STRESS ANALYSIS FOR AN OPERATING SOFC STACK

Mark Boersma, Bryce Sharman and Mehrzad Tabatabaian
Global Thermoelectric, Inc.
4908 - 52nd Street S. E.
Calgary, Alberta T2B 3R2, Canada

ABSTRACT
Thermal stress analysis for a SOFC stack under combined pressure loading and rejected heat due to stack operation is presented. An integrated temperature function of the rejected heat generated by the cell and ambient temperature settings provided a first-hand study of the effects of thermal expansion, stress and deformation for a steady-state case. The analysis involved using the temperature gradient across the cells while 20 Amps was being drawn from them. To allow for uniform thermal expansion of the stack without causing erroneous stresses, the constraints were set to allow frictionless expansion in the horizontal plane on the bottom compression plate while also fixing the bottom compression plate in the vertical plane. Displacement observations showed maximum expansion and stresses occurring at maximum temperature once the unit cell reached that temperature, however, minor changes were seen in stress variations, displacements and deflections.

INTRODUCTION
The study of thermal expansion and contraction within solid oxide fuel cells (SOFC) due to thermal cycling is an essential aspect in the development of the fuel cell system. Unless each material has the same coefficient of thermal expansion, thermal stresses will undoubtedly appear within the unit cell and the stack. Modeling such a phenomenon allows further in-depth studies and observations that ordinarily might not be seen inside the hot-box region. The information derived from the dynamic model is aimed at optimizing component design, assisting in failure analysis and reducing the amount of design iterations in order to achieve desired results.

The finite element model developed at Global Thermoelectric using ANSYS (1) incorporates thermal variations within the fuel cell and applied pressure and constraints. The heat transfer and stress analyses have been separated and were sequentally analyzed; first the heat transfer, followed by the stress analysis. The assumption behind this idea is that stresses do not affect the heat transfer in the fuel cell.

An investigation of Global’s modeling results has allowed for further studies of thermal stresses and component optimization.
Background

Solid oxide fuel cells run at high temperatures (700-800°C), causing thermal expansions within each material. Unless the rates of thermal expansion are matched for every material inside the stack, stresses will become apparent. Even with pressures acting on the unit cell from external sources and compressing the unit cell together, the materials still expand and displacements still occur. Although it is known that the stresses exist and do have an effect on the unit cell, they have not been studied in any great detail. Possible attempts at investigating this phenomenon might help to have better insight into the behaviour of the unit cell and the stack.

What is currently known is the temperature inside the hot box when the unit cell is put through the thermal cycling. From this information, analyzing the stresses throughout the unit cell and information on internal heat generation from within the unit cell using theoretical data acquired from a computational fluid dynamics analysis on an adiabatic unit cell, some results were derived. Analyzing both the steady-state and transient structural behavior of a unit cell due to the thermal stresses from internal heat generation and external heat acting on the unit cell would allow a better chance to understand what each condition offered to the problem. Once the analysis was completed, it was compared with results from a variety of other analyses that are currently being researched and tested.

Several trials with different boundary conditions were attempted. After analyzing several trial scenarios with varying boundary conditions for the steady-state case, the boundary conditions that produced the most accurate results involved constraining the bottom surface so that it could not move in the vertical direction, \( U_y = 0 \), and applying a surface pressure to the top interconnect. These boundary conditions were also used for the transient analysis.

In determining the theoretical displacements, deflections and stresses, the equations that were focused upon included (2):

\[
\sigma = E\alpha(\Delta T) \tag{1}
\]

where \( \sigma \) is the stress, \( E \) is the modulus of elasticity, \( \alpha \) is the coefficient of thermal expansion and \( \Delta T \) is the change in temperature.

Another important aspect of analyzing the unit cell was measuring the thermal shock resistance (TSR) (3). This particular characteristic is applied only to the ceramics being used in the unit cell and measures the maximum rate at which the temperature changes.

\[
TSR = \frac{\sigma \cdot k}{E \cdot \alpha} \tag{2}
\]

where \( \sigma \) is the fracture strength, \( k \) is the thermal conductivity, \( E \) is the Young's modulus, and \( \alpha \) is the coefficient of thermal expansion.

1474 Electrochemical Society Proceedings Volume 2003-07
RESULTS

Unit Cell

Once the mesh had been created, the thermal analysis for the steady-state solution was performed. The temperature distribution that was applied throughout the unit cell was collected from the CFD results that show the internal heat generation with an adiabatic boundary condition (4). A polynomial curve fit for the temperature was created using MathCAD. The equation is set under adiabatic boundary conditions for the unit cell, with a structured mesh and input current of 32 Amps. The thermal analysis was solved and the results are shown in Figure 1. It should be noted that this temperature distribution can be altered or modified at any time to account for other factors or include other conditions in the future that might arise. ANSYS contains a wide variety of thermal and mechanical options that continue to be explored. The model is capable to a variety of different inputs and can also be exposed to many different situations.

The maximum displacement magnitude occurs at the opposite corner from the heat source (Figure 2). Most of the unit cell was at the lower end of the pressure, and some elevated stress levels at the location where the manifolds connect to the inlets. The cell (cathode, anode and electrolyte) acted in much the same fashion the unit cell itself. The displacement was maximized at the opposite end of where the cell was the hottest while the corners showed some slight elevations in stress. The seals were also analyzed; they revealed displacements in every direction (most likely due to the pressure), and had stresses and strains acting in a similar manner.

The transient analysis for the unit cell mesh had several different aspects compared to steady state analysis. Since this analysis incorporated a certain length of time, the main idea was to capture the change in temperature affecting the stresses and displacements. The temperature distribution that changed over time was taken from the stack unit cell distribution when the stack is placed in a hot box. Using this temperature distribution over time, the transient thermal analysis could be transferred over to a structural case and also analyzed over time with the thermal loading. It should also be noted that no internal heat generation was added to this case. Although this gives a less accurate solution for now, it does show the effects of a higher temperature on the unit cell. Results of the transient analysis showed an expanding and contracting unit cell that corresponded to the thermal change in time. The largest deflections were seen at the maximum applied temperature although once that temperature was reached, little to no more deflections or displacements took place. The stresses also behaved in a similar fashion, increasing until the maximum temperature was experienced but not really further increasing once that temperature had been reached. In the cool down process, the unit cell contracted in size and the stresses decreased in magnitude.

Stack

When the displacement magnitude was analyzed for the cross-sectional area (Figure 3), it showed that the least amount of displacement was in the lower center region of the stack. It also revealed a parabolic like behaviour throughout the stack beginning at the center. Last, several components of the stack were studied for the displacement as well. These components showed the same behavior as the stack, although not to the same degree. The highest displacements are located in each corner (Figure 4). As one goes progressively further down, the displacement in the vertical direction decreases.
However, one can extrapolate that this would have the same effect with a larger stack, which would extend down far further. Also, the displacement in the horizontal direction remains approximately the same as one progressed down as well, indicating uniform expansion throughout the cell. Friction, however, was excluded in this analysis. After studying the cross sectional area and the cell and interconnect the same trend continued, in that most of the components exhibited a minimal Von Mises stresses throughout them. After studying the cross section and the interconnect cell, the diagrams showed a slightly elevated stress concentration near the edges of the cell. This is perhaps attributed to the fact that there is a large amount of mass stacked at those particular regions, including many seals, cells and interconnects.

**Thermal Shock Resistance**

The thermal shock resistance (TSR) applied really only to the cell and its material properties. This was calculated as a precaution to check the maximum rate at which the cell could withstand a change in temperature. This value proved to be well above the rate at which Global both heats and cools the stack.

![Figure 1. The thermal gradient across the unit cell.](image1)

![Figure 2. The displacement magnitude of the unit cell at steady state.](image2)
CONCLUSIONS

Design iterations have been reduced with the use of stress analysis modeling, and design improvements and alterations have been made as a result. With preliminary modeling on SOFC designs, iterations can be further reduced and problematic areas can be analyzed before design completion.

ACKNOWLEDGEMENTS

The development of SOFC technology is supported by Global Thermoeletric, Inc., and assistance has been provided by the ANSYS technical support.

REFERENCES

1. ANSYS User Guide 6.0, 2001.

2. F. P. Beer and E. R. Johnston, Jr., Mechanics of Materials, 2nd ed., pp. 64-65 (1992).

3. K. Chawla, Composite Materials Science and Engineering, 2nd ed., p. 245 (1998).

4. W. Dong, G. Price, B. Wightman, D. Ghosh, M. Tabatabaian, "Modeling of SOFC stack and system components," presented at 5th European SOFC Forum, J. Huijsmans, Editor, 2, pp. 929-936, Lucerne, Switzerland, (2002).