Engineering Failure Analysis 16 (2009) 874–887

Contents lists available at ScienceDirect

Engineering Failure Analysis

journal homepage: www.elsevier.com/locate/engfailanal

# Reliability and life study of hydraulic solenoid valve. Part 1: A multi-physics finite element model

S.V. Angadi [a], R.L. Jackson [a,*], Song-Yul Choe [a], G.T. Flowers [a], J.C. Suhling [a], Young-Kwon Chang [b], Jung-Keol Ham [b]

[a] *Department of Mechanical Engineering and NSF Center for Advanced Vehicle Electronics (CAVE), Auburn University, 270 Ross Hall, Auburn, AL 36849, USA*

[b] *Korea Testing Laboratory, Seoul, South Korea*

ARTICLE INFO

*Article history:*
Received 3 July 2008
Accepted 2 August 2008
Available online 15 August 2008

*Keywords:*
Automotive failures
Failure mechanism
Finite element analysis
Thermal expansion
Valve failures

ABSTRACT

A comprehensive multi-physics theoretical model of a solenoid valve used in an automobile transmission is constructed using the finite element method. The multi-physics model includes the coupled effects of electromagnetic, thermodynamics and solid mechanics. The resulting finite element model of the solenoid valve provides useful information on the temperature distribution, mechanical and thermal deformations, and stresses. The model results predict that the solenoid valve is susceptible to a coupled electrical–thermo-mechanical failure mechanism. The coil can generate heat which can cause compressive stress and high temperatures that in turn could fail the insulation between the coil wires. The model facilitates the characterization of the solenoid valve performance, life and reliability and can be used as a predictive tool in future solenoid design.



## 1. Introduction

A solenoid valve (SV) is an electromechanical device used to control the flow of gas or liquid by passing an electric current through a coiled wire, thereby altering the valve position. A schematic of a SV and the cross-section of an unfailed SV are shown in Figs. 1 and 2, respectively. SVs are used in various applications ranging from automobiles (as in transmission control, hydraulic power brake system, anti-lock brakes, traction control, etc.), aerospace and nuclear power plants to irrigation and water treatment, boom control in an agricultural vehicle. They also are widely used for domestic purposes, namely, washing machines, gardening and commercial dishwashers. Due to the extensive use of the solenoid valve it is very important to fully understand its behavior and the mechanisms which govern its reliability.

To characterize and improve the overall performance (involving electromagnetic, mechanical and thermal fields) of a SV, a two part paper series is considered. In this paper (that is, Part 1), solenoid valves used in the control of automobile transmissions are investigated to characterize their performance and reliability through theoretical modeling (i.e., multi-physics finite element modeling). The second paper will investigate the solenoid valve using experimental methods.

The SV finite element model predictions are used to determine the testing methodology. Additionally, experimental test results of SVs are correlated and validated with the finite element model predictions and results. A detailed discussion on the experimental test set up, experimental methodology, and the accompanying results can be found in the Part 2 paper.

Most theoretical models of solenoid valves are designed to consider their dynamic characteristics so that a control scheme can be designed and optimized [1–9]. Most of these previous works do not consider the coupled thermo-mechanical

* Corresponding author. Tel.: +1 334 844 3340; fax: +1 334 844 3307.
*E-mail address:* robert.jackson@eng.auburn.edu (R.L. Jackson).

1350-6307/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.engfailanal.2008.08.011

