© Universiti Tun Hussein Onn Malaysia Publisher's Office

IJIE

Journal homepage: http://penerbit.uthm.edu.my/ojs/index.php/ijie

ISSN: 2229-838X e-ISSN: 2600-7916

The International Journal of Integrated Engineering

Finite Element Analysis for Stress Distribution in a Proton Exchange Membrane Fuel Cell Stack

Nurato[1,2], Edy Herianto Majlan[1,*], Wan Ramli Wan Daud[1], Teuku Husaini[1], Masli Irwan Rosli[1], Abu Bakar Sulong[1], Darwin Sebayang[2]

[1]Fuel Cell Institute, Universiti Kebangsaan Malaysia, 43600 Bangi, Selangor Darul Ehsan, Malaysia

[2] Department Mechanical Engineering, Universitas Mercu Buana, Jakarta, Indonesia

*Corresponding Author

DOI: https://doi.org/10.30880/ijie.2019.11.07.029
Received 16 July 2019; Accepted 20 October 2019; Available online 15 November 2019

Abstract: The component design of a proton exchange membrane fuel cell (PEMFC) considerably affects pressure distribution in the PEMFC stack by creating uniform and effective pressure distribution. Assembly and component designs are essential in the PEMFC system to achieve optimal performance and durability of the PEMFC stack. Inadequate pressure in the stacking process can damage the cells and cause leakage and contact resistance. Moreover, an uneven distribution of pressure produces hot spots that can damage the system. Achieving the optimal design with reduced production cost requires pressure distribution simulation during the assembly. Finite element analysis (FEA) was used to analyze system behavior with pressure variation during assembly by using Autodesk Inventory software. This study discussed the geometric modeling and FEA of the tensile distribution of the PEMFC stack. The detailed components reported on the geometry, dimensions, and mechanical properties of PEMFC components, such as membranes, gas diffusion layers, end plates, and bipolar plates. Results from the simulations showed no significant difference in the deformation of cells in the PEMFC stack, with changes in tensile distribution.

Keywords: proton exchange membrane fuel cell, stress distribution, assembly, simulation

1. Introduction

A proton exchange membrane fuel cell (PEMFC) is an electrochemical device that directly converts chemical reaction energy into electricity. The chemical reaction mechanism is as follows [1][2] :

Anode : $H_2 \rightarrow 2H^+ + 2e^-$ (1.1)

Cathode : $\frac{1}{2} O_2 + 2H^+ + 2e^- \rightarrow H_2O$ (1.2)

Net Reaction : $H_2 + \frac{1}{2} O_2 \rightarrow H_2O$ (1.3)

The reaction between fuel and oxygen results in water as waste that is safe for the environment. The proton exchange membrane (PEM) fuel cell is a means to reduce dependence on fossil fuels. However, much work is needed before the fuel cell becomes a major fuel source. PEMFC is a type of fuel cell that uses a water-based polymer membrane, as its electrolyte, with a platinum-based electrode. Its ability to operate at low temperatures (less than 100 °C) results in high

*Corresponding author: edy@ukm.edu.my / edyhm71@gmail.com

efficiency and low pollution because the resulting waste is water. Many developed countries have invested in research to improve the efficiency of PEMFC, which can be applied in everyday life, such as in mobile electronic products [3].

The stack in which the current chemical reaction in the fuel cell produces direct current is the heart of a fuel cell system. Single cells produce only a small amount of electricity and only for small applications. A larger capacity of more than one fuel cell is produced by combining the cells in a series. The fuel cell set can consist of dozens of fuel cells that are determined by the capacity and application used. During the installation of the section, i.e., the installation phase, the pressure must be observed to obtain a stable fuel cell component. Damaged installations can cause problems, such as fuel leaks, cell damage, and binding resistance, with gaskets. The installation process in the PEMFC section is an important step in controlling the critical performance of the system [4][5]. Too many uneven distributions of pressure produce hot spots that can damage the system[6]. The above case can affect fuel cell performance because of the applied pressure.

In addition, excessive emphasis on the stacking process can interfere with the performance of the fuel component, such as the gas diffusion layer, which affects the performance of the fuel cell [7]. Furthermore, the bipolar plate made of graphite can show cracks [8]. Therefore, the design parameters of the stack influence the achievement and lifetime of the PEMFC. To predict the deformation caused by mounting pressure in a single PEMFC channel and 11 cells, Taymaz developed a model using finite element analysis (FEA)[9]. FEM model was used by Karvonen et al. to investigate pressure distribution in a multi-cell stack and to compared various end plate structures and material clamping force schemes [10]. Mikkola et al. studied a three-dimensional finite element analysis model to calculate the pressure distribution in a fuel cell stack [11]. FEA is a complicated problem-sharing method of the smallest element that can be solved in relation to one another. In this study, the finite element method (FEM) is used to investigate and analyze the PEMFC stack sectional stress distribution.

