Design of a CFRP composite monocoque: simulation approach

Tatthep Kankan트*, Ekkarin Phongphinnittana and Baramee Patamapromh

Department of Mechanical and Aerospace Engineering

*Corresponding Author: tatthep.k@gmail.com, Telephone Number 0-2555-2000 ext. 8315, Fax Number 0-2586-9541

Abstract. This article deals with the design of CFRP composite monocoque chassis for Formula Student race car. The design objective is to maximize a specific torsional stiffness of the monocoque and also satisfy the safety requirement assigned by Formula SAE. The sandwich structure has been used regarding to its high flexural rigidity per weight. The thickness and stacking sequences of composite plies have been optimized for each particular zone of the monocoque chassis using the FEM simulation.

1. Introduction

Formula SAE is a design competition for engineering students to have an opportunity to apply their knowledge to design a downscale formula race car, so called a formula student. Among various components, chassis is the main one that affects the behavior of a race car and provides a safety for drivers. The chassis needs to have a sufficient stiffness and satisfy the safety standard qualified by Formula SAE. Currently, most of chassis in this competition are a tubular steel space frame which has limitations of weight. The replacement of space frame by a lightweight structure will improve a performance of a race car [1]. A monocoque made from carbon fiber-reinforced polymer (CFRP) composite seems to be a good candidate thanks to its very high strength-to-weight ratio. Although the use of CFRP composite structure for monocoque chassis in race car or high-performance car is very common, the detail on thickness and stacking of composite is always confidential. Moreover, this material is very costly and highly depends on its fabrication process. Thus, the simulation-based design is preferable to optimize the material parameters. A hybrid monocoque has been chosen in this study to economize the budget of the production process. This type of monocoque refers to a composite cockpit with conventional steel tube sub-frame (Figure 1). One of the key properties of racing cars is a torsional stiffness of chassis. High torsional stiffness will result in a good handling performance especially in cornering [2]. This property has been studied for different tube cross-section of space frame [3] and depends on a class of vehicle as shown in Table 1. In order to provide a high torsional stiffness, the composite sandwich structure has been used regarding to its high flexural rigidity per weight [4]. This structure consists of composite skin and core materials. The foam core material has been chosen since it has a good energy absorption and relatively low price comparing to a high-performance honeycomb core. This study aims to optimize the thickness and stacking sequences of composite plies for each particular zone of the monocoque chassis. The FEM simulation has been used in cooperation with analytical method in order to maximize its specific torsional stiffness and simultaneously satisfy the safety requirement of the Formula SAE. The results from this study will be used in the future work for a crash simulation of this monocoque.
Table 1. different chassis torsional stiffness of vehicle class

| Vehicle    | Torsional stiffness (N·m/degree) |
|------------|----------------------------------|
| FSAE       | 1000 – 5000                      |
| Passenger  | 5000 – 20000                     |
| Sports     | 15000 – 40000                    |
| Formula One| 10000 – 100000                   |

2. Materials

In the study, the sandwich structure has been used thanks to its high flexural rigidity per weight. The increased distance between two rigid skins by lightweight core can notably improve the structural stiffness while slightly adds an additional weight to the structure. The sandwich structure consists of skin and core material as shown in Figure 2 and 3. The CFRP composite is designated for a skin material while the closed-cell rigid foam is used as a core material. The mechanical properties of these two materials will be implemented in the FEM simulation.

2.1 Composite

The CFRP composite for skin material is fabricate from a carbon fiber woven fabric with 3k, 2x2 twill weave pattern and thickness of 0.25mm as a reinforcement phase and epoxy resin as a matrix phase. The 3k means 3,000 filaments per tow and the number of fibers in warp and weft direction is equal (2x2) [2]. The hand lay-up process was used to fabricate specimens for mechanical properties characterization. The curing took about 3-4 hours at room temperatures. The mechanical properties were characterized from the previous study [5] and summarize in Table 2.

