Chapter 4
Simulation of Three-point Bending Sandwich Composite Panels Through Finite Element Analysis

Zulzamri Salleh and Goh Wei Kee

Abstract In marine industry, the usage of sandwich composite structures has been increased due to these structure can provide strong properties that the manufacturer desires. The useful properties of these structure are not only useful in marine industry but also in other industries such as aviation, automotive and aerospace industries. The main propose of this study is to provide useful simulation testing results for the future references by comparing the results of three-point bending tests through experimental and finite element analyses of sandwich composite structures. This research is conducted with fiberglass and a core material which is carbon from pine wood with different weight percentage (10, 20,30, 40 and 50 wt%). It was found that the sandwich panel with higher content of pine wood carbon 50 wt% has achieved the maximum stress 25.268 kPa and strain of 3.184×10^{-7} mm/mm. When the percentage of the pine wood-activated carbon increased or the core's diameter increased, all the values of maximum stress, maximum strain and maximum displacement also have been increased. This has also shown that specimens with higher percentage of the pine wood-activated carbon or the core's diameter can withstand more stress before the specimen rupture.

Keywords Three-point bending · Pine wood · Stress–strain

4.1 Introduction

According to American Society of Testing and Materials (ASTM Composite Standard), the sandwich composite structure concept is explained as follows [1]:

> A structural sandwich is a special form of composite comprising of a combination of different materials that are bonded to each other so as to utilize the properties of each separate component to the structural advantage of the whole assembly.

Z. Salleh (✉) · G. W. Kee
Universiti Kuala Lumpur Malaysian Institute of Marine Engineering Technology, Lumut, Perak, Malaysia
e-mail: zulzamri@unikl.edu.my

Fig. 4.1 Sandwich composite structure [1]

A sandwich composite structure usually consists of two thin layers known as the facing sheets and a core between the layers, so that the structure can have a maximum mechanical properties (see Fig. 4.1). In order to complete a sandwich structure, the core must be glued to the two facing sheets and forming two additional glued sides. To form a good quality of sandwich composite structure, the facing sheets must be able to withstand all the applied loads and bending moment and the core must provide the shear strength needed to the structure. So, thin facing sheets are enough to obtain great stiffness, and a lightweight core which is able to withstand the shear forces and able to make the two facing sheets that attached to it being stable.

By identifying the type of facing sheets and core used, the types of sandwich composite structures can be classified. For the type of core used, sandwich composite structures can be classified into three types, that is, the honeycomb core sandwich composite, foam-type core sandwich composite and corrugated core sandwiches. The facing sheets can be divided into two types, metallic and non-metallic sheets. The metallic facing sheets are made of, for example, aluminum, stainless steel and titanium. Non-metallic facing sheets usually use fiber-reinforced plastics (FRP) with different resins like epoxy and polyester. The types of core and facing sheets chosen to use will depend on the application of the sandwich composite structure [1].

Nowadays, the demand of sandwich composite structures in the industry becomes higher (see Fig. 4.2). This requires the quality of the sandwich composite structure also being high enough. Thus, the core materials must have low density, sufficient thermal insulation, sufficient thickness, sufficient shear stiffness, sufficient stiffness to prevent local buckling and sufficient shear strength to prevent core shear failure. The facing materials must have high tensile and compressive strength, good impact resistance, good water resistance, good environmental resistance, good surface finish and high stiffness and high flexural rigidity [1].