2. Materials and Methods

This study used the software simulation of Autodesk Inventor Professional 2018. This simulation is useful for performing analysis to prove the validity of the design. This is much more practical and economical when designing a design before making it a physical prototype. Pressure analysis is performed by Autodesk Inventor using finite element analysis method (FEA). Finite element analysis (FEA) is a computerized method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow, and other physical effects. Finite element analysis shows whether a product will break, wear out, or work the way it was designed. It is called analysis, but in the product development process, it is used to predict what is going to happen when the product is used. FEM is to discretize the model for analysis that can be solved in relation to one other. Practical applications are often known as FEA [12].

FEA can be used for engineering systems in which the only difference is the type of analysis performed [13][14]. Nowadays, many types of manufacturers use FEA, which may include dynamic analyzers, flow analysis, static analysis, and heat analysis. In this study, the models used contained one model of a single fuel cell stack and a fuel cell with 11 cells. Figure1 is a photograph of a single stack used for experiments in a laboratory. The single cell model in exploded is displayed (Figure 2) along with all components (endplate, current collector, bipolar plates, mea, gasket, bolt and nut) rendering Autodesk Inventor Professional software to perform FEA on the installation process of the fuel cell membrane. Different bolt tightness strengths were simulated in this study.

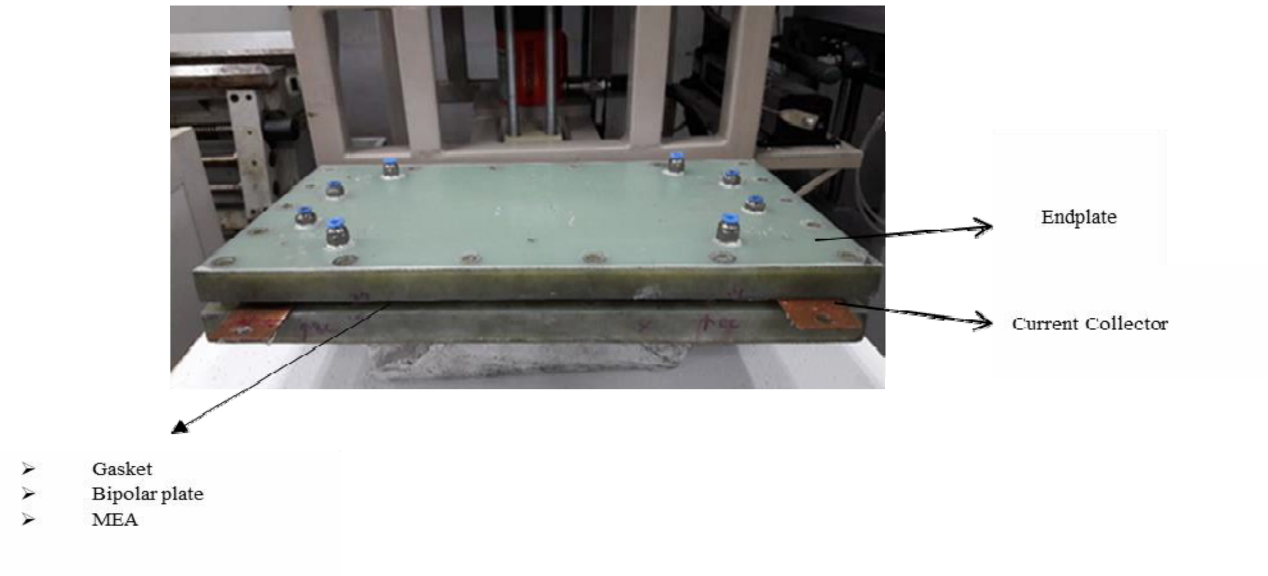


Figure 1 - Photograph stack PEMFC single cells in the lab

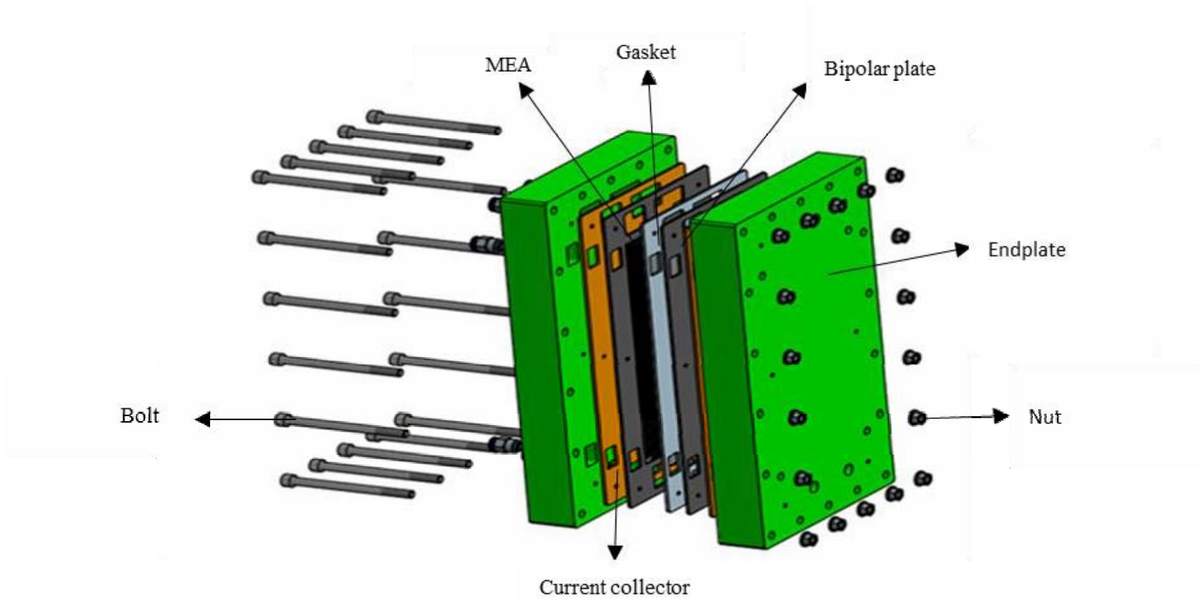


Figure 2 - Assembly of fuel cell stacks

3. Results and Discussion

