications which combine these two by over molding SFRT on CFRT inserts are still in its infancy. One of the hurdles is the lack of good CAE simulation capability for such applications. This paper describes the CAE tools that are developed using LS-DYNA to successfully model static and dynamic behavior of such parts. Material 58 in LS-DYNA is used for modeling the CFRT material while a User Defined Material Law models the SFRT material and they are coupled together through suitable contact definitions. Its applicability is verified through a number of examples varying from very simple to complex configurations

Introduction

Short fiber reinforced thermoplastics (SFRT) such as glass filled Polyamide 6 and 66 are widely used in the Automotive Industry as a metal replacement. A number of automobile components such as air intake manifolds, valve covers and increasingly oil pans are made of these materials. Some of the benefits of these materials are high strength to weight ratio, lower density compared to metals, flexibility, manufacturability and parts integration. Continuous fiber reinforced thermoplastics (CFRT) is well established in the aerospace industry due to its high stiffness & strength to weight ratio. They can also be tailored to have high stiffness & strength in certain directions by optimizing number of layers and fiber angles. They are increasingly being adopted in the automotive industry because of the various ongoing light weighting initiatives in the industry. This will only accelerate in order to meet the new CAFÉ (Corporate Average Fuel Economy) standards of almost 55 mpg in 2025. By combining the two i.e., by over molding SFRT (Polyamide 6/66) over CFRT inserts the complementary advantages of these two materials can be exploited. One of the challenges is CAE simulation of such parts. In this paper, finite element analysis (FEA) techniques to accurately model over molded short and continuous fiber thermoplastic composites are described. Many examples are also given to validate the FEA methodology.

SFRT (Glass Filled Polyamide 6)

Injection molding is the most commonly used method for making SFRT parts. The properties of the part are dependent on the fiber orientation which in turn is affected by gating locations and other process parameters. The most common approach for modeling these in FEA is to use isotropic properties based on testing of tensile samples. These tensile samples have highly aligned fibers and tend to overestimate stiffness and strength. BASF has developed a new methodology called ULTRASIM® [1], [2] which uses an anisotropic material model based on glass fiber orientation. The first step is to carry out a moldflow analysis of the injection molding process which simulates the manufacturing. The glass fiber orientation at various locations of the part is obtained as an output from moldflow analysis software. Extensive characterization of BASF materials has also been carried out for various conditions. This includes testing for various fiber orientation, strain rates, temperatures and moisture content. The above information is used to create an anisotropic material model. Some of the features of the material model are nonlinearity, strain rate dependency, asymmetry in tension & compression and an advanced failure criteria. The material model is implemented as a USER DEFINED MATERIAL LAW in LS-DYNA.

An example showing the application of this material model is described here. An oil pan made of Ultramid A3WG7 (Polyamide 66) is shown in Figure 1. It is has a single gate at the top for mold filling and the FEA model is generated by use of ULTRASIM® based on fiber orientation data from Moldflow software. The FEA model is based on a mid-plane shell model with five integration points through the thickness. The fiber orientation variation is also captured through the thickness. A modal analysis is performed on the oil pan FEA model by fixing it at the bolt locations. A test was also conducted by mounting the part on a shaker and doing a frequency sweep. A comparison of the mode shapes (iso-contour) from the test and FEA is shown in Figures 2 & 3. The modal frequency values from the test and the FEA are also within 2 %. Both show excellent correlation.

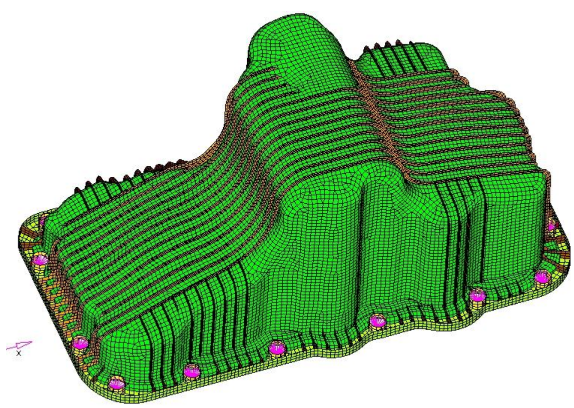
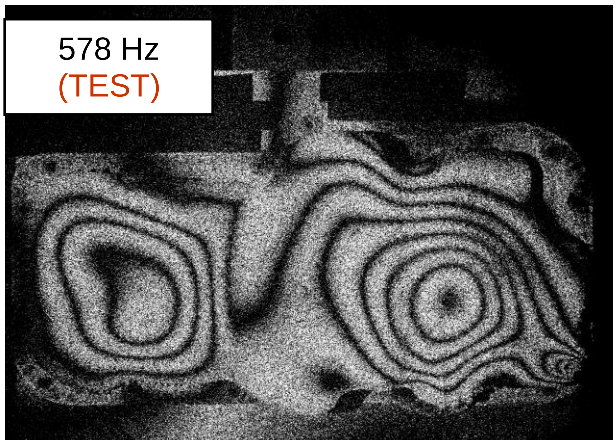


Figure 1 Plastic Oil Pan made of Ultramid A3WG7 (Glass Filled Polyamide 66)