4.2 Methodology

The three-point bending test is a test that is performed on the materials or products to obtain their ultimate flexural load, which is the maximum transverse force that the materials or products can tolerate without permanent damage. This test is commonly carried out by a universal testing machine (UTM) with three-point or four-point bend fixture. The three-point bending test is the one of the most common procedures for products testing (see Fig. 4.3). The main objective to perform this bending testing is to gain the following data [3]

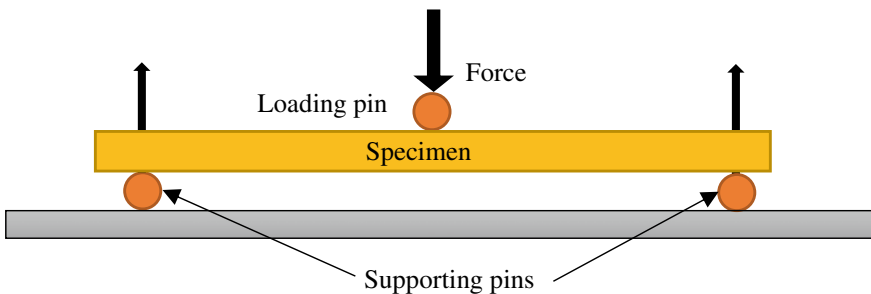**Fig. 4.2** Sandwich panels market—growth, trends and forecast (2019–2024) [2]

Fig. 4.3 Basic set-up for three-point bending test

(a) Flexural strength: Maximum load a material can withstand before its failure.
(b) Flexural modulus: Degree of slope of stress or strain curve. This is to indicate the material's stiffness.
(c) Yield point: The point where a material no longer displays elastic behavior. The material has become plastic, and from this point ahead, the further increase of loading will cause permanent defect on the material.

As stated by Hansdah, Tata Steel (2012), in engineering mechanics, materials that are used for flexural tests are [4]:

(a) Composite sandwich materials (e.g., carbon fiber for car's spoiler).
(b) Woods (e.g., walnut for propeller).
(c) Metals (e.g., aluminum 7075 in aircraft's wing).
(d) Brittle materials (e.g., ceramic for construction).

A specimen with rectangular shape and flat cross-section is placed on two parallel supporting pins in this test. A mean loading pin is applied in the middle of the loading force (see Fig. 4.3).

Values for modulus of elasticity in bending, flexural stress, flexural strain and the flexural stress–strain response of the material can be gained by carrying out the three-point bending test. The advantages of performing this test is to ease the specimen preparation and testing. The disadvantage of this test is that the result gained is very sensitive to the specimen and loading geometry and strain rate.

According to ASTM Designation C393-00, the flexural test of sandwich composite panel can be conducted. The test must specify the following factors [5]:

(a)   The testing machine and its accuracy
(b)   The specimen's shape and dimension
(c)   Analyzing result's method.

The flexural test can be carried out by two methods either as the three-point or as the four-point bending method. The four-point bending test is due to there is a lack of destructive influence of the pressure stamp on the top part of the specimen tested. Because of this disadvantage of the four-point bending test, the three-point bending test became most commonly used method. Specimen that is subjected to the four-point bending testing shows an increase of the flexural strength of the sandwich composite structure. The failure of the sandwich composite mainly occurs in the core of the material in flexural tests. There will occur a sliding crack in the core caused by shearing forces. When the ultimate shear strength is increased proportionally to the density of the used core, the level of porosity will be decreased.

Simulation is a process that uses computer software to simulate an actual situation by using experimental data. By using simulation, the structure's maximum allowable bending stress can be determined and the possible amount of force required to make the structure fail can be predicted. The sandwich composite structure was designed using a design software that is SpaceClaim with the specification stated in the American Society of Testing and Material Designation C393-00. Then, testing was performed testing by using finite element analysis simulation computer software, that is, CREO simulation. Comparison of results obtained from simulation computer software was made.