The power that works on the fuel cell arrangement is bolt-binding power. The Von Mises stress analysis results of the FEA simulation obtained in the fuel cell stack was subjected to reinforcing forces : 508.7 N , 678.3 N, 847.9 N , and 1017.5 N. Von Mises criterion is based on the distortional energy, and states that a material element initially yields when is absorbs a critical amount of distortional strain energy which is equal to the distortional energy in uniaxial tension at the point of yield [15]. The property must be undefined, such that the simulation would not be executed. Material assumptions were fixed, indicating that no change is due to the time and temperature in the material property structure. The following table presents the components and properties of the materials used for the simulation model:

Table 1 - Dimensions and properties of the components

Component	Materials	Dimension (mm)	Young's Modulus (MPa)	Poisson's
End plate	Fiberglass	450 x 250 x 24	7.20E +04	0.21
Current collector	Copper	400 x 200 x 1.75	1.21E +05	0.34
Bipolar plate	Composite grafite	400 x 200 x 1.82	4.10E +03	0.17
MEA	Carbon sheet	305 x 129 x 0.3	2.20E +09	0
Gasket	Silicon/PTFE	400 x 200 x 0.6	1.19E +06	0.4[illegible]
Bolt/Nut	Stainless steel		1.97E +11	0

3.1. Simulation Constraints

Constraints provided restrictions on the movement or displacement of the model. In this paper a static analysis model with fixed constraints to maintain this model by eliminating translation and rotational movements. This constraint is done by reference to the position of the pedestal in the modeled design product. Constraints can overcome constraints, pin constraints, and friction constraints. Key step in structure analysis using finite element method is meshing process, in which continuous object systems are analyzed for discrimination so that the main structure becomes elements of smaller size and finite and finer numbers. The arrangement of the grid design of fuel cell arrangement was simulated, as shown in Fig. 4. This model used 327,839 elements and 568.570 nodes. Figure 5 shows that the overall design in blue is below 0.1–0.2 MPa. Data show that the fuel cell stack design model structure is safe to use and operational. The maximum von Mises 1 MPa stress value was observed in the biggest bolt tightening strength, which was 1017.5 N. The maximum value of the stress calculation was found in the bolt component of the binder. This is due to a very drastic change in the bolt construction, which is to change the bolt head diameter to the bolt itself.