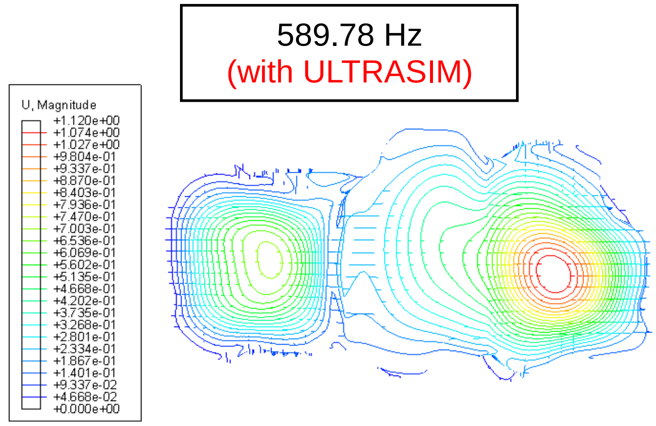
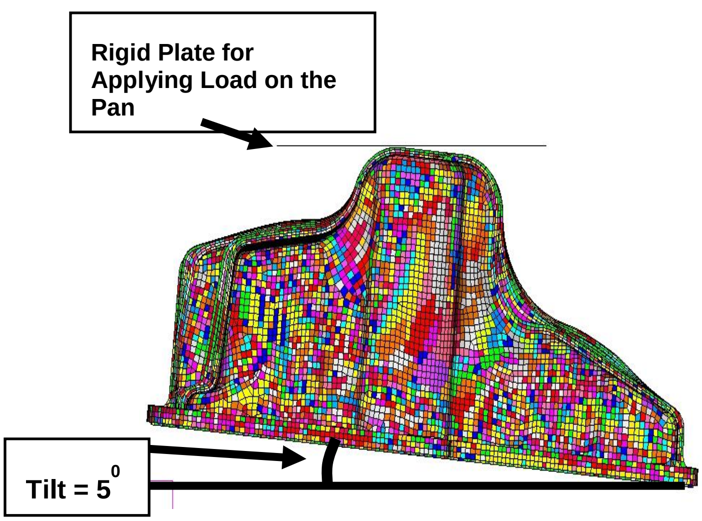


Figure 2 Mode Shape & Frequency from Test

Figure 3 Mode Shape & Frequency from FEA

A quasi-static crush load is also carried out on a tilted oil pan using a rigid plate as shown in Figure 4. The various colors in the FE model are a representation of the anisotropy in the model. The FE model is generated using ULTRASIM® and the quasi-static analysis is carried out in LS-DYNA. Figure 5 shows the location of the crack after the test. In the FE analysis as shown in Figure 6 shows the initiation of the crack in the same location as the test.



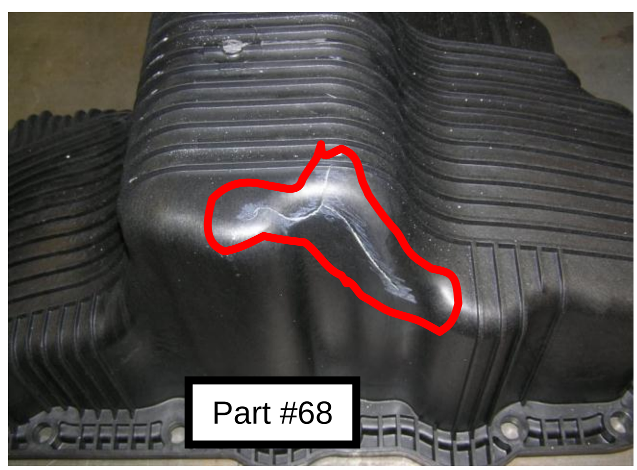


Figure 4 Crush Load on Oil Pan

Figure 5 Oil Pan showing Initial Crack

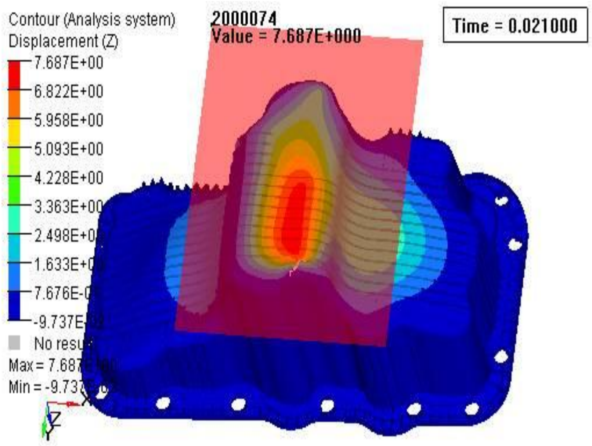


Figure 6 FEA Simulation of Crush Load

Displacement of Plate at Initial Crack		Reaction Force at Initial Crack		Internal Energy at Initial Crack	
Test	CAE	Test	CAE	Test	CAE
6.0 mm	6.55 mm	13.8 kN	13.2 kN	36.5 J	40 J

Table I Comparison of FEA and Test Data

Table I shows a comparison of displacement, reaction force and internal energy calculated from the test and FEA. The values compare very well.