The five models were designed by using the SpaceClaim software by following the dimensions that are stated in ASTM C-393(00) (ASTM 2000) (0) which is 10 wt% activated carbon (core: 4.0 mm), 20 wt% activated carbon (core: 4.5 mm), 30 wt% activated carbon (core: 5.0 mm), 40 wt% activated carbon (core: 5.5 mm) and 50 wt% activated carbon (core: 6.0 mm). After completing the design, the design was imported to the CREO software to run the simulation testing. Before running the simulation testing, there are a few steps that need to set. The first step is setting the load and constrains (see Fig. 4.4). The second step is setting for the specimen the material which is CF-FG and soft pine wood (same properties with mangrove wood-activated carbon) (see Fig. 4.5). The third step is to create a refined mesh of all the surface of the specimen (see Fig. 4.6). The fourth step is running the AutoGem
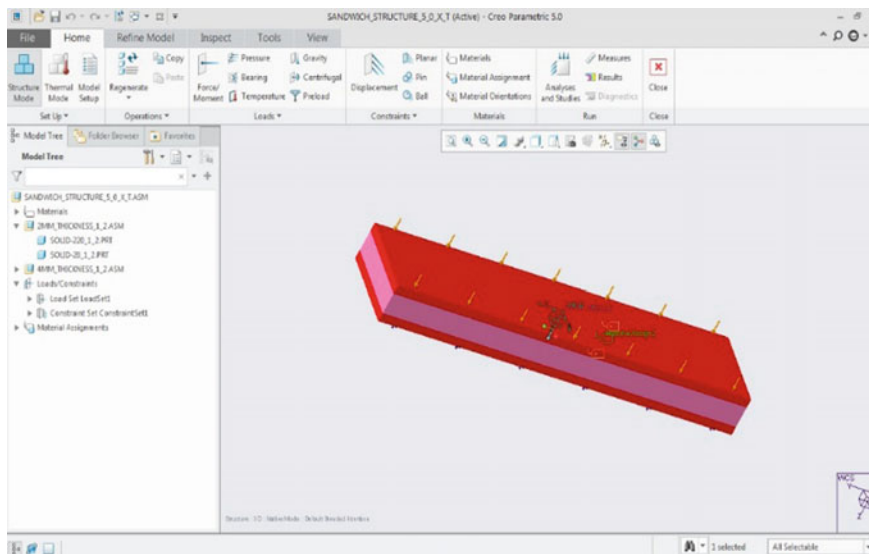
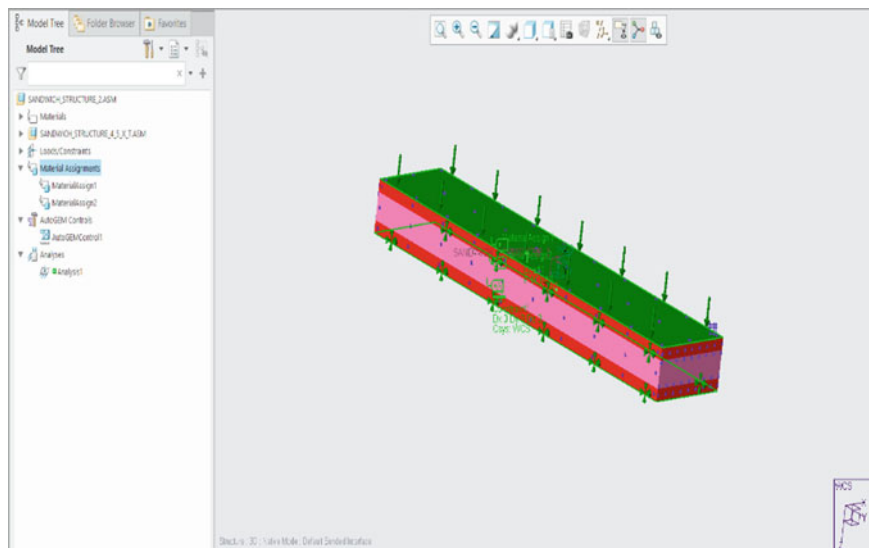**Fig. 4.4** Setting the load and constrains
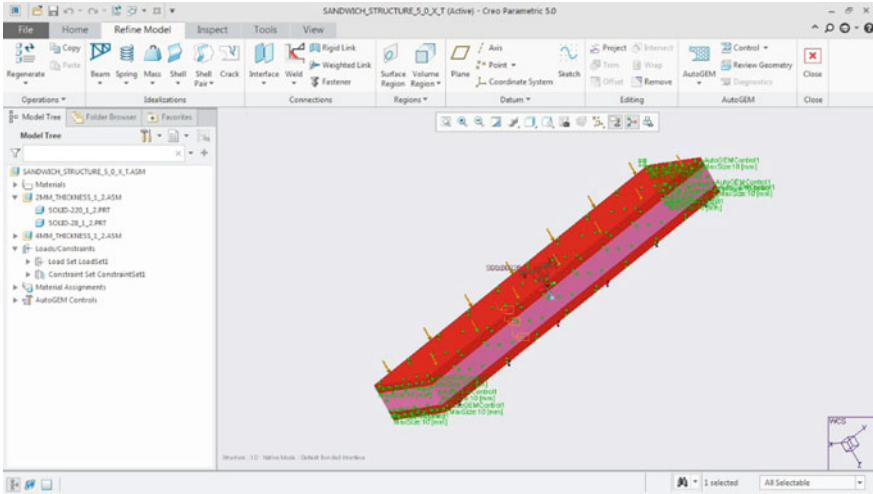