3.2. Safety Factor

Safety factor is a factor used to evaluate the safety of a structure, where the strength of a material must exceed the actual strength [16]. Safety factor (SF) is important in overall structural analysis and planning. This factor is the subject of this research, and it has been widely discussed among mechanical engineers, especially in structural engineering. SF can be expressed in many ways. Typically, SF is a ratio of two quantities that have the same units, such as strength/stress, critical load/plied load, load to fail part of expected service overload, maxim um cycles/applied cycles, or maximum safe speed/operating speed [17][18]. In the above cases, SF was the factor used to evaluate the safety of the structure, where the strength of material should exceed its actual strength. The value of SF was derived from the comparison between the strengths of the resulting material from the construction, which are subsequently divided by the maximum von Mises pressure. Based on simulation results, the minimum SF exceeded the value of 1, indicating a secure structural state. Therefore, the structure of fuel cell arrangement is safe to use. $Sf = \frac{Sy}{\sigma_{Max}} \geq 1$ where as Sf: safety factor, Sy: material strength, σ max: maximum stress (von mises stress). The design is considered conservative if the value of Sy is always greater than σ max [19].

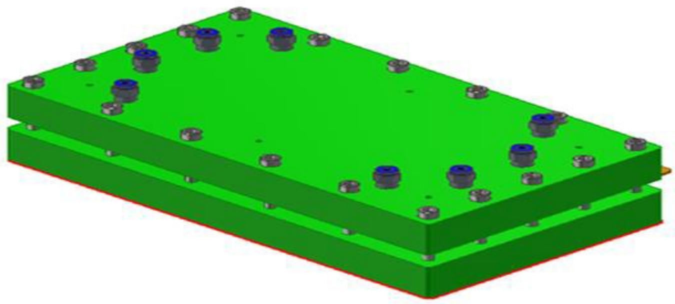


Figure 3 - Fixed constraint determination on fuel cell stacks

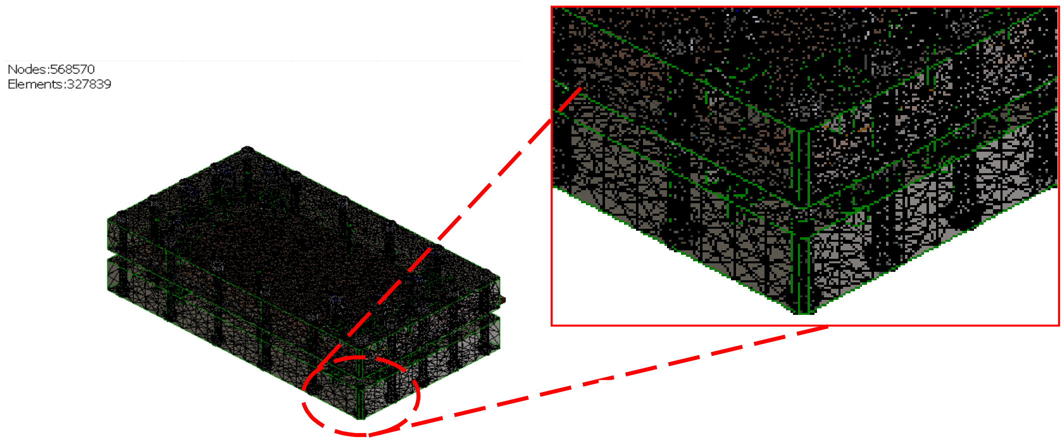


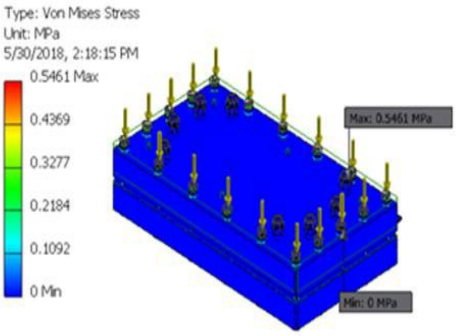
Figure 4 - Mesh view

3.3. Displacement Analysis

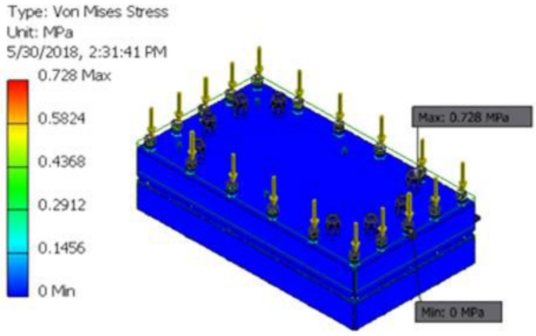
Figure 6 shows the result of the displacement analysis from the stress analysis, which obtained the result of the displacement simulation in the overall structure (shown in blue with a value below 4.2 x10^{-5} mm). A maximum displacement of 2.1 x 10^{-4} mm was obtained in the largest recorded strength (1017.5 N), which was affixed to the bolt component. The displacement results showed the deformed shape of the model in a scaled representation, based on the specified load conditions [20].