CFRT Laminate

The CFRT laminate that was developed has unfilled Polyamide 6 as the matrix material/binding agent and continuous fibers made of glass. These parts are normally thermoformed after a kitting process. There are various material models within LS-DYNA for modeling CFRT [3], [4],[5],[6]. MAT_58 (*MAT_LAMINATED_COMPOSITE_FABRIC) is one of the widely used material models for CFRT laminates. In [4], detailed discussion about the material models and their advantages and disadvantages is given. In the present case, MAT_58 is adopted for modeling the CFRT laminate. The procedure as discussed in [4] is followed to estimate the various parameters. In order to get an initial estimate of these parameters in MAT_58, a series of tests were conducted on uni-directional layers based on ASTM standards. The tests that were conducted were fiber tension, fiber compression, matrix tension, matrix compression and shear. The results from these tests formed the basis of MAT_58 cards. LS-OPT was also used to estimate some values by fitting some test results. Tests were also conducted on laminates of different lay ups to verify and adjust estimated values in MAT_58. The goal was to predict the softening and initial failure and not so much the post failure response which is a very difficult proposition. Some of the tests that were conducted are shown in Figures 7-12. Tests were done on laminates with (0-90-0)sym lay ups in tension & compression along particular orientations. Results comparison of the test and FEA are also listed and show good correlation.

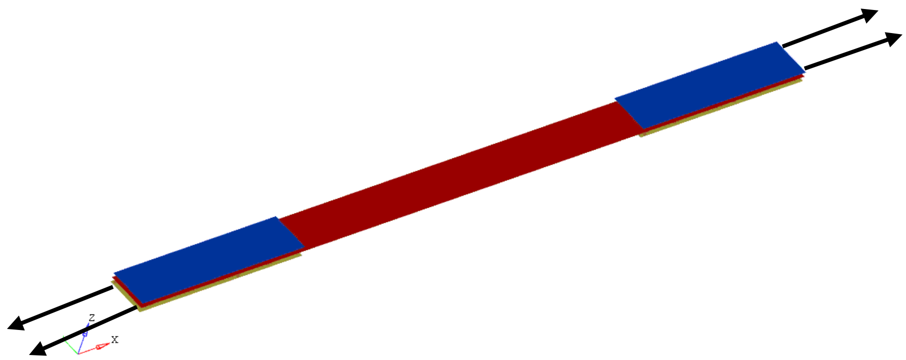


Figure 7 CFRT Laminate with (0-90-0)sym layup, tensile test along 0^0 Orientation