Fig. 4.5 Assigning material properties

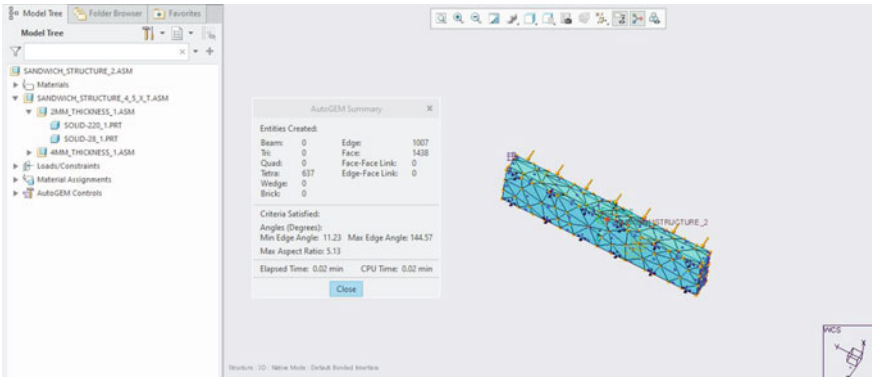Fig. 4.6 Refined mesh of all the surfaces

Fig. 4.7 Running the AutoGem

to know whether the specimen can be simulated or not (see Fig. 4.7). The last step is running the simulation (see Fig. 4.8).

4.3   Results and Discussion

In the CREO simulation software, there are three types of data or results that can be obtained which are stress, strain and displacement. All the data or results for each specimen was collected and tabulated.
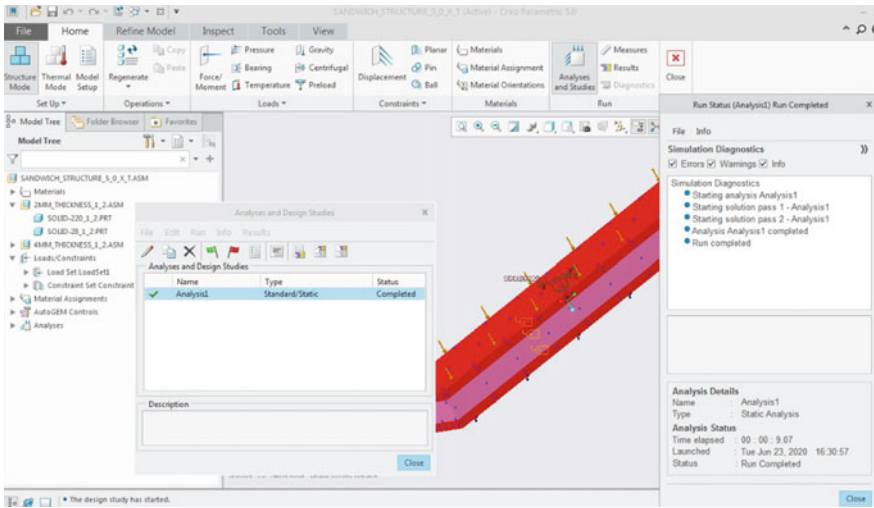
Fig. 4.8 Running simulation

Specimen 1 = core with 10 wt% activated carbon (core: 4.0 mm).
Specimen 2 = core with 20 wt% activated carbon (core: 4.5 mm).
Specimen 3 = core with 30 wt% activated carbon (core: 5.0 mm).
Specimen 4 = core with 40 wt% activated carbon (core: 5.5 mm).
Specimen 5 = core with 50 wt% activated carbon (core: 6.0 mm).

