International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878 (Online), Volume-8 Issue-4, November 2019

Numerical Modelling of Glass Fiber Reinforced Polymer (GFRP) Cross Arm

A. Alhayek, A. Syamsir, V. Anggraini, Z. C. Muda, N. M. Nor

Abstract: ***Composites are not isotropic like their metal counterparts, e.g. steel and aluminum, as they are made of two distinctive phases known as the matrix and the reinforcing phases. In addition, weight, fiber direction, fiber composition and even the manufacturing process are all critical factors in determining the strength, stiffness and the behaviour of a composite member. All of that create more challenging designing and manufacturing approaches. This paper shows how to model a GFRP cross arm using SOLIDWORKS to create the 3D geometrical model because it has an intuitive and easy to use user interface, and ANSYS to create the numerical model and the analysis for its great and comprehensive capabilities in the finite element analysis. The cross arm was found to be safe against the failure modes of fiber, matrix, in-plane shear, out-of-plane shear and delamination under all load cases which satisfies the ultimate limit state requirements but the concern was on the serviceability limit state which had a deflection of 34 mm.***

Keywords: Numerical Modeling; Cross Arm; GFRP; Serviceability Limit; Failure Modes

I. INTRODUCTION

Glass fiber reinforced polymer (GFRP) composites are being increasingly used in construction fields due to their very high strength-to-weight ratio, high corrosion resistance, high strength, low thermal conductivity and electrical insulation characteristics. Although carbon fiber reinforced polymer (CFRP) composites provide significantly higher strength and stiffness than GFRP, they are not favored in such applications for their considerably higher cost and electrical conductivity properties. These composites are not isotropic like their metal counterparts, e.g. steel and aluminum, as they are made of two distinctive phases known as the matrix and the reinforcing phases. In addition, weight, fiber direction, fiber composition and even the manufacturing process are all critical factors in determining the strength, stiffness and the behaviour of a composite member. This creates more challenging designing and manufacturing approaches in order to come up with an economical design that can withstand all kinds of loads and their internal stresses [1-4].

The nature of fiber reinforced polymer (FRP) makes analysing them at different levels a necessity including fiber/resin, lamina, laminate and structure levels. Moreover, any geometrical modifications will easily lead to changes in material constituents, fiber volume fraction, lamina thickness, strength and even stiffness. Although many investigations on fiber reinforced polymers were conducted for aerospace structures, the outcomes are not readily applicable for civil engineering applications, such as pultruded composites, for several reasons. Those reasons start from the fact that aerospace applications use advanced manufacturing techniques and stricter quality control which enable them to produce superior composites to those produced by other more economical means for construction uses as Fig. 1 demonstrates [5-8].


(a) (b)

Fig. 1 The difference in quality and accuracy of stacking sequence of composite laminates [5] (a) High quality composites (b) Pultruded composites

II. METHODOLOGY

As Fig. 2 shows, this paper's numerical model went through two main stages starting by creating the 3D model in SOLIDWORKS program. Then the remaining steps were done in the finite element analysis software ANSYS, including establishing the numerical model and running the simulation. Whereas for the material properties, they were taken from Wagner's Composite Fibre Technologies (WCFT), Australia.

Manuscript published on November 30, 2019.
* Correspondence Author

A. Alhayek*, Institute of Energy Infrastructure, Universiti Tenaga Nasional, Jalan IKRAM-UNITEN, 43000 Kajang, Selangor, Malaysia.

A. Syamsir, Institute of Energy Infrastructure, Universiti Tenaga Nasional, Jalan IKRAM-UNITEN, 43000 Kajang, Selangor, Malaysia.

V. Anggraini, School of Engineering, Monash University Malaysia, Jalan Lagoon Selatan, Bandar Sunway, 47500 Subang Jaya, Selangor, Malaysia

Z. C. Muda, Institute of Energy Infrastructure, Universiti Tenaga Nasional, Jalan IKRAM-UNITEN, 43000 Kajang, Selangor, Malaysia.