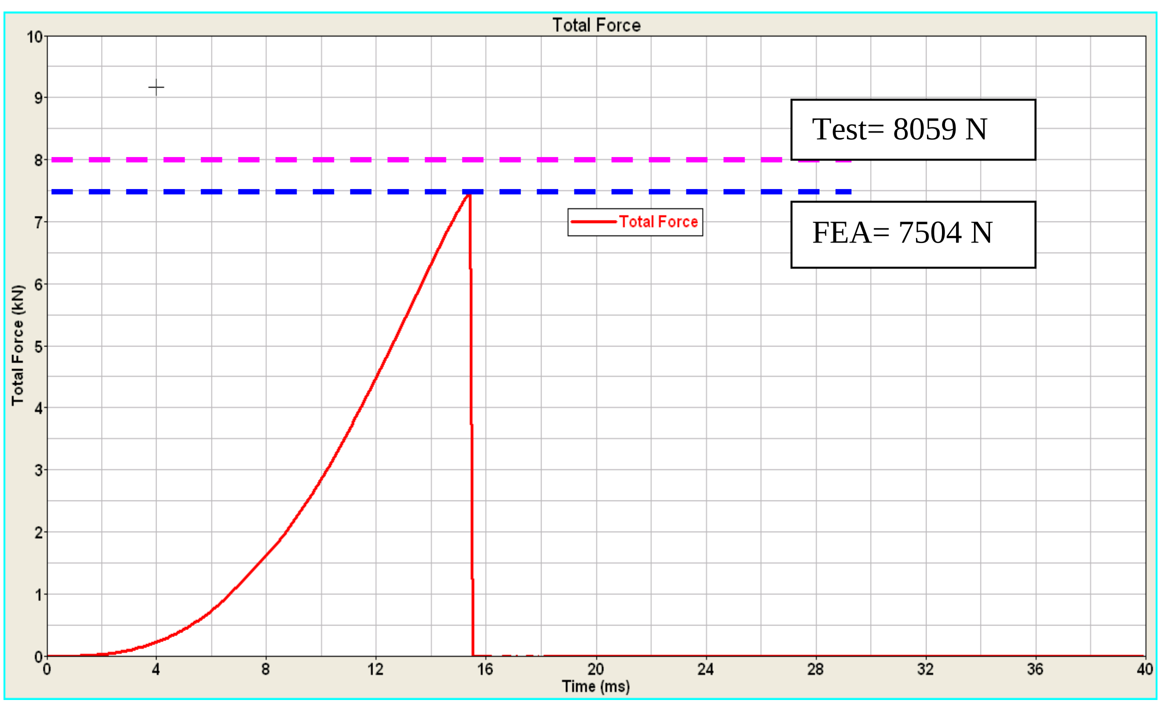


Figure 8 Comparison of Force from Test and CAE

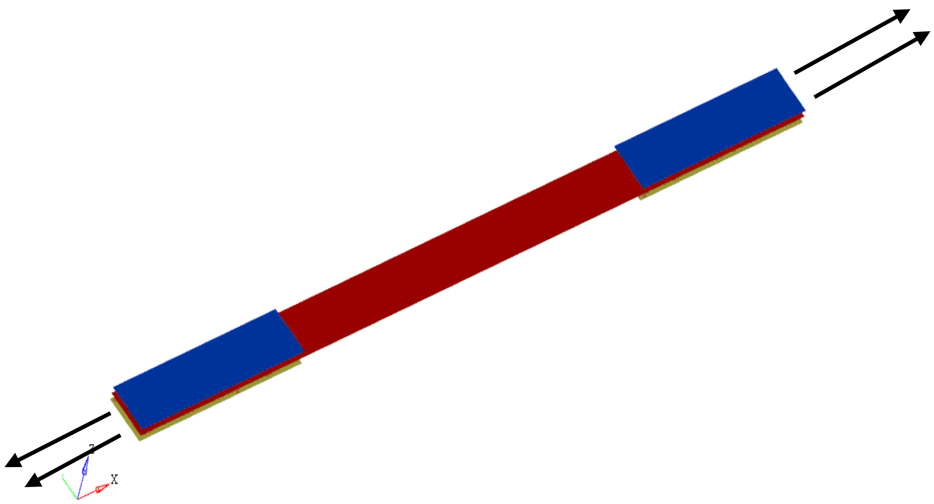


Figure 9 CFRT Laminate with (0-90-0)sym layup, tensile test along 90^0 Orientation