3.4. 11-Fuel Cell Stack Simulation

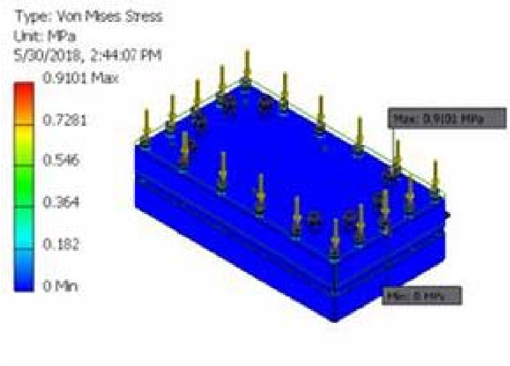
The 11-fuel cell stack assembly was simulated on the basis of the study on single-cell assembly. The von Mises stress from the FEA simulation analysis obtained in the fuel cell stack state was filled with various reinforcing strengths of 508.7 N, 678.3 N, 847.9 N, and 1017.5 N. Fig. 7 shows the meshing used for the model in which the mesh type used is tetrahedral in shape with elements equivalent to 1,087,860 pieces. In Figure 8, the overall design, which is shown in blue, is below 0.21 MPa. Data indicate the fuel cell stack model structure is safe to use and operational. The maximum von Mises 1.064 MPa stress value was obtained in the largest bolt tightening strength (1017.5 N). This maximum value was due to the tensile pressure that laid in the bolt component. In the tension joint, the bolt and clamped components of the joint are designed to transfer an applied tension load through the joint by way of the clamped components by the design of a proper balance of joint and bolt stiffness. The purpose of a bolt or group of bolts in all tensile and in most shear joints is to create a clamping force between two or more things, which we'll call joint members [21].



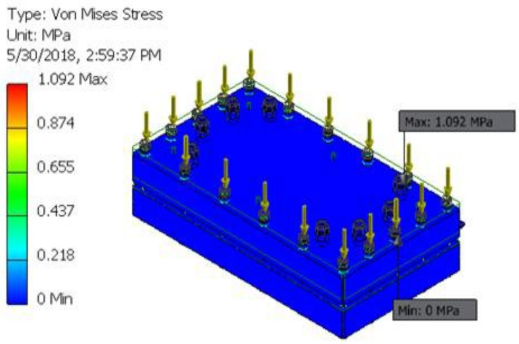
508.7 N



678.5 N

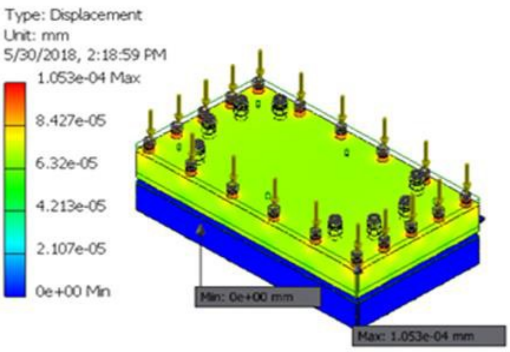


847.9 N

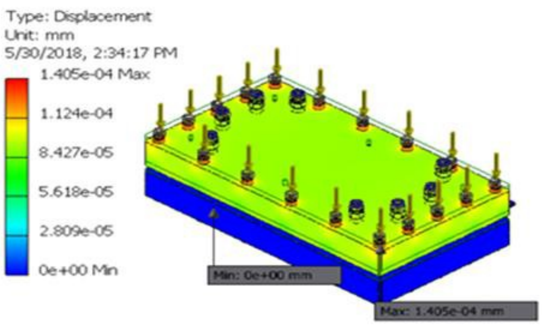


1017.5 N

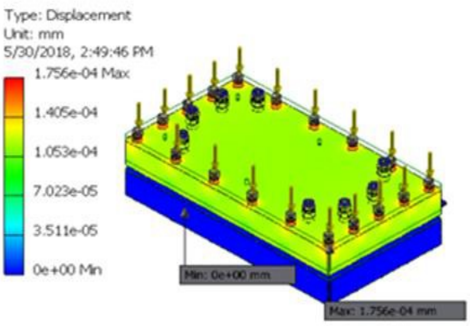
Figure 5 - Von Mises stress analysis result