N. M. Nor, Faculty of Engineering, Universiti Pertahanan Nasional Malaysia, 57000, Kuala Lumpur, Malaysia.

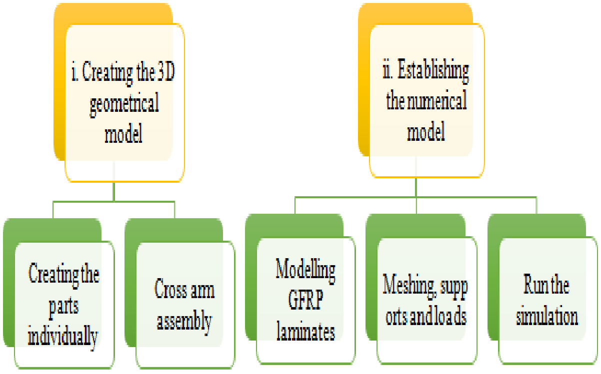

Fig. 2 Overview of the methodology

2.1 Creating The 3D Geometrical Model

The cross arm 3D model was generated in SOLIDWORKS program including all members and connections. The main and tie members have been modelled as surface objects, i.e. without a thickness, to allow for ply build up later in the finite element analysis software. The main members had a total length of 4832 mm with a square cross section having a width of 127 mm. As for the tie members, they had 4747 mm in length and a rectangle cross section having 102 mm and 76 mm in dimensions. As for connections parts, they were created as solid parts and were used to connect the members together and to provide support and loading regions. Finally, all parts were assembled in the same program as well to form a complete model. Figure 3 demonstrates the completed cross arm 3D model which are consisting of tie and main members.

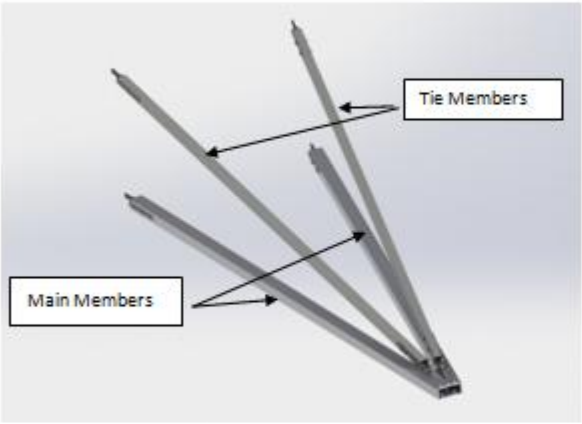

Fig. 3 Cross arm 3D model

2.2 The Finite Element Numerical Model

The completed 3D cross arm parts from SOLIDWORKS were transferred to ANSYS program to develop the three-dimensional finite element model in order to perform the required linear analysis. This includes defining the material properties, modelling the (GFRP) laminates with fibers' directions, meshing, supports and loads acting on the cross arm. The ACP (Pre) module was used to model the composite members and the Mechanical Model module to model the steel connections parts as it is shown in Fig. 4.

As mentioned previously, the materials were taken from Wagner's Composite Fibre Technologies (WCFT), Australia. A new glass unidirectional fibers material was created and all orthotropic properties were keyed in including orthotropic elasticity, orthotropic stress limits and orthotropic strain limits. Tables 1 and 2 illustrate all the GFRP material properties that were used in this simulation.

Table. 1 Vinylester material properties [9]

Property	Value	Unit
Density	1070	Kg/m3
Young's Modulus	4000	MPa
Poisson's Ratio	0.33	
Shear Modulus	1503.8	MPa
Tensile Ultimate Strength	90	MPa