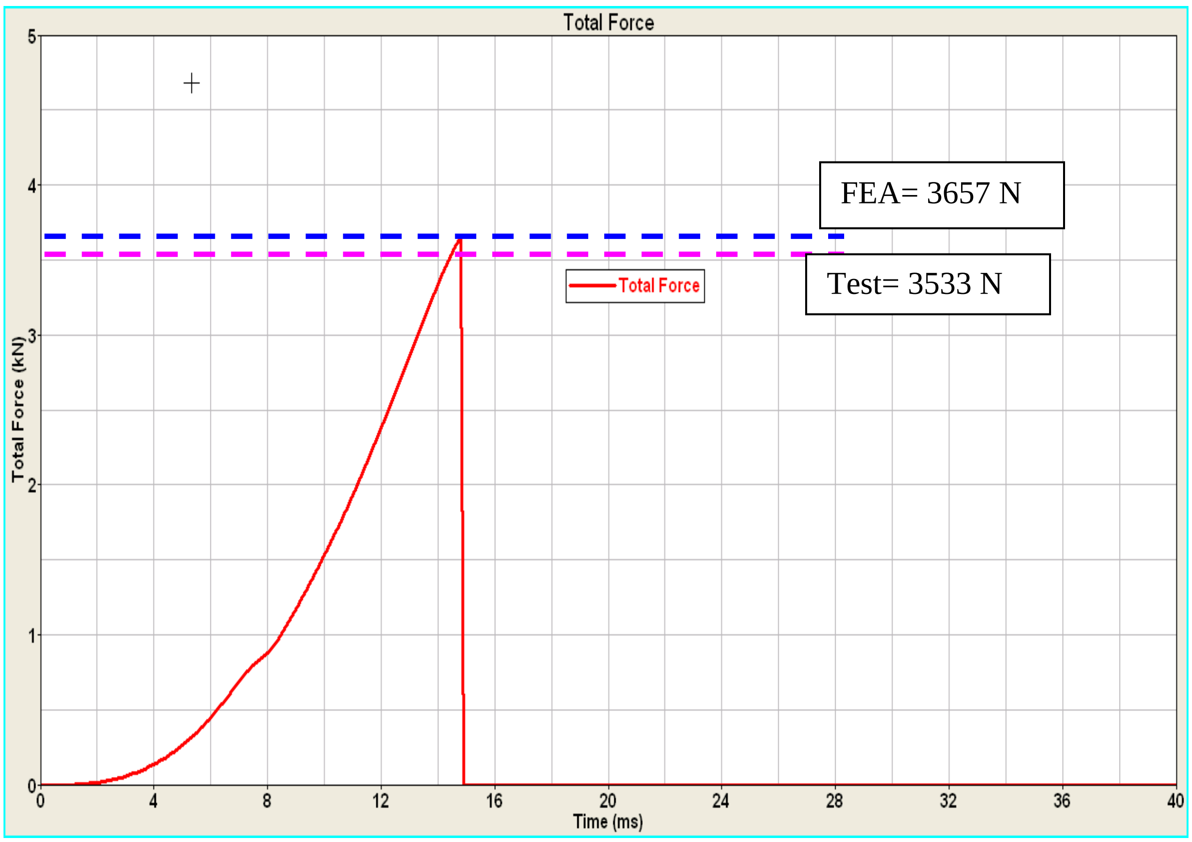


Figure 10 Comparison of Force from Test and CAE

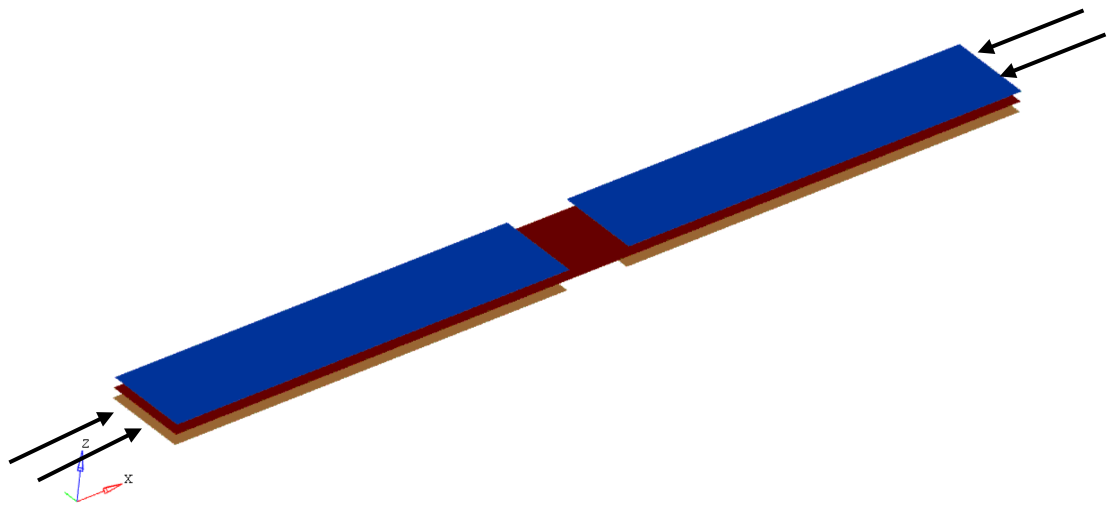


Figure 11 CFRT Laminate with (0-90-0)sym layup, compression along 0^0 Orientation

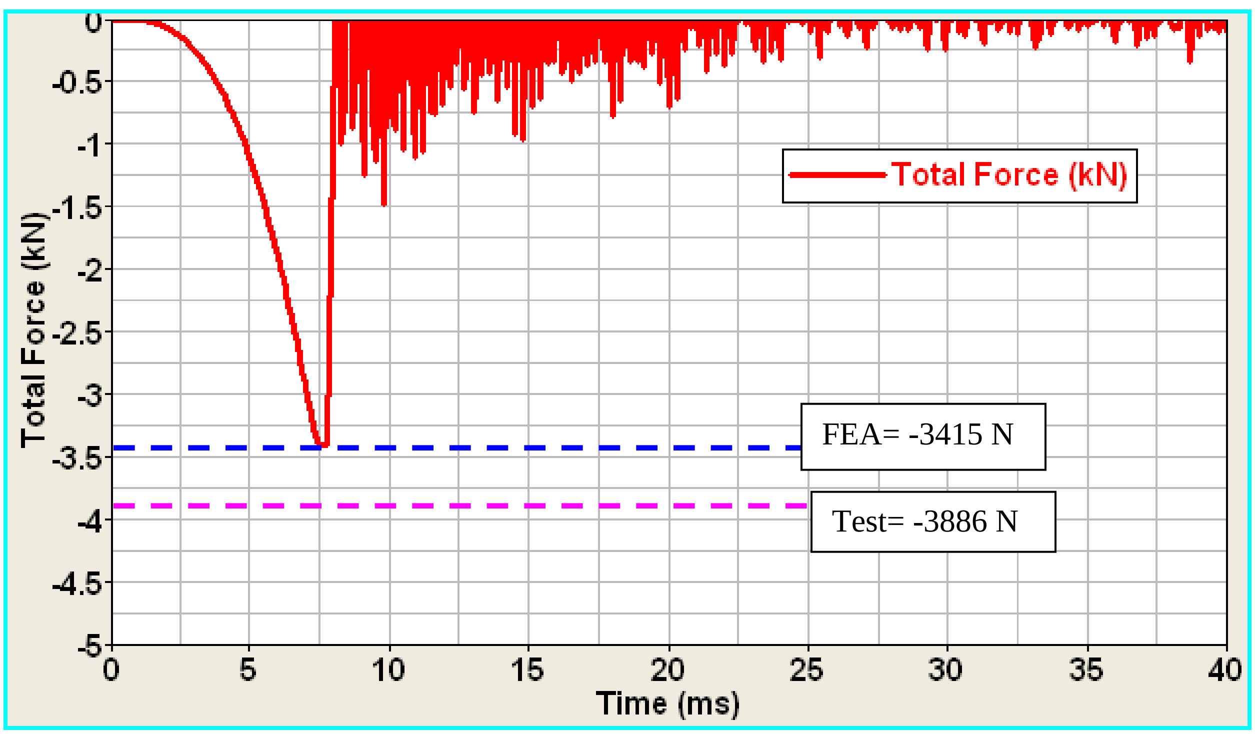


Figure 12 Comparison of Force from Test and CAE

Over Molded SFRT on CFRT Inserts

The next step was to look at modeling and verifying over molded SFRT (Glass Filled Polyamide 6) on CFRT inserts. The CFRT inserts are manufactured by thermoforming. The lay ups are pre-defined in the kitting process before being thermoformed. The CFRT inserts are held inside the injection molding tool before being over molded by SFRT. The FEA model involves modeling the mid plane surface of the CFRT and SFRT as shell elements. The materials models for SFRT and CFRT are defined as described in the previous sections. The coupling between the two is accomplished by tied contact definition in LS-DYNA. A number of parts were manufactured and tested and correlated with a FEA model. The first example, as shown in Figure 13, was over molding a tensile sample and conducting a tensile test. Figure 14 shows the total force from the FEA and its comparison with the test value which matches very well.