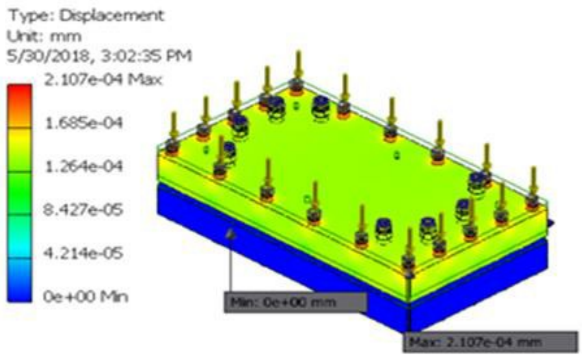
508.7 N



678.5 N



847.9 N



1017.5 N

Figure 6 - Displacement analysis result

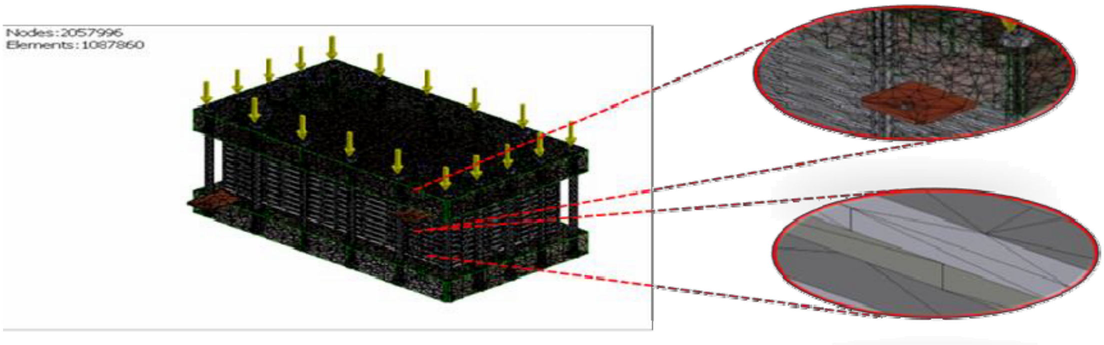
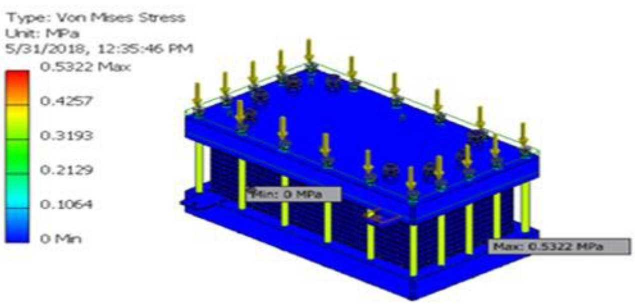
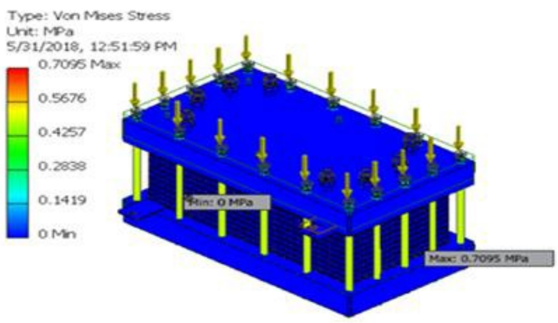


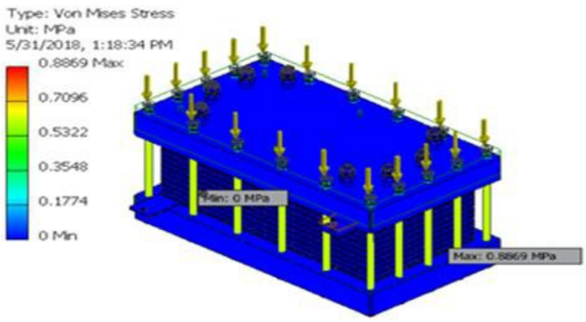
Figure 7 - Mesh of the 11-fuel cell stack