After defining all materials properties, the 3D geometric model of the composite members was imported into the ACP (Pre) module, i.e. block A, in ANSYS Workbench. Then it was meshed with a mesh size of 20 mm which was investigated and determined to maintain a balance between computational time and results accuracy. Quadrilateral elements were used as much as possible and the aspect ratio was inspected and was found that almost all elements had a ratio of 1 or very close to it as illustrated in Fig. 5. After that, the plies were created and stacked up based on the configuration and fibers direction needed with the fabric thickness desired. The typical configuration was set as Muttashar et al. [12] mentioned, in which they used the same hollow sections from Wagner's Composite Fibre Technologies (WCFT), Australia, that were modelled in this paper. The stack up consisted of 9 plies of GFRP with a thickness of 0.7 mm per ply and fiber directions (0^{o} /+45^{o} /0^{o} /-45^{o} /0^{o} /-45^{o} /0^{o} /+45^{o} /0^{o}) in which 0^{o} direction represents the member's longitudinal direction.

Table. 2 GFRP orthotropic material properties [10,11]

Property		Notation in ANSYS	Value	Unit
Density		ρ	2050	Kg/m^{3}
Orthotropic Elasticity	Young's Modulus X direction	E1	36.3	GPa
	Young's Modulus Y direction	E2	10.8	GPa

	Young’s Modulus Z direction	E3	10.8	GPa
	Poisson’s Ratio XY	v12	0.28	
	Poisson’s Ratio YZ	v13	0.09	
	Poisson’s Ratio XZ	v23	0.28	
	Shear Modulus XY	G12	4	GPa
	Shear Modulus YZ	G13	3	GPa
	Shear Modulus XZ	G23	4	GPa
Orthotropic Stress Limits	Tensile X direction	Xt	596	MPa
	Tensile Y direction	Yt	55	MPa
	Tensile Z direction	Zt	55	MPa
	Compression X direction	Xc	550	MPa
	Compression Y direction	Yc	120	MPa
	Compression Z direction	Zc	120	MPa
	Shear XY	Sxy	86	MPa
	Shear YZ	Syz	44	MPa
	Shear XZ	Sxz	86	MPa

For the connection parts, they were imported into the Mechanical Model module, i.e. block D, and the default structural steel was chosen from the existed engineering data library in ANSYS. Whereas for the mesh, since these parts are responsible for the boundary conditions and carrying the loads, a finer mesh of 10 mm was generated.

In the next step, all models were transferred to Static Structural module, i.e. block B, as shown in Fig. 4 as solid elements to determine the boundary conditions, loads and to perform the necessary analysis. The contact surfaces between all parts were taken as “Bonded” in which no sliding nor separation between faces or edges is allowed and it allows for a linear solution. In order to create a more realistic pinned support, a cylindrical support was chosen for all end connections as illustrated in Fig. 6 in which the tangential component was left free to allow for rotation, whereas the axial and radial components were fixed to prevent any movement along all axis.