It can be concluded from Figs. 4.9 and 4.10 that there are differences in the values of stress, strain and displacement between all the five specimens. When the percentage of the mangrove wood-activated carbon increases or the core's diameter increases, all the value of maximum stress, maximum strain and maximum displacement
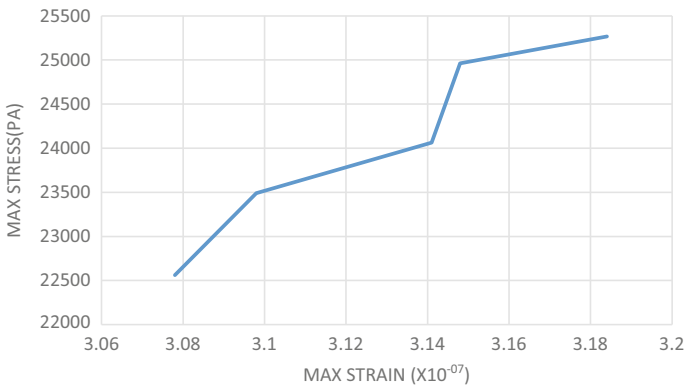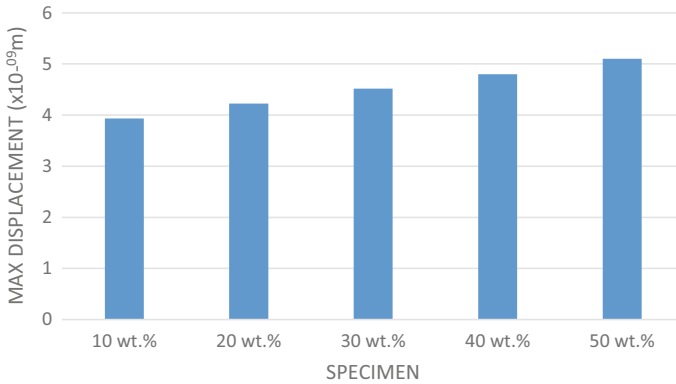

Fig. 4.9 Stress–strain diagram

Fig. 4.10 Displacement diagram

also increase. This shows that a specimen with higher percentage of the mangrove wood-activated carbon or the core's diameter, the more stress that the specimen can withstand before the specimen's rupture. Other than that, it can be concluded that by increasing the mangrove wood-activated carbon or the core's diameter, the mechanical properties of the specimen also increase. Since the mechanical properties increase, the specimen with higher mangrove wood-activated carbon or the core's diameter is more suitable for marine industry application. Based on the data gained from the simulation, both with the highest carbon content (6 mm) recorded the highest stress and strain. For the double layer with 50 wt% carbon content, the maximum stress is 25.27 kPa, while for the sample with 10 wt%, the minimum strength is 22.56 kPa. The details of the results are shown at Table 4.1.

The strain the sample with 50 wt% carbon content also recorded the highest which is 3.184×10^{-7}, while for the double layer, it is 3.078×10^{-7}. This data can be simply used for better understanding that the more layers and higher carbon content of the core, the highest maximum point of stress and strain can withstand before the specimen fails. This shows that the activated carbon content can influence the fiberglass-reinforced plastic with increased their contents. The details of the sample simulation results can be seen in Fig. 4.11.