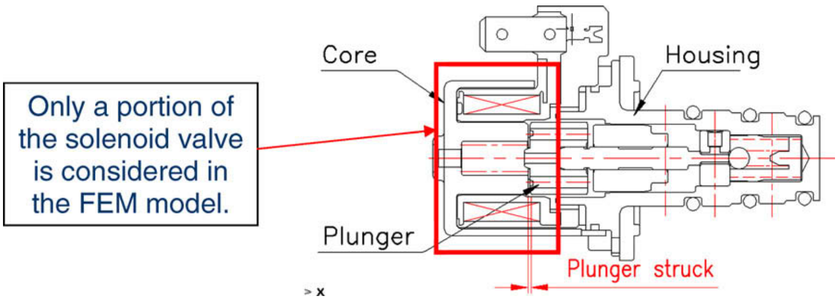


**Fig. 1.** Schematic of solenoid valve and portion considered in model.

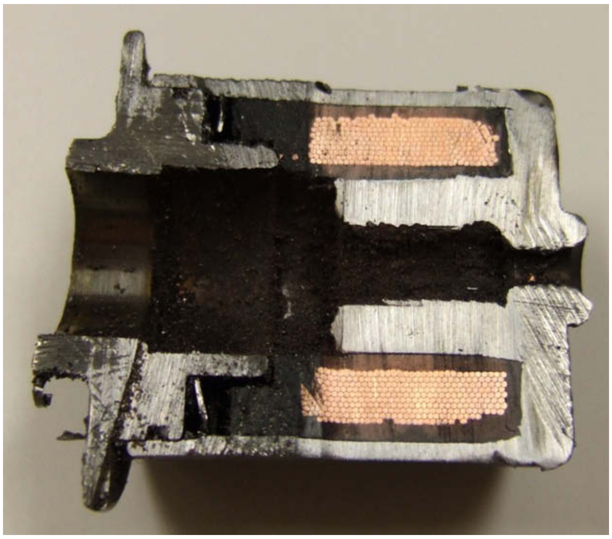

**Fig. 2.** Cross-section of an unfailed solenoid valve.

behavior of the solenoid valve. However, there are a few past works that have modeled the pseudo-static performance of the solenoid valve using finite elements and other computational methods [10,11]. The current work will develop a new multi-physics model of a solenoid valve which will consider the true coupled nature of the mechanisms that govern solenoid performance and reliability. In this work, the focus is made mostly on the thermo-mechanical failure mechanisms of the solenoid valve. It should also be noticed that in many cases these various failure mechanisms do not occur independently. Several mechanisms may be initiated or progressed due to the occurrence of another mechanism. For instance, the Joule heating could cause the solenoid temperature to rise significantly. The seals in the solenoid could then degrade due to the elevated temperatures, then causing the solenoid to leak and perhaps fail.

## 2. Modeling methodology

A multi-physics (thermal, mechanical and electro-magnetic) model of the solenoid valve (SV) is developed in the finite element package Ansys™ (see Fig. 3), the results of which are discussed in the following section. The model solves the coupled fields of equations and thus captures the effects not normally considered by conventional uncoupled finite element models. For instance, the heat generated by the solenoid coil will be due to Joule heating. This heating will increase the temperature which will cause thermal expansion and stresses in the coil and the surrounding parts. Therefore, these different fields are coupled together and for improved accuracy, should be solved simultaneously. This results in a very powerful tool that can be used to characterize solenoid valve performance.

The mechanical field of the problem considers the stresses and strains of the material, and how it will deform and possibly fail due to over stressing. The theory of elasticity is used to model the deformations in the material. Then three dimensional Hooke's law, which relates the stresses and strains, is given in cylindrical coordinates $(r,\theta,z)$ as

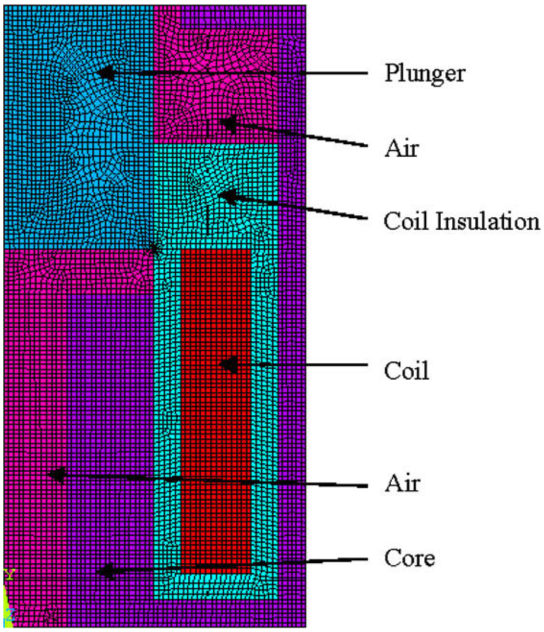


**Fig. 3.** A plot of the finite element mesh used to model the solenoid valve.

$$\varepsilon_r = \frac{1}{E}[\sigma_r - \nu(\sigma_\theta + \sigma_z) + \alpha\Delta T] \tag{1}$$

$$\varepsilon_\theta = \frac{1}{E}[\sigma_\theta - \nu(\sigma_r + \sigma_z)] + \alpha\Delta T \tag{2}$$

$$\varepsilon_z = \frac{1}{E}[\sigma_z - \nu(\sigma_r + \sigma_\theta)] + \alpha\Delta T \tag{3}$$

$$\gamma_{r\theta} = \frac{2(1+\nu)}{E}\tau_{r\theta} \tag{4}$$

$$\gamma_{rz} = \frac{2(1+\nu)}{E}\tau_{rz} \tag{5}$$

$$\gamma_{\theta z} = \frac{2(1+\nu)}{E}\tau_{\theta z} \tag{6}$$

where $E$ is the elastic modulus, $\nu$ is the Poissons ratio, $\sigma$ is the normal stress, $\tau$ is the shear stress, $\varepsilon$ is the normal strain, $\gamma$ is the shear strain, $\alpha$ is the thermal expansion coefficient, and $\Delta T$ is the change in temperature of the material. In the current analysis the solenoid valve will be modeled as being axisymmetric in geometry and loading. For axisymmetric cases, the displacement/strain relations are