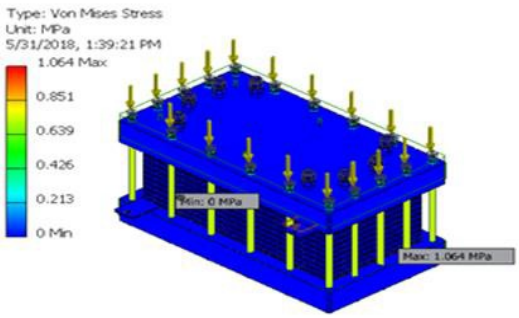
508.7 N



678.3 N



847.9 N



1017.5 N

Figure 8 - von Mises stress analysis result

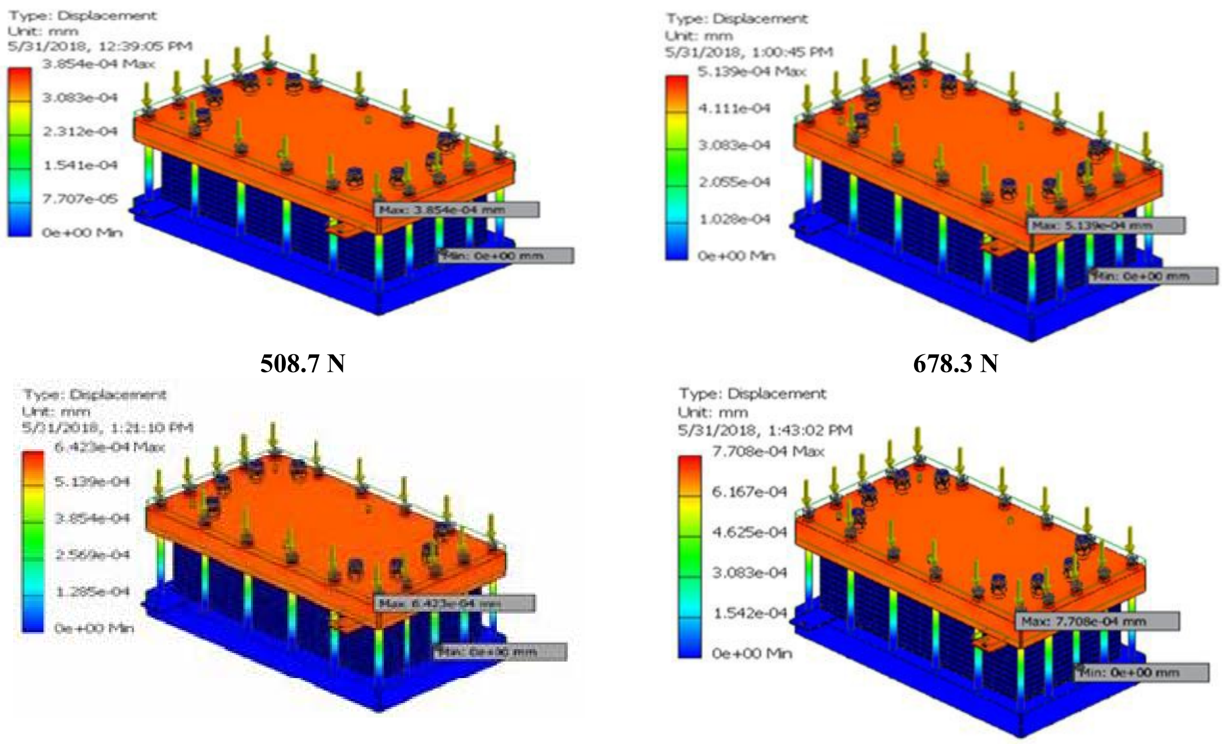
The SF value was derived from the comparison between the strength of the material from the construction divided by the maximum pressure tag. Figure 9 shows the displacement analysis result from stress analysis. The displacement simulation on the overall structure showed a blue color with a value below 1.542 x 10^{-4} mm, whereas the maximum displacement of 7.708 x 10^{-4}mm was observed in the bolt component of the binder. Simulation results were used to optimize cell performance. A detailed study of the cell experiments will be discussed in next papers.

4. Conclusion

Through FEA, parameters can be easily adjusted, and simulations can be performed without requiring materials or components to achieve the maximum standardization possible. The FEA model is only an approximation of the physical properties of structural components because materials in the model are homogeneous with linear response to stress. The FEA of PEMFC stack assembly is used on the basis of the Autodesk Inventory software to understand the deformation and pressure problems during installation. For single cells, the maximum value of von Mises stress is 1 MPa, and the maximum displacement is 2.1 x 10^{-4} mm. These values occur in the largest bolt tightening strength, i.e., 1075.5 N. The distribution of stress and deformation in a single cell and 11 cells with different loads reveals that construction is not affected by the maximum value of von Mises stress and is a good reference for actual assembly.

Acknowledgment

The authors acknowledge the financial support given by the Ministry of Higher Education Malaysia through Research Grant FRGS/1/2017/TK10/UKM/02/6.