Table 4.1 Details of stress and strain results

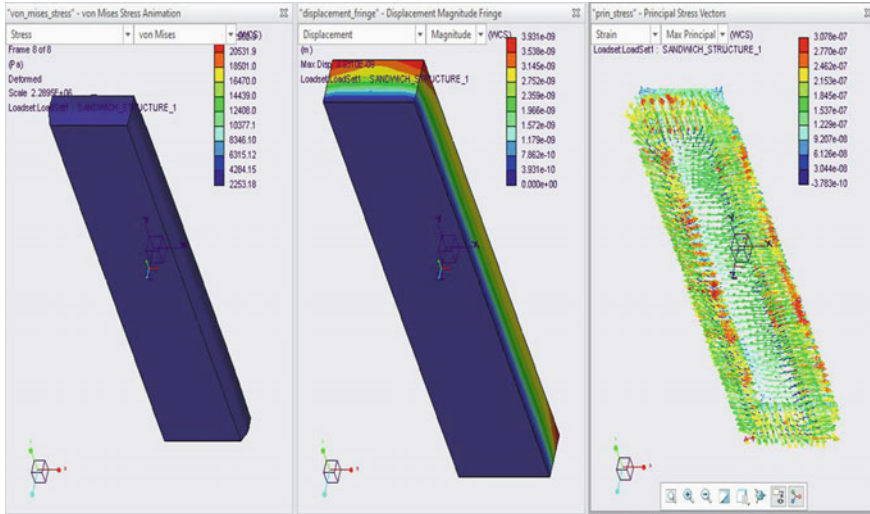| Sample weight percentage (wt%)	Stress (kPa)	Strain (mm/mm)	Max. displacement (m)
10	22.56	3.078×10^{-7}	3.913×10^{-9}
20	23.49	3.098×10^{-7}	4.225×10^{-9}
30	24.06	3.141×10^{-7}	4.515×10^{-9}
40	24.96	3.148×10^{-7}	4.799×10^{-9}
50	25.27	3.184×10^{-7}	5.102×10^{-9}

Fig. 4.11 Sample 1 simulation flexural testing result

4.4 Conclusion

In conclusion, this research was carried out to explore the relationship between stress and strain by using finite element analysis. The research was done by FEA method using the PTC CREO software. From the data gained, it will ease the production team to study the properties of the material. Overall, the objectives are achieved while conducting this project. Firstly, the design of the sandwich structure followed the ASTM C393-00 for double layer and triple layer. Second, the analysis of data gained from FEA allows to decide the best mechanical properties.

The mechanical behavior of the sample shows to be enhanced with increasing weight percentage of activated carbon powder. In the experimental study, the specimen with 50 wt% (10 mm) was able to withstand about 25.2682 kPa of stress, while for the sample with 10 wt%, the limit of the stress was 22.56 kPa.

For the displacement, the sample with 50 wt% recorded 5.102×10^{-9} mm the highest, while for sample with 10 wt% showed a displacement of 3.913×10^{-9} mm. However, it still experiences in the limitation and deficiencies such as low of impact strength, low in tensile strength, brittleness and low conductivity and others. The stress experienced on the specimen is different based on the percentage of carbon content in the specimens.

References

1. Nallagula S (2006) Behavior and flexure analysis of Balsa wood core sandwich composites: experimental, analytical and finite element approaches. Eng Struct 31(9):2032–2044
2. Wong YC, Mahyuddin N, Aminuddin AMR (2020) Development of thermal insulation sandwich panels containing end-of-life vehicle (ELV) headlamp and seat waste. Waste Manage 118:402–415
3. Lokesh KS (2018) Synthesis and study on effect of thickness on 3-point bending strength of sandwich composites. Int Res J Eng Tech 5(08):2395–2456
4. Sadhinoch M, Kampczyk M, Mulder R (2019) Statistics of fracture in 3-point bending. Mater Sci Eng C 651(1):012075
5. Freitas DST, Kolstein H, Bijlaard F (2011) Sandwich system for renovation of orthotropic steel bridge decks. J Sandw Struct Mater 13(3):279–301