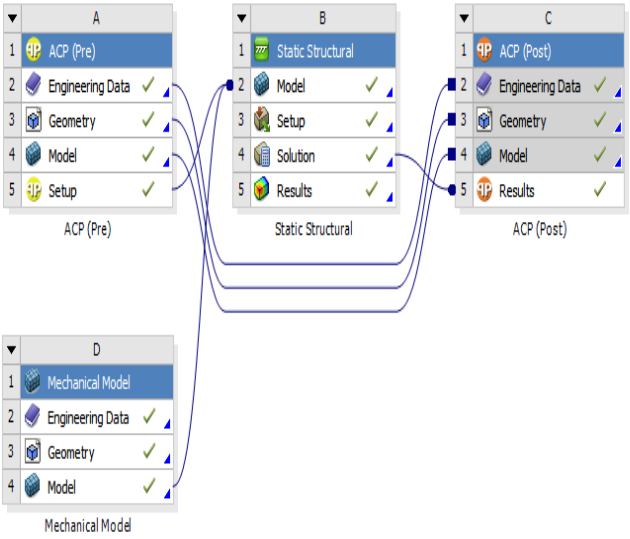

Fig. 4 Overview of the project schematic in ANSYS

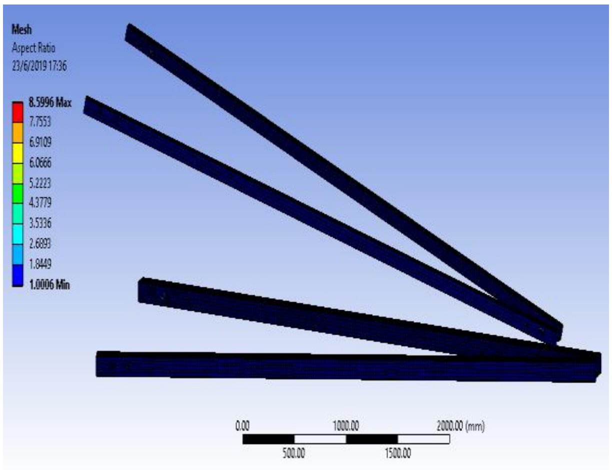

Fig. 5 The aspect ratio for the mesh elements

There are two load cases that were applied on the cross arm as shown in Table 3. The vertical component was simulated as a bearing load whereas the transverse and longitudinal loads were simulated as normal forces as shown in Fig.7. Moreover, the standard earth gravity acceleration 9.81 m/s2 was added to include the self-weight of the cross arm. All other settings were left as default or program controlled.

Table. 3 Load Cases

Load Case		Vertical (N)	Transverse (N)	Longitudinal (N)
Ultimate Limit State	1. Normal	42496	23436	0
	2. Broken wire	20545	10834	32224
Serviceability Limit State	1. Normal	21248	11718	0

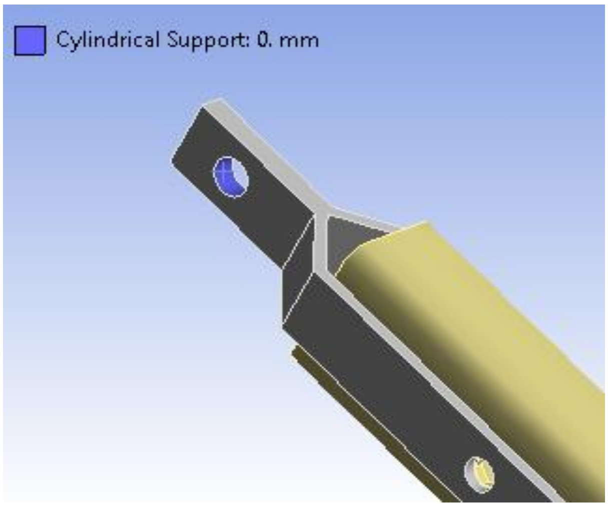

Fig. 6 The cylindrical support used

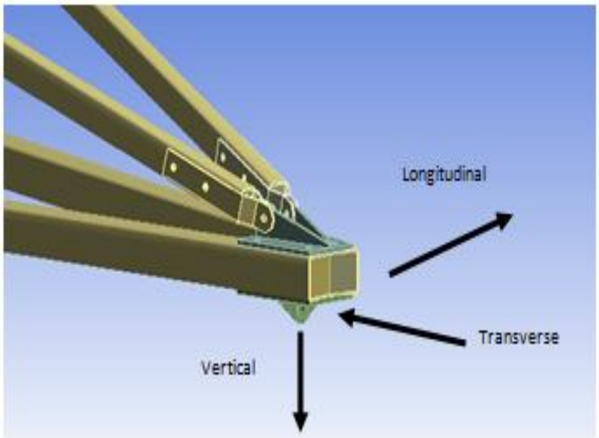

Fig. 7 Load components orientations

III. RESULTS AND DISSCUSION

3.1 Numerical Verification

The verification process was conducted on the findings of Muttashar et al. [12] paper since their experiments were based on GFRP cells from the same source, Wagner's Composite Fibre Technologies (WCFT), Australia. Therefore, the same material properties and plies fibers orientations were applicable. A single empty cell case (1-C-H-0) was chosen for the verification with the descriptions as shown in Table 4 and Fig. 8.