Another example, as shown in Figures 15-18, is a side member of a seatback. The black portion is the CFRT and the ribbing pattern (blue color) is over molded. In one of the tests, the part is mounted on a fixture and pushed by a rigid plate until it fails. There is a bending load on the part similar to what it will encounter in a seatback. The part fails due to buckling because of the high compression in the front wall (see Figure 18). In Figure 16, in the FEA a similar failure is observed near the bolts i.e., buckling failure due to compression. The force and displacement of the plate are monitored and are depicted in Figure 17. The figure also shows the force displacement curve from the CAE simulation for dry and conditioned state of the material. The test data matches closely with conditioned data in both the displacement and the peak load predicted. Another test was conducted on the same part. In this case, the structural member was pushed at an angle of 45^0. There is a combination of twisting and bending load in this case. This is again to replicate twisting load on a seatback member. The failed parts from the test and images of the FEA solution are shown in Figures 19-20. The location of the initial crack and the failure matches quite well. Table II gives the values of the maximum displacement and force from the test and FEA and they are in agreement.

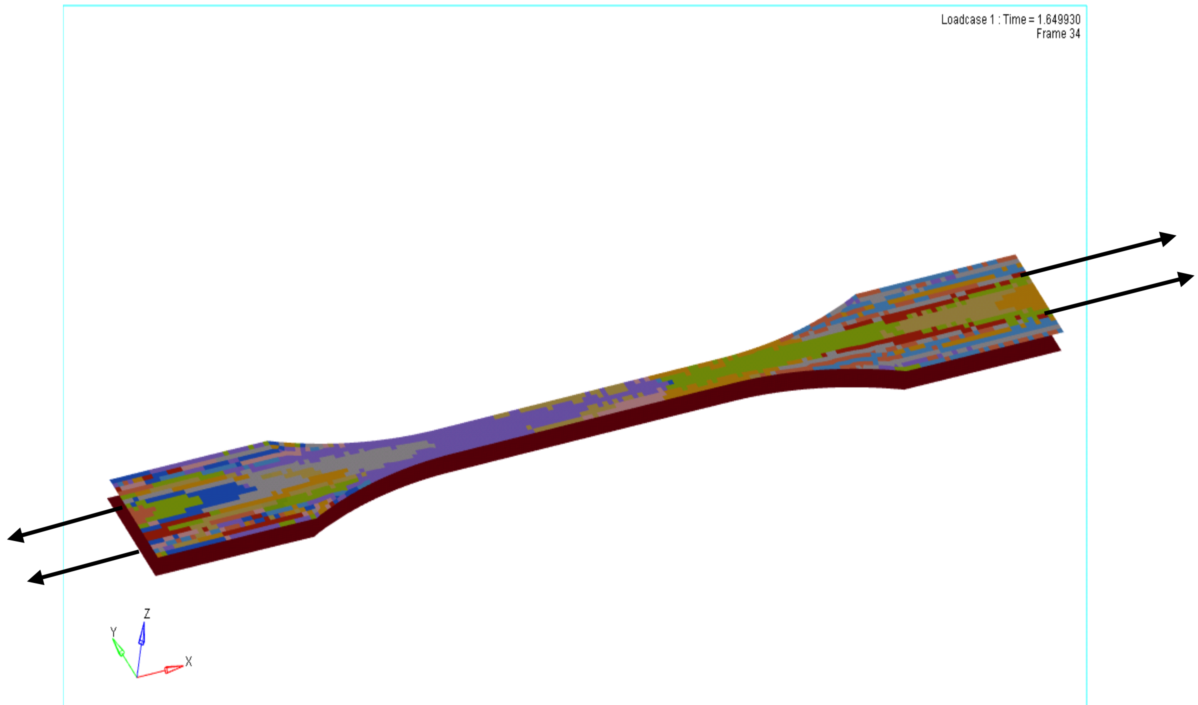


Figure 13 Over Molded SFRT (PA 6) over CFRT Laminate (0-90-0)sym, tensile test along 0^0