508.7 N **678.3 N**

847.9 N **1017. 5 N**

Figure 9 - Displacement analysis result

References

[1] S. Revankar, and P. Majumdar. (2014). *Fuel Cells: Principles, Design, and Analysis*. CRC Press

[2] F. Barbir. (2005). "PEM Fuel Cells. Theory and Practice," pp. 1–436.

[3] C. H. Chien *et al.* (2016). "Effects of bolt pre-loading variations on performance of GDL in a bolted PEMFC by 3-D FEM analysis," *Energy*, vol. 113, pp. 1174–1187.

[4] I. Gatto, F. Urbani, G. Giacoppo, O. Barbera, and E. Passalacqua. (2011). "Influence of the bolt torque on PEFC performance with different gasket materials," *Int. J. Hydrogen Energy*, vol. 36, no. 20, pp. 13043–13050.

[5] C. Carral, N. Charvin, H. Trouvé, and P. Mélé. (2014). "An experimental analysis of PEMFC stack assembly using strain gage sensors," *Int. J. Hydrogen Energy*, vol. 39, no. 9, pp. 4493–4501.

[6] S. J. Lee, C. D. Hsu, and C. H. Huang.(2005). "Analyses of the fuel cell stack assembly pressure," *J. Power Sources*, vol. 145, no. 2, pp. 353–361.

[7] X. Q. Xing, K. W. Lum, H. J. Poh, and Y. L. Wu.(2010). "Optimization of assembly clamping pressure on performance of proton-exchange membrane fuel cells," vol. 195, pp. 62–68.

[8] Y. Zhou, G. Lin, A. J. Shih, and S. J. Hu. (2009). "Multiphysics Modeling of Assembly Pressure Effects on Proton Exchange Membrane Fuel Cell Performance," *J. Fuel Cell Sci. Technol.*, vol. 6, no. 4, p. 041005.

[9] I. Taymaz and M. Benli. (2010). "Numerical study of assembly pressure effect on the performance of proton exchange membrane fuel cell," *Energy*, vol. 35, no. 5, pp. 2134–2140.

[10] S. Karvonen, T. Hottinen, J. Ihonen, and H. Uusalo. (2008). "Modeling of polymer electrolyte membrane fuel stack end plates," *J. Fuel Cell Sci. Technol.*, vol. 5, no. 4, pp. 1–9.

[11] M. Mikkola, T. Tingelöf, and J. K. Ihonen.(2009). "Modelling compression pressure distribution in fuel cell stacks," *J. Power Sources*, vol. 193, no. 1, pp. 269–275.

[12] M. A. H. A. Ssomad, R. M. Hudzari, M. N. A. Noordin, S. M. Sapuan, N. Norhayati, and A. J. Soran. (2013) "Finite element analysis for stress distribution of hand tool harvester," in *Procedia Engineering*, vol. 68, pp. 219–224.

[13] H. Samekto, "Design Modifications of a Thin Wall Part from Aluminium to Magnesium," *Int. J. of Integrated Engineering*, vol. 2, no. 1, pp. 79-87

[14] M. I. Maksud, M. S. Yusof, and M. M. A. Jamil. (2012). "Study on Finite Element Analysis of Fine Solid Lines by Flexographic Printing in Printed Antennas for RFID Transponder," *Int. J. of Integrated Engineering*, vol. 4, no. 3, pp. 35–39.

[15] Gaudos.E.E. @005). *Fracture mechanics, An Introduction*, Second Edi., vol. 123, no. 10. Springer.

[16] E. J. Hearn. (1997). "Mechanics of Materials 1: The mechanics of elastic and plastic deformation of solids and structural materials," vol. 102, no. 27, pp. 9469–9474.

[17] Jack A. Collins, Henry R. Busby and John H. Staab. (2010) "Mechanical Design of Machine Elements and Machine, A Failure Prevention Perspective " Second Edition, The Ohio State University, pp 7, 2010.

[18] R. L. Norton. (2006) *Machine Design An Integrated Approach*, Third Edit. Massachusetts: Pearsen Prentice Hall.

[19] Shigley's. (2011). *Mechanical Engineering Design*, Ninth Edit. McGraw-Hills.

[20] Curtis Waguespack. (2014). *Mastering Autodesk® Inventor® 2014 and Autodesk® Inventor LT™ AUTODESK Official Press SYBEX* .

[21] J. H. Bickford. (2007). "Introduction to the Design and Behavior of Bolted Joints," *Fourth Edition Non-Gasketed Joints CRC Press.*