Table. 4 Description of the GFRP verified beam [12]

Specimen	L_t (mm)	L (mm)	a (mm)
1C-H-0	2000	1350	525

Figure 9 shows a single hollow member, similar to the members used in this project, was modelled and simulated in ANSYS. The experimental deformation was found to be 18.8 mm from Muttashar et al. research while the simulated one resulted in 18.75 mm with a 0.26% difference.

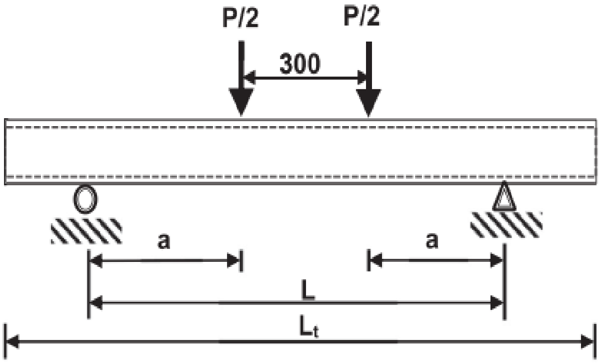

Fig. 8 Flexural test setup based on Muttashar et al. [12]

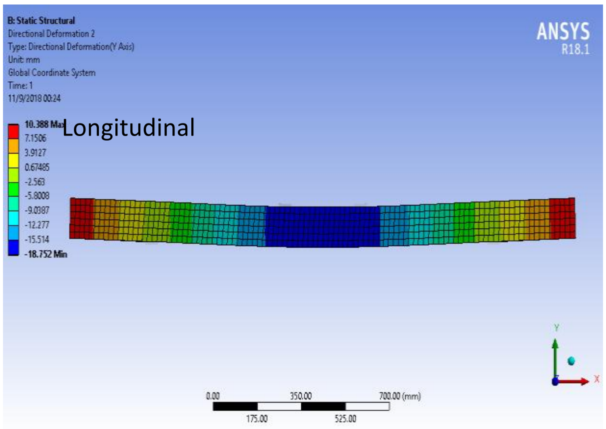

Fig. 9 Deformation result of the verified member

3.2 The Cross Arm Simulation

As demonstrated in Table 5, the ULS broken wire loading case is the most critical as it had a maximum equivalent von Mises stress of 105 MPa happening at the last (outer) ply in main members as demonstrated in Fig. 10. In addition, the ULS normal loading case had a total deflection of 63.74 mm which is more than the ULS broken wire load case by around 8% but the deflection modes were different having some lateral torsional buckling in the broken wire case as shown in Fig 11.

Table. 5 Maximum deflection and stress results

Load Case	Total Deflection (mm)	Max Stress (von Mises) (MPa)				
		All members	1st Ply Tie Members	Last Ply Tie Members	1st Ply Main Members	Last Ply Main Members
ULS Normal	63.7	90.0	87.4	87.5	90.0	90.0
ULS Broken Wire	58.6	105.0	89.4	96.9	98.6	105.0
SLS Normal Case	34.0	46.0	44.8	44.8	46.0	46.0

A Composite Failure Tool was used in ANSYS with the criteria being the maximum stress in order to calculate the factor of safety against the different failure modes. Table 6 shows that in the GFRP cross arm, the composite members were fairly safe under the ultimate limit state loads against multiple failure modes including fibers and matrix failures, in and out of plane shears failures, and delamination failure. Therefore, the issue was concluded to be a serviceability requirement rather than being of ultimate limit state with some notes about the delamination failure in the ULS broken wire case.