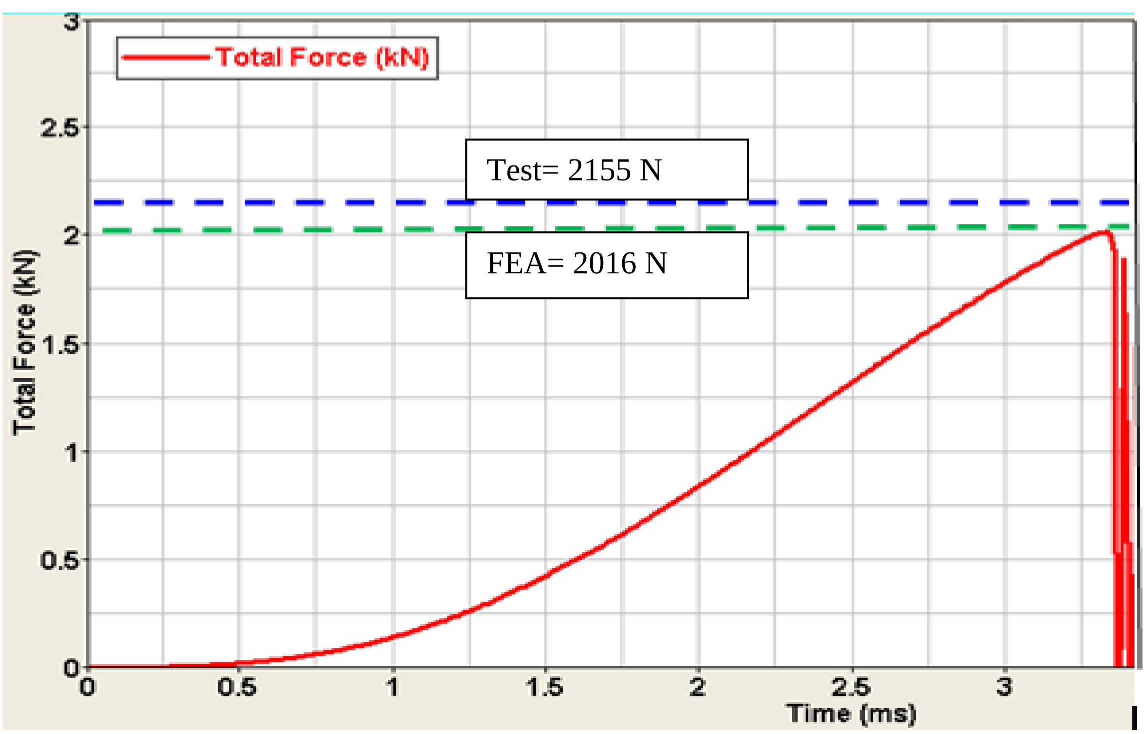


Figure 14 Comparison of Force from Test and CAE

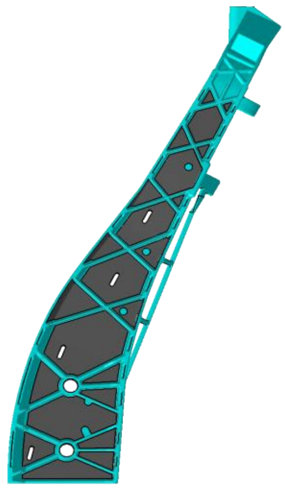


Figure 15 Over Molded SFRT (PA 6) over CFRT Insert

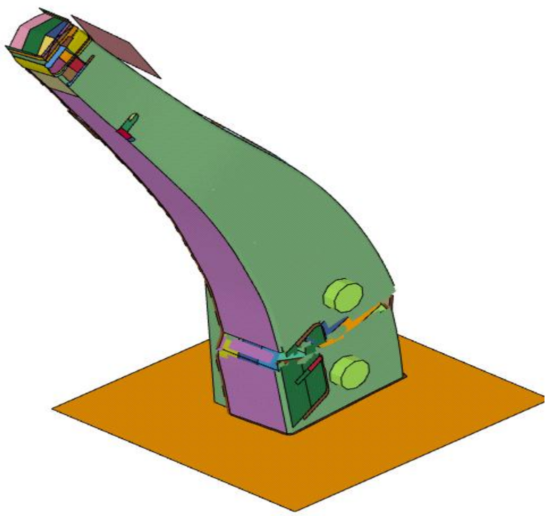


Figure 16 Failure after Push Test (FEA)