Table 2. Properties of Carbon composite

| Materials Properties | $E_1$ (GPa) | $E_2$ (GPa) | $E_3$ (GPa) | $v_{12}$ | $v_{23}$ | $v_{13}$ | $G_{12}$ (GPa) | $G_{23}$ (GPa) | $G_{31}$ (GPa) | $S_{11}$ (MPa) | $S_{22}$ (MPa) | $S_{33}$ (MPa) |
|----------------------|-------------|-------------|-------------|----------|----------|----------|----------------|----------------|----------------|---------------|---------------|---------------|
| Carbon               | 58.04       | 58.04       | 3.5         | 0.04     | 0.38     | 0.38     | 2.70           | 1.25           | 1.25           | 577.04        | 577.04        | 58.87         |
2.2. Foam

For a core material, PVC closed-cell rigid foam from DIAB has been chosen. The advantage of closed-cell rigid foam is energy absorption by its plastic deformation. It simultaneously enhances the crash performance of the monocoque chassis. The thickness of foam core is fixed to 20 mm. Its mechanical behavior and material properties have been characterized from the previous study and summarized in Table 3 and Figure 4.

**Table 3. Properties of PVC foam**

| Materials       | $E$ (MPa) | $\nu$ | $S_{\text{yield}}$ (MPa) |
|-----------------|-----------|-------|------------------------|
| Rigid Foam      | 34.15     | 0     | 0.88                   |

**Figure 4. PVC foam behavior under compression tests**

3. FEM Simulations

According to the FSAE Structural Equivalency Spreadsheet (SES) [6], there are the minimum requirements of mechanical properties including flexural rigidity (EI), area (A), yield tensile strength (YTS), ultimate tensile strength (UTS), max load at mid-span to give UTS for 1m long tube, max deflection at mid-span to give UTS for 1m long tube, and energy absorbed up to UTS, known as the baselines, for alternative material to be used in chassis design [5]. All properties baselines are equivalent to space frame steel tube properties for safety issues. Each material has different baseline requirements. For composite materials, the minimum bending or flexural modulus of the T3.30 Laminate test in SES is required.

The simulation approach is preceded in this study. Two stages of FEM simulations have been carried out using Abaqus/CAE. The simulation of three points bending test on sandwich structures with different design of composite stacking needs to be initially performed in order to ensure that each design is satisfied the flexural modulus baseline. Then, the potential designs will be used in the second simulation stage which is the composite monocoque simulation. The objective of this simulation is to investigate structural torsional stiffness of monocoque when using the given composite architectures. For Formula SAE, the monocoque structure can be divided into 7 sections as shown in the Figure 5 and the Table 4. Each section also requires different baseline.

**Table 4. Overview of monocoque section**

| Colour   | Section Name                           |
|----------|----------------------------------------|
| Blue     | Front Hoop Bracing (FHB)               |
| Red      | Front Bulkhead Support Structure (FBSS)|
| Orange   | Front Floor (FF)                       |
| Green    | Top Cockpit (TC)                       |
| Yellow   | Side Impact Structure Side (SISS)      |
| Black    | Side Impact Structure Floor (SISF)     |
| Brown    | Back Support (BS)                      |
3.1 Sandwich structure bending simulation

The numerical sandwich structure specimen was created according to the FSAE structure Equivalency Spreadsheet (SES), Laminate test result T3.30. The width of sandwich panel is recommended to 275 mm and the span length between supports is set to 400 mm. The foam core thickness is fixed to 20 mm. The sandwich panel contains 3 layers of composite on each side with the thickness of 1 mm per layer. The 3D model was created in Abaqus with C3D8R element type. The global mesh size was set to 3 mm which leads to 202,400 elements. All support pins and loading pin of bending test were defined as a rigid body. Only sandwich panel specimen was allowed to deform. The outer composite layers were assigned a different fiber orientation defined as SSP (Stacking Sequence Pattern) for each simulation. In this study, 4 SSPs have been investigated and summarized in Table 5. The two support pins were fixed and a 7500N force was applied at the loading pin as shown in Figure 6. The sandwich pattern design from this simulation stage was guaranteed to meet the SES sheet requirement.