Table. 6 Failure safety factors

Load Case	Fibers s1	Matrix s2	In-Plane Shear s12	Out-of-Plane Shear s13	Out-of-Plane Shear s23	Delamination s3
ULS Normal	4	2.1	1.4	1.4	1.7	3
ULS Broken Wire	4	1.2	3.5	2	1.4	1

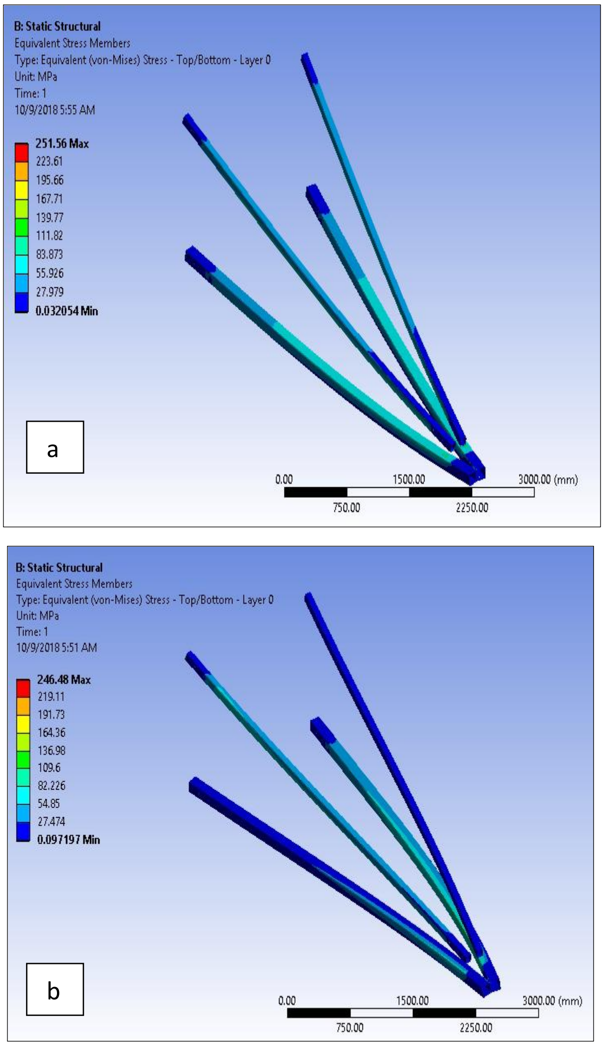

Fig. 10 Stress distribution for a) ULS normal load case. b) ULS broken wire load case

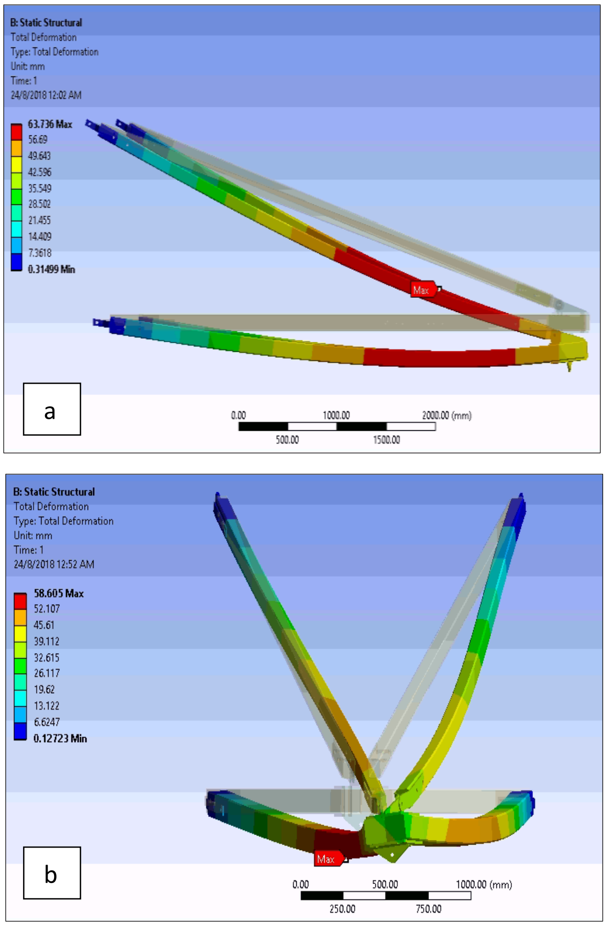

Fig. 11 Deflection for a) ULS normal loading case b) ULS broken wire case

Retrieval Number: D5162118419/2019©BEIESP
DOI:10.35940/ijrte.D5162.118419
Journal Website: www.ijrte.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