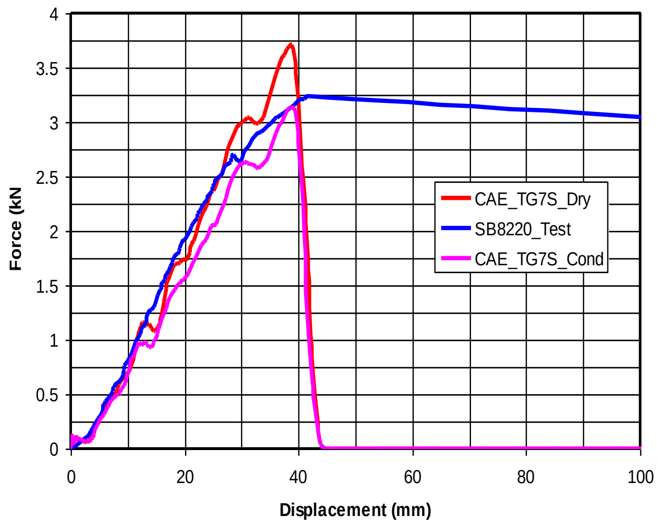


Figure 17 Force-Displacement Curve from Test & FEA

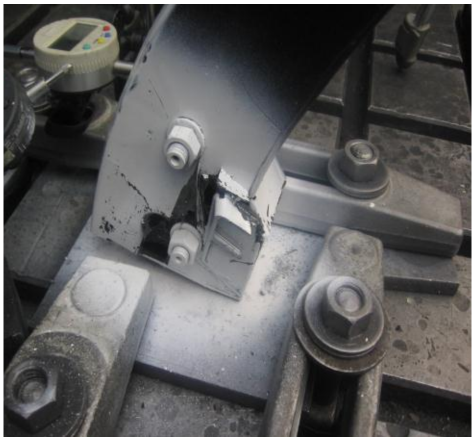


Figure 18 Failure due to Compression

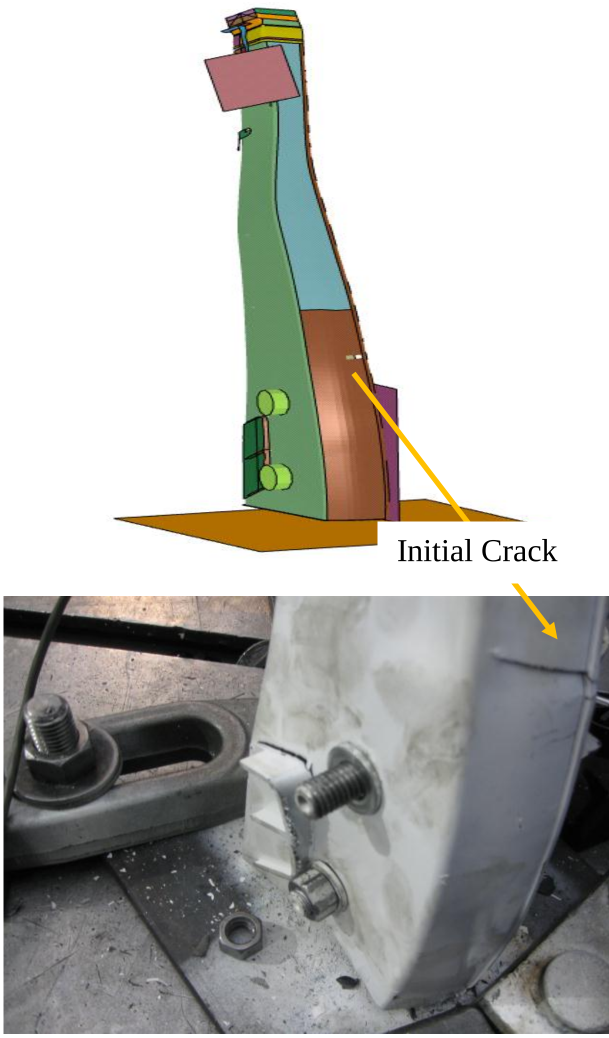


Figure 19 Crack Initiation from Test & FEA

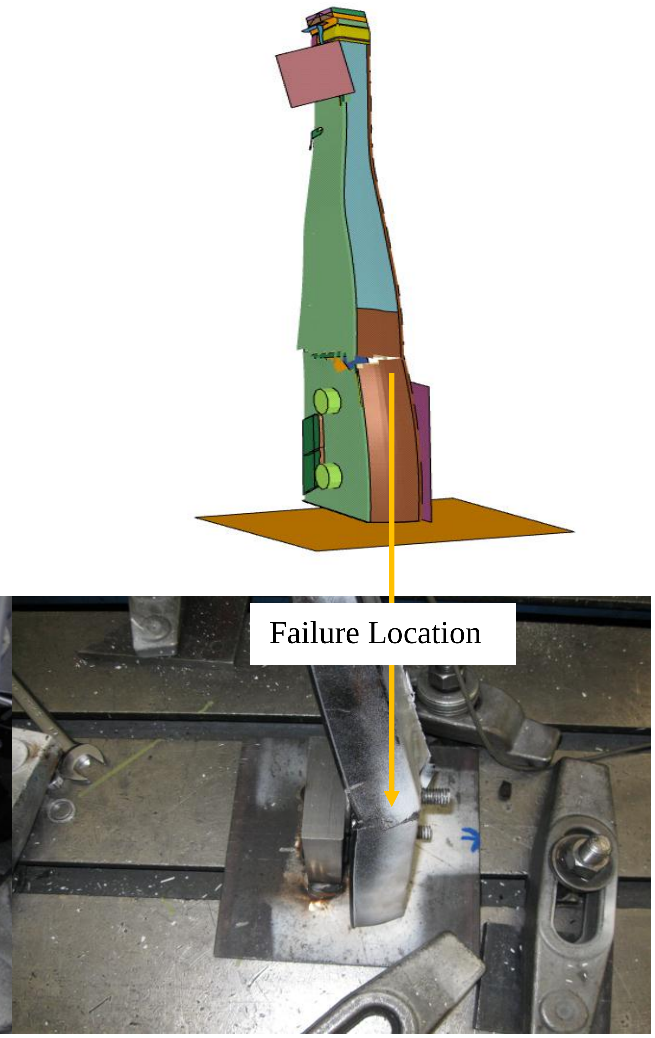


Figure 20 Failure Location from Test & FEA

Table II   Comparison of FEA and Test Data

	Max. Disp.	Max. Load
Test	58 mm	2.8 kN
FEA	55 mm	2.65 kN

Conclusions

A methodology is described to model in LS-DYNA parts made of over molded SFRT (Glass Filled Polyamide 6) on CFRT inserts. The CFRT laminate which uses an unfilled Polyamide 6 as the matrix and glass as the continuous fibers is modeled as MAT_58 (*MAT_LAMINATED_COMPOSITE_FABRIC). A number of standardized tests were carried out to determine the various parameters in the cards. The over molded SFRT is modeled by a User Defined Material Law which uses an anisotropic material model based on fiber orientation obtained from a moldflow software. This is based on ULTRASIM® methodology developed internally at BASF. A number of examples for both materials and combined parts are shown to verify its validity. Its application to seating back frames is subject of another paper.

References

[1] Stefan Glaser, Andreas Wuest, Bernhard Aumer, Integrative Simulation, Composite Materials, Kunststoffe International, 7/2008, p 60-63

[2] Stefan Glaser, Internal Document, BASF

[3] Hallquist, J., et al. LS-DYNA Keyword User's Manual, Version 971, 2007

[4] Schweizerhof, K., Weimar, K., Munz, Th., Rottner, Th., Crashworthiness Analysis with Enhanced Composite Material Models in LS-DYNA – Merits and Limits, LS-DYNA World Conference, 1998, Detroit, MI, USA

[5] Matzenmiller, A., Lubliner, J., Taylor, R.L., A Constitutive Model for Anisotropic Damage in Fiber-Composites, Mechanics of Materials 20 (1995)

[6] Carney, K., Melis, M., Fasanella, E.L., Lyle, K.H., Gabrys, J., Material Modeling of Space Shuttle Leading Edge and External Tank Materials For Use in the Columbia Accident Investigation, 8th International LS-DYNA Users Conference, 2004