International Journal of Chemical, Materials and Biomolecular Sciences
ISSN: 2415-6620
Vol:3, No:3, 2009

Thermo Mechanical Design and Analysis of PEM Fuel cell Plate

Saravana Kannan Thangavelu

***Abstract*—**Fuel and oxidant gas delivery plate, or fuel cell plate, is a key component of a Proton Exchange Membrane (PEM) fuel cell. To manufacture low-cost and high performance fuel cell plates, advanced computer modeling and finite element structure analysis are used as virtual prototyping tools for the optimization of the plates at the early design stage. The present study examines thermal stress analysis of the fuel cell plates that are produced using a patented, low-cost fuel cell plate production technique based on screen-printing. Design optimization is applied to minimize the maximum stress within the plate, subject to strain constraint with both geometry and material parameters as design variables. The study reveals the characteristics of the printed plates, and provides guidelines for the structure and material design of the fuel cell plate.

***Keywords*—**Design optimization, FEA, PEM fuel cell, Thermal stress

I. Introduction

FUEL cell is an electrochemical device that applies hydrogen fuel and oxidant air to produce electricity, is considered as a zero-emission option to reduce air pollution and greenhouse effect [1]. Polymer Exchange Membrane (PEM) fuel cell is a promising alternative to the automotive internal combustion engine as a clean power plant due to its light weight, low operation temperature, and high efficiency [2].

A low-cost fuel cell gas delivery plate manufacturing method, using screen-printing, has been introduced recently introduced [2]. Where in the technique applies liquid phase conductive composite materials and a layer deposition process to build the 3D channel structure of the plate. The flow channels of a screen-printed plate are formed one solid Substrate using polymer-based composite material. In a fuel cell application, these fuel cell plates are routinely heated up to 100^{o}C with structure pressure loads.

The temperature induced thermal stress is found to be much higher than the structure loading stress. Thus, the thermal stress plays a key role in the structure design and optimization of the plates.

This study applies the virtual prototyping technique, which combined the computer aided design (CAD), and finite element analysis (FEA), on the fuel cell plate structure design that covers both plate geometry and plate material. Conventionally, most computational structure design deals with only geometry, which focuses on the shape of the design for mature materials with known performance. In this study, the material and its properties for the fuel cell plate is also to be determined.

The purpose of fuel cell plate structure design is to obtain an improved fuel cell gas delivery plate with certain materials based on the screen-printing manufacturing method [3]. The optimal design will provide the ideal geometry of structure and broader material performance for the materials development. The design optimization of plate geometry is carried out using key geometry parameters as design variables. The material design uses both geometry parameters and the key material parameter, the Yong's modulus, as design variables. Both of the geometry design and material design are conducted by minimizing the maximum stress within the plate, subject to the maximum strain allowed. The FEA structure simulation includes linear elasticity analysis, nonlinear elasticity analysis, and prestress analysis. These analyses are used as measurements of design objectives and constraints in the optimization

II. Computational Modeling

A. Solid Modeling

The model of a prototype fuel cell gas delivery plate is illustrated in Fig1. The linked parallel channels are connected to an air inlet and an air outlet at each end. Instead of machining, the prototype plate is formed by depositing multiple layers of composite material onto a solid substrate, using screen-printing to form the walls of the flow channels. The parametric solid model of the plate, generated using ANSYS, is constructed as a rectangular shaped building layer structure. The key design parameters of this solid model include: thickness of the printed layer, *H*, channel wall thickness *W*, and channel length, *D*. these parameters are used later as design variables in the optimization.

T.Saravana Kannan is with Swinburne University of Technology, Jalan Simpang Tiga 93350 Kuching Sarawak Malaysia (phone: +60 82 416353 Ext 7655; fax: +60 82 423594, e-mail: sthangavelu@swinburne.edu.my).

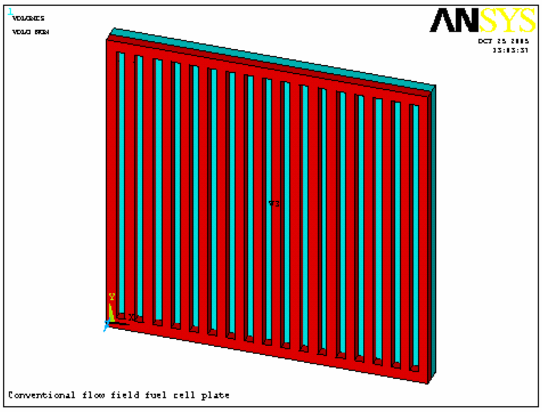

Fig. 1 Fuel Cell Flow Field Channels Gas Delivery Plate

B. FEA Structure Model

The finite element analysis is used to calculate the stress and strain in the printed channel wall with the assumption of a good adhesion between the building layer and the substrate. The model consists of a solid substrate (graphite or coated metal) with a "printed" flow-field channel wall layer (polymer-based composites). A uniform steady temperature distribution is assumed at any temperature step. The zero stress state or the initial condition for the calculation is at room temperature. The temperature of the entire plate is raised up to 100^{o}C to simulate the working condition of the PEM fuel cell.

Thermal stress is induced by the heating process due to the mismatch of the substrate and the printed channels with different thermal expansion coefficients. Since the polymer and polymer composites are quite thermal sensitive within this temperature range, nonlinear finite element analysis might be needed. The calculation is carried out by first using a linear FEA model, then to determine whether geometry nonlinearity and/or materials non-linearity needs to be applied and to calculate again if a nonlinear model are needed. For the fuel cell application, the plate has to undertake repeated temperature loads and structure loads. Both linear and nonlinear analyses are thus based on the elasticity assumption.

III. Design Optimization

In this present work, the design optimization is carried out for both plate geometry and material. Two different design objectives and two different sets of design variables are used in tandem. The most challenging issues facing the fuel cell plate material development are good conductivity, excellent electro-chemistry stability, sound structure integrity and low cost. Three key material properties, maximum stress, maximum strain, and stiffness, are directly related to the structure design requirement. For composite material development, minimum yield stress, minimum failure strain, and minimum young's modulus will support a broader selection of composite material composition.

The maximum von Mises stress relates to the lifetime of the structure. By selecting the minimum maximum von Mises stress, σ_{vm}, as the objective function, the optimal design will provide lowest material strength requirement, which the new developing composite material should be reached. Another material performance, maximum 1st principal strain, ε^1, is defined as a constraint for the optimization to satisfy material deformation restriction. The selections of design variables are made for two different considerations, plate geometry and plate materials, as discussed in the following subsections.

A. Geometry Design

In the plate geometry design optimization, traditional geometry parameters are used as design variables. The material parameters are assumed to be constants with their values determined from similar materials. The optimization considers the worst-case condition, minimizes the maximum von Mises stress, σ_{vm}, of the structure, subject to the constraint of the maximum strain level, [ε], the maximum allowed 1st principal strain ε^1. These stress and strain are evaluated using the ANSYS finite element analysis model. The optimization is carried out two variables for the plate geometry: the wall thickness W, and the height of the printed layer H.

The optimization is defined as:

$$\underset{\text{for } W,H}{\sigma_{vm}} = \text{minimum stress} \tag{1}$$

$$\text{Subject to: } \varepsilon^1 <= [\varepsilon] \tag{2}$$

Where the design variables, $X = [W, H]^T$

B. Materials Design

In the plate material design optimization, both geometry parameters and the material performance parameter of the plate are used as design variables. By adding the material stiffness as design variable, the optimal design will find the ideal value of the stiffness (or the Young's Modulus) of the composite material. The optimization shares the same objective and constraint functions of the plate geometry optimization.

The optimization is carried out using three key variables for the plate geometry and material: the wall thickness, *W,* and the height of the printed layer, *H*, and the material stiffness (or Young's Modulus), *E*.

The optimization is defined as:

$$\underset{\text{for } W,H,E}{\sigma_{vm}} = \text{minimum stress} \tag{3}$$

Subject to: $\varepsilon^1 <= [\varepsilon]$

Where the design variables,

$X = [W, H, E]^T$

IV. Results and Discussion

A. FEA Structure Analysis

The geometry model in Fig.1 is generated using ANSYS to form the finite element analysis model. It includes: epoxy-based composites, D= 0.055626 m, W = 0.001651 m, H = 0.001778 m; mass density 1299.52 Kg/m3, Young's modulus E = 3 GPa, Poisson's ratio 0.37, and coefficient of thermal expansion 6e-5 m/m^{o}K. Quadratic 10 node tetrahedral elements are used for the model. The element division along the long wall side is16, along all other parts are 6, and spacing ration is –2. Based on the geometry above, the total elements of 13470 are generated with high density at the end and corner area.

TABLE I
LINEAR AND NON LINEAR STRUCTURE ANALYSIS

Simulation set	Max VonMises stress (nodal)	1st Principal Strain (element)	Max Displacement	Comment
(1)Structure loads only	1.25 MPa	270 με	0.496 μm	Safe
(2).Temperature load only	45.069 MPa	11343 με	13.3 μm	un safe
(3)structure and temperature load	47.5 MPa	12050 με	15.6 μm	un safe
(4)Temperature prestress	46.82 MPa	11604 με	13.4 μm	un safe
(5). Nonlinear	17.24 MPa	9127 με	13.6 μm	Safe

TABLE II
DESIGN OPTIMIZATION IN LINEAR AND NONLINEAR ANALYSIS

Design Set	Design variable	Starting point	Optimum	Minimal stress
(1) Linear geometry design	W,H	W=0.001651m, H=0.001778m	W=0.00153 m H=0.00158 m	44.75 MPa
(2) linear material design	W,H,E	W=0.00153m, H=0.001778m E= 3 GPa	W=0.00179m H=0.00199 m E = 1.02 GPa	16.23 MPa
(3) Nonlinear geometry design	W,H	W=0.00153m, H=0.00158m	W=0.000525m H=0.00101 m	16.07 MPa

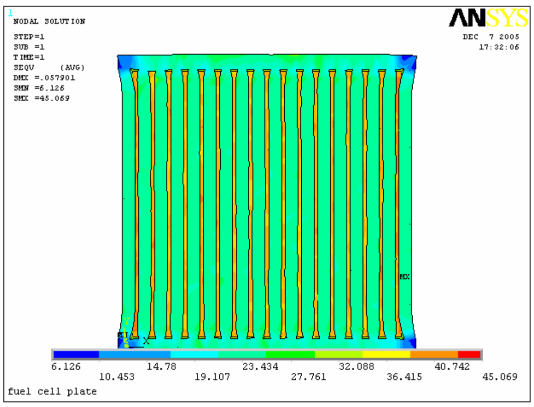

Fig. 2 Thermal Stress Distributions with Temperature load

Due to the complex geometry of the plate, the FEA model needs 12400 to 13600 elements under the three geometry variables, D, W, and H, used in the design optimization. Moreover, some of these elements have high aspect ratio and large element edge angle (e.g. >165^{o}), and the model is subject to joint temperature and structure loads. Therefore different load steps and sub steps are necessary to achieve good convergence. In the analysis, 5 sub steps are needed for each load step. The equilibrium iteration is automatically determined by the ANSYS program.

The analysis is first conducted using linear material elasticity analysis. The simulated plate is subject to inner channel pressure load, top uniform pressure, and temperature field. The result using different data sets of the analysis is shown in Table I. Set (1) of this linear analysis is carried out with pure structure loads, including fixed bottom boundary condition, 204 KPa inner channel gas pressure, and 544 KPa uniform stacks compressing top pressure, without a temperature field. Set (2) of the linear analysis covers only temperature load: fixed bottom boundary condition, and an 100^{o}C temperature field with 20^{o}C room temperature as reference, without any structure loads. Set (3) includes both structure and temperature loads: fixed bottom boundary condition, 204 KPa inner channel pressure, 544 KPa uniform top pressures, and a 100^{o}C temperature field.

The results are given in the table 1.According to the results on sets (1) and (2) from table 1, the temperature induced thermal stress is much higher than that of the structure loads. In test (2), the epoxy- based composites material is working at a high stress level quite close to the material strength of 50-60 MPa. Test (1) and (3) indicates the temperature field play a key role on the maximum stress and strain occurred on the structure. The nonlinear materials elasticity analysis, set (5) of Table1, produces a maximum von Mises stress 17.24 MPa, a 1st principal total strain 9127με and a maximum displacement 13.5 μm. These are with in the working range for epoxy materials. The typical stress distribution, the result simulation set (2) is shown in Fig.2

In summery, the finite element analysis indicates that high stress level is induced by the temperature load. Material nonlinearity needs to be considered in the structure design. In addition, the corner effect of maximum stress and strain is important for the geometry design.

B. Optimal Geometry and Material Design

The structure optimizations of the plate include both geometry and material considerations. One of the built-in optimization functions of ANSYS, advanced zero order method or sub-problem method, is used for the plate structure optimization. The routine applies a search approach, in which a few sets of objective and constraint functions are approximated by several fitted surfaces. The optimization routine searches through the fitted surface, instead of the original function surfaces, to improve search efficiency. The fitting controls use quadratic functions for the objective and constraint functions.

Design optimization is applied to both linear and nonlinear material analysis, based on the Simulation Sets (3) and (5) of Table I, discussed previously. Based on the geometry design consideration of linear analysis set (3), the geometry parameters wall thickness, W, and layer thickness, H, are as design variables of the Design Set (1), which W ranging from 0.5 mm to 1.8 mm; H ranging from 1 mm to 2 mm. The chosen starting point is at W = 0.001651 m, and H = 0.001778 m. The 1st principal strain is used as the design constraint with an upper limit at 12000 με. The objective function, von Mises stress, is minimized.

In this optimization, all other items, i.e. Young's modulus E = 3 GPa, are fixed same as the previous linear analysis simulation set (3). The search often ends prematurely at local optima. Many restarts are tested at different start points, including the identified true global minimum at W = 0.00153 m and H = 0.00158m. The minimum stress of Design Set (1) is 44.75MPa, which is still very high for the epoxy based composite material, working around 100^{o}C (refer to Table II). No acceptable geometry design for the fuel cell plate can be achieved based on this selected materials. Modification on the material thus needs to be considered.

The material Design Set (2) applies both geometry parameters and material's Young's modulus E as design variables. The E is allowed to vary from 1 GPa to 10 $GPa,$ while W and H can change over the same range as Design Set (1). For easily comparison, the optimization routine starts from the design optimum of the Design Set (1). The optimization routine converged at a minimum stress of 16.23 MPa with the geometry parameters W = 0.00179 m, H = 0.00199 m, and material parameter E = 1.02 $GPa,$ respectively. This result has acceptable stress/stain levels and geometry parameters. The lower Young's modulus of 1.02 GPA still provides a reasonable strain value 11885 με. The value is used to identify the new composite material.

When applying design optimization through nonlinear FEA analysis, the material does not has to be changed, or the Young's modulus E of the material needs not to be modified, since the stress on the model simulation set (5) is only 17.24 MPa. Nonlinear geometry Design Set (3) in Table II uses the same geometry parameters and range of the Design Set (1) as design variables. The starting point of the search (W =0.00153 m, H = 0.00158 m) is the same optimum of Design Set (1) for the ease of comparison.

The optimization converged after 21 iterations with the best design found at: wall thickness 0.525 mm and layer thickness 1.01 mm, which are at almost at the lower bounds of the two design variables and the design constraint, 1st principal strain 7615 με. The minimized objective function von Mises stress has been lowered to 16.07 MPa. All values at this design optimum are ideal for the fuel cell plate structure.

Fig. 3 shows a plot of the Maximum von Mises stress vs. Layer Thickness in optimization search. Alternative structure design can be selected based on this plot. The optimal design, at 16.26 MPa, has 5.68% improvement compared to the original structure analysis of 17.24 MPa.

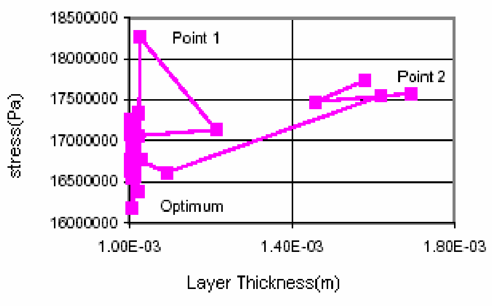

Fig. 3 Maximum Von Mises Stress Vs Layer Thickness

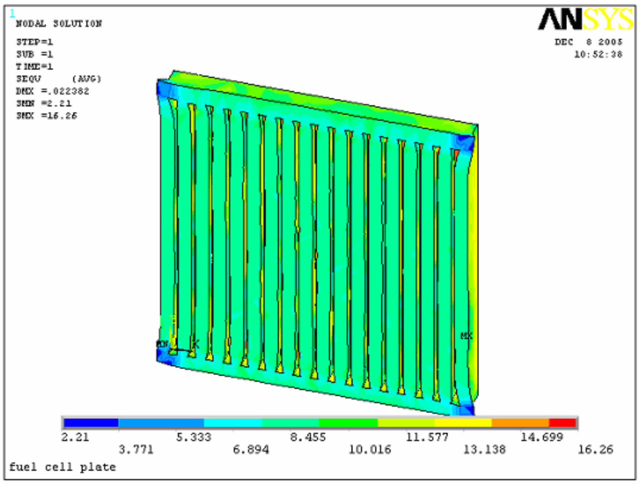

Fig. 4 Minimal Stress in Non Linear Optimizations

TABLE III
ALTERNATIVE STRUCTURE DESIGN COMPARISONS (NON LINEAR)

Parameters	Initial	Point 1 (Max Stress)	Point 2 (Converged)	Optimal
W (m)	0.00153	0.00138	0.000507	0.000525
H (m)	0.00158	0.00103	0.001003	0.001007
Von Mises stress (MPa)	17.24	18.27	17.09	16.26
Improvements	-	+5.96%	-0.87%	-5.68%

Detailed comparison of the optimization results are listed in Table III. The ratio between *H* and *W* is approximately 2, which is acceptable for the printing process.

In summery, the optimal structure design of the fuel cell plates, which includes both geometry design and material design, provides abundant information on the plate geometry and corresponding material. Design Set (2) shows that the minimum stress level can be achieved with larger allowable geometry dimensions and lower Young's modulus material, given the linear material consideration. Design Set (3) indicates that acceptable stress values can also be obtained with smaller geometry dimensions under the Point 1 Optimum Point 2 assumption that the composite material possesses a nonlinear material property. The material does not have to be altered and an unchanged Young's modulus can be used.

The optimal structure design of the printed plate identifies the minimum maximum stress that can be used for the composite material, ensures that the maximum stress and strain are within the allowed range, and specifies the optimal plate geometry and material. These optimizations provide a broader scope to the feasible composite material, by identifying the lowest acceptable strength of the plate, in the channel material development. The optimizations also specify the corresponding stiffness relating to the Young's modulus of the composite material to be developed.

V. CONCLUSION

In this work, advanced computer modeling and finite element structure analysis are used as virtual prototyping tools for the optimization of the plates at the early design stage. The design optimization provides a useful tool to combine both geometry and material designs for the plate structure to form a close form. The design analysis and optimization can provide guidelines for the development of the fuel cell plate and many other mechanical components of similar types.

ACKNOWLEDGEMENT

The support extended by the CADD Center Pondicherry and Management of the IFET College of Engineering, Villupuram (T.N) India is thankfully acknowledged and thankful to the management of Swinburne University of technology, Sarawak Malaysia.

REFERENCES

[1] Dong, Z. "Fuel Cell Flow-Field Structure Formed by Layer Deposition," UK Patent GB 2 336 712 A.(1999)
[2] Menon, U., Chapter 6: "Rapid Prototyping of Hardware and Software, Concurrent Design of Products, Manufacturing Processes and Systems" Gordon and Breach Science publishers, p133, (1998).
[3] Pastula, M.E.,, Radiator Stack PEM Fuel Cell Architecture, System Modeling and Flow Field Design, M. A. Sc. Thesis, University of Victoria.(1997)
[4] Perry, M. L., Newman, J., and Cairns , E. J., Mass Transport in Gas Diffusion Electrodes: A Diagnostic Tool for Fuel Cell Cathodes, Journal of the Electrochemical Society, v145, n1, p5-151(1998)
[5] Shi, L. T., and Tu, K. N. "Finite Element Stress Analysis of Failure Mechanisms in a Multilevel Metallization Structure", Journal of Applied Physics, v77, n7, (1995).

T.Saravana Kannan received B.Eng from Madurai Kamaraj University, Tamilnadu India and M.Tech from Pondicherry University, Pondicherry India in the years 1999 and 2006 respectively. He is currently working as lecturer at School of Engineering and Science, Swinburne University of Technology Sarawak Malaysia. He has presented and Published 5 international conference papers. His current research interest is direct energy conversion, renewable energy and Thermo fluids. He is having more than 10 year teching experience and He is a Life member of Indian Society for Technical Educations.