IV. CONCLUSION

With the increase of using composite materials in civil engineering applications, robust and accurate models and simulations are becoming more important. SOLIDWORKS provided a very easy and intuitive user interface that allowed the 3D model of the cross arm to be modelled simply and quickly. In addition, ANSYS had a very strong and complete package for modelling the composite materials starting from defining the material properties to choosing the fibers direction and lamina thickness. Also, ANSYS had a comprehensive module to check the results of the composite materials after the analysis was done. The cross arm was found to be safe against the failure modes of fiber, matrix, in-plane shear, out-of-plane shear and delamination under all load cases which satisfies the ultimate limit state requirements but the concern was on the serviceability limit state which had a deflection of 34 mm:

ACKNOWLEDGMENTS

The authors express their gratitude to Universiti Tenaga Nasional (UNITEN), Malaysia for supporting this research under UNIIG 2018, through Project No: J510050801 and BOLD 2025. Special thanks to those who have contributed to this project directly or indirectly.

REFERENCES

1. D. H. Deitz, I. E. Harik, M. Asce, H. Gesund, and F. Asce, “Physical Properties of Glass Fiber Reinforced Polymer Rebars in Compression,” J. Compos. Constr., vol. 7, no. November, pp. 363–366, 2003.
2. Y. Akematsu, K. Kageyama, and H. Murayama, “Basic Characteristics of Electrical Discharge on CFRP by Using Thermal Camera,” Procedia CIRP, vol. 42, no. Isem Xviii, pp. 197–200, 2016.
3. A. Nadhirah et al., “Properties of Fiberglass Crossarm in Transmission Tower - A Review,” Int. J. Appl. Eng. Res., vol. 12, no. 24, pp. 15228–15233, 2017.
4. N. M. Nor, S. T. Agusril, M. Y. Alias, A. M. A. Zaidi, and A. Shohaimi, “Dynamic Analysis of Sandwiched Composite Foldable Structure under Heavy Vehicle Load,” Appl. Mech. Mater., vol. 110–116, no. October, pp. 2331–2336, 2011.
5. H. Xin, Y. Liu, A. S. Mosallam, J. He, and A. Du, “Evaluation on material behaviors of pultruded glass fiber reinforced polymer (GFRP) laminates,” Compos. Struct., vol. 182, pp. 283–300, 2017.
6. H. Xin, A. Mosallam, Y. Liu, C. Wang, and Y. Zhang, “Analytical and experimental evaluation of flexural behavior of FRP pultruded composite profiles for bridge deck structural design,” Constr. Build. Mater., vol. 150, pp. 123–149, 2017.
7. Agusril and N. M. Nor, “Simulation Analysis of a Foldable Carbon Fiber Reinforced Polymer Bridge Prototype,” 2011 Natl. Postgrad. Conf. - Energy Sustain. Explor. Innov. Minds, NPC 2011, no. November 2015, 2011.
8. S. T. Agusril, N. M. Nor, and Z. J. Zhao, “Failure Analysis of Carbon Fiber Reinforced Polymer (CFRP) Bridge Using Composite Material Failure Theories,” Adv. Mater. Res., vol. 488–489, no. March, pp. 525–529, 2012.
9. P. Domone and J. Illston, Construction Materials: Their Nature and Behaviour, 4th Editio. 2010.
10. Wagners CFT Manufacturing Pty Ltd., “Product Guide,” 2016.
11. Ansys Inc., “ANSYS Composite PrepPost User’s Guide,” vol. 18.1, no. April. p. 440, 2017.
12. M. Muttashar, A. Manalo, W. Karunasena, and W. Lokuge, “Flexural behaviour of multi-celled GFRP composite beams with concrete infill: Experiment and theoretical analysis,” Compos. Struct., vol. 159, pp. 21–33, 2017.