| Code  | Stacking sequence                           |
|-------|---------------------------------------------|
| SSP1  | [0 / 0 / 0 / Core / 0 / 0 / 0]             |
| SSP2  | [0 / 45 / 0 / Core / 0 / -45 / 0]          |
| SSP3  | [0 / 30 / 0 / Core / 0 / -30 / 0]          |
| SSP4  | [0 / 60 / 0 / Core / 0 / -60 / 0]          |

3.2 Monocoque torsional stiffness simulation

The hybrid model was used to simulate the torsional stiffness of the monocoque chassis. This model uses shell elements for the composite structure part and 2-node linear beam section element for the steel tubes both of sub-frame and suspension components as shown in Figure 2. The composite sandwich properties including core thickness, skin layer thickness and stacking sequence of composite layers were assigned to shell elements while steel properties and cross-sectional geometries of space frame tubes were given to 2-node linear beam section elements in Abaqus. The boundary conditions were set as a typical torsional stiffness test; the rear A-beams were fixed at both sides and the force was applied to two font A-beams as shown in Figure 6. The displacement under loading was obtained from the simulation to calculate the angle of twist using eq.1. Finally, the torsional rigidity was determined by eq.2 using the angle of twist and the applied force.

\[
TR = \frac{FL}{1000 \cdot \theta} \quad (1)
\]

\[
\theta = \frac{2|d| \cdot 180}{l \cdot \pi} \quad (2)
\]

where

- \( TR \) = Torsional Rigidity (N·m/degree)
- \( d \) = displacement in vertical axis (mm)
- \( l \) = length between left A-beam to right A-beam (mm)
- \( F \) = testing load (N)
- \( \theta \) = angle twist

Figure 7. Boundary condition of torsional stiffness simulation
4. Simulation results

4.1 Sandwich structure bending result

![Simulation results](image)

**Figure 8.** 3-point bending contour of a vertical deflection (Y-axis)

The vertical deflection of 3-point bending simulation is shown in Figure 7 as the contour displacement in Y-axis. The bending of flexural modulus of the 4 SSPs is defined as a slope of applied force-displacement curves (Figure 4). The results show that all SSPs pass the minimum requirement from SES laminate testing at 190 N/mm.

![Simulation results](image)

**Figure 9.** Simulation results of 3-points bending test

In addition to the minimum flexural modulus of sandwich structure used in monocoque chassis (the first stage simulation), 4 out of 7 sections of monocoque which are greatly affect the torsional stiffness; Front Hoop Bracing (FHB), Front Bulkhead Support Structure (FBSS), Side Impact Structure Side (SISS) and Side Impact Structure Floor (SISF) have to pass their minimum requirements assigned by FSAE SES. Each particular zone has different baseline which relatively compare the sandwich structure to a number of steel tube as shown in Table 6. We noted that the SISS and SISF were considered together known as Side Impact Structure (SIS). The required mechanical properties; EI and YTS for each zone have been calculated and compare to its baselines in term of percentages (Table 4). The linearity limit at 6200 N in Figure 4 has been inputted to FSAE Structural Equivalency Spreadsheet for YTS calculation and compare to YTS baseline (steel). The percentages of EI and YTS for the three mandatory zones need to be over 100% so that they can be used in monocoque model to simulate the torsional stiffness.
4.2 Monocoque torsional stiffness result

The simulation result of vertical displacement for angle of twist calculation and local in-plan shear stress ($S_{12,local}$) for Model 1 is demonstrated in Figure 6. Table 5 is the summary of all models angle of twist and torsional stiffness along with their composite designs.

![Contour plot of Y-deflection and shear stress](image)

**Table 6.** The result of structure equivalent in percent

| Zone   | Number of tubes | EI       | Yield tensile strength |
|--------|-----------------|----------|------------------------|
| FHB    | 1               | 100.3%   | 161.1%                 |
| FBSS   | 3               | 101.3%   | 154.0%                 |
| SIS    | 3               | 115.8%   | NA                     |

**Table 7.** Summary of torsional stiffness in different patterns.

|        | FHB  | FBSS | FF   | TC   | SISS | SISF | BS   | Degree ($\theta$) | Torsional stiffness (N·m/degree) |
|--------|------|------|------|------|------|------|------|-----------------|---------------------------------|
| Model 1| SSP1 | SSP1 | SSP1 | SSP1 | SSP1 | SSP1 | SSP1 | 0.6505          | 5336.63                         |
| Model 2| SSP1 | SSP1 | SSP2 | SSP2 | SSP1 | SSP1 | SSP1 | 0.5710          | 6079.57                         |
| Model 3| SSP2 | SSP1 | SSP1 | SSP2 | SSP2 | SSP1 | SSP1 | 0.5204          | 6671.41                         |
| Model 4| SSP3 | SSP1 | SSP1 | SSP3 | SSP3 | SSP1 | SSP1 | 0.5538          | 6267.43                         |
| Model 5| SSP4 | SSP1 | SSP1 | SSP4 | SSP4 | SSP1 | SSP1 | 0.5596          | 6204.73                         |
| Model 6| SSP2 | SSP2 | SSP2 | SSP2 | SSP2 | SSP2 | SSP2 | 0.4900          | 7085.46                         |
5. Discussion

The results from the 3-points bending simulation show that the orientations of fiber which interprets by composite stacking do not affect the flexural modulus as long as they contain the 0° woven ply (1.2% max. difference). This can be explained by the nature of composite that preferable supports a given load in fiber direction so that the 0° woven ply will support almost all bending load. On the other hand, the fiber orientation has highly effect on the torsional stiffness as the results of monocoque simulation. The models 1-7 show the modification of torsional stiffness depending on their SSPs. The maximum torsional stiffness was obtained from Model 6 at 7085 Nm where the SSP2 \[0/45/0/Core/0/-45/0\] was applied to all monocoque section. This shows a good agreement with the theory of composite mentioning that the ±45° composite stacking sequence is preferable to withstand the torsional load. Comparing to the torsional stiffness of metallic space frame at 1900-2000 N·m, the composite monocoque is far better in this concern and will definitely provide a better handling for the race car. However, in the real-world application, the monocoque does not have only torsional load but also bending load or even impact load in case of crash. The use of Model 6 may not be optimized for all loading. Hence the criteria to design a composite monocoque should also have considered together the other structure properties, especially the energy absorbed capacity which relates directly to the safety issue of drivers.

6. Conclusion

This study is to design the composite monocoque using simulation approach to maximize its torsional stiffness. The conclusion can be made as follows:

- The flexural modulus from all proposed composite stacking patterns is relatively the same since all designs contain a 0° woven ply which is a dominated ply to support bending stress
- Fiber orientations which represent by composite stacking have an effect of torsional stiffness of monocoque structure
- FEM simulation simplifies the difficulty to optimize a composite stacking that satisfy the safety standard and maximize the torsional stiffness at the same time
- The future work will focus on the effect of core material thickness which also relate to the crash energy absorption of the monocoque

References

[1] Takahashi N, Kageyama Y, and Kawamura N 2011 Research of Multi-Axial Carbon Fiber Prepreg Material for Vehicle Body (No. 2011-01-0216). SAE Technical Paper.
[2] Sampo E 2010 Chassis torsional stiffness: analysis of the influence on vehicle dynamics. In SAE 2010World Congress&Exhibition.
[3] Liu CH, Li G, Ma YH and Yang XG 2018 Torsional Stiffness Comparison of Different Tube Cross-Sections of a Formula SAE Car Space Frame. In MATEC Web of Conferences 153 04002 EDP Sciences.
[4] Gay D and Hoa SV 2007 Composite materials: design and applications. CRC press. vol 4-chapter 4, 69-81.
[5] Patamaprohm B, et al, “Improving the impact resistance of CFRP composite structures”, The 32nd Conference of Mechanical Engineering Network of Thailand, 3-6 July 2018, Mukdahan, Thailand
[6] SAE International [Internet]. Pennsylvania: 2018 FSAE structural Equivalency Spreadsheet V1.3 Available from: https://www.fsaeonline.com/cdsweb/gen/DocumentResources.aspx