$$\gamma_{r\theta} = \gamma_{\theta z} = \tau_{r\theta} = \tau_{\theta z} = u_\theta = 0 \tag{7}$$

$$\varepsilon_r = \frac{\partial u_r}{\partial r} \tag{8}$$

$$\varepsilon_\theta = \frac{u_r}{r} \tag{9}$$

$$\varepsilon_z = \frac{\partial u_z}{\partial z} \tag{10}$$

$$\gamma_{rz} = \frac{\partial u_r}{\partial z} + \frac{\partial u_z}{\partial r} \tag{11}$$

where $u$ is the normal displacement. The equations for continuity are also satisfied in the FEM software. Notice that in Eqs. (1)–(3), the strains are also dependant on the temperature of the material. Since the temperature in the solenoid valve will not be uniform, the thermal field must also be solved to obtain temperature.

The two-dimensional steady-state heat transfer equation using cylindrical coordinates $r$ and $\theta$ is

$$\frac{\partial^2 T}{\partial r^2} + \frac{\partial^2 T}{\partial z^2} + \frac{1}{r}\frac{\partial T}{\partial r} + \frac{1}{r^2}\frac{\partial^2 T}{\partial \theta^2} = \frac{1}{k}Q(r,\theta,z) \tag{12}$$

where $Q(r,\theta)$ is the volumetric heat generation, $T$ is the temperature distribution, which must be periodic or constant around the circumference. In the solenoid valve problem there will be several sources of heat such as Joule heating and friction. Since the contact force on the plunger surfaces should be small, the friction force should be small and the frictional heating should be negligible. Probably the most significant source of heating will be from the Joule heating of the coil. Therefore, the thermal and electrical fields are coupled, and the mechanical and thermal fields are coupled. The electrical and thermal fields may also be affected by the deformations of the solenoid, as it may affect the flow of current and heat.

The finite element model is a multi-physics model and so it automatically calculates the Joule heating occurring locally within the solenoid coil when electrical current is applied. The Joule heat generated is calculated from the equation:

$$Q' = \frac{Q}{V} = \frac{I^2 R}{V} = \frac{I^2 \rho L}{A^2 L} = \frac{I^2 \rho}{A^2} \tag{13}$$

where $Q$ is the heat generated, $Q'$ is the volumetric heat generated, $V$ is the volume, $A$ is the cross-sectional area of the wire, $L$ is the length of the wire, $I$ is the current and $\rho$ is the electrical resistivity. As expected, the total heat generated increases with the current squared (see Fig. 4). The heat generated can reach 1 W or more. The Joule heat generation is calculated over the entire surface of the solenoid. It is assumed in the current analysis that the current is distributed evenly through the cross-section of the solenoid coil. Although the heat generation is uniform, the temperature distribution will not be.

In addition, heat may be convected away from the solenoid valve through the air. This will be modeled as free convection. A more complete description of the convection modeling is given later. This is because convection is difficult to analytically model, and the current work will be empirically determined by comparison to the experimental measurements.

The electromagnetic, thermal and structural (that is, the directly coupled multi-physics) modeling of the SV under investigation in this work is being performed with the measured dimensions of the actual SV product, the known applied current density, and the expected material properties of the various parts. However, to improve computational time only a portion of the SV will be modeled in the FEM software (see Fig. 1). The resulting finite element mesh is shown in Fig. 3. The mesh was refined to satisfy mesh convergence.

Multi-physics modeling of a SV using Ansys™ gives insight into the temperature distribution and the location of the highest temperature in the SV. It will also make predictions for the mechanical and thermal deformations and stresses due to high temperatures in the SV, and thus the resulting mechanical stresses and deformations on parts of the SV.

Most of the necessary material properties are readily available. The properties used for the materials of the solenoid valve in the current study are shown in Table 1.

It is also important to apply realistic boundary conditions to the finite element model. Since the model considers many different fields (thermal, mechanical and electromagnetic), several different sets of boundary conditions must be applied (see Fig. 5). As shown, the deflections in the $y$-direction are held constant on the top surfaces. One point is used to keep the model from translating in the $x$-direction. Free convection is assumed on all the outer boundaries of the solid materials, except on the line of symmetry, where axisymmetric boundary conditions are assumed. Zero normal magnetic flux is also assumed on all the outer boundaries.

To model the convection of heat away from the surfaces of the solenoid valve, the finite element model uses *Newton's law of cooling* (the following equation) to predict the heat loss due to convection ($q$):

$$q = hA\Delta T \tag{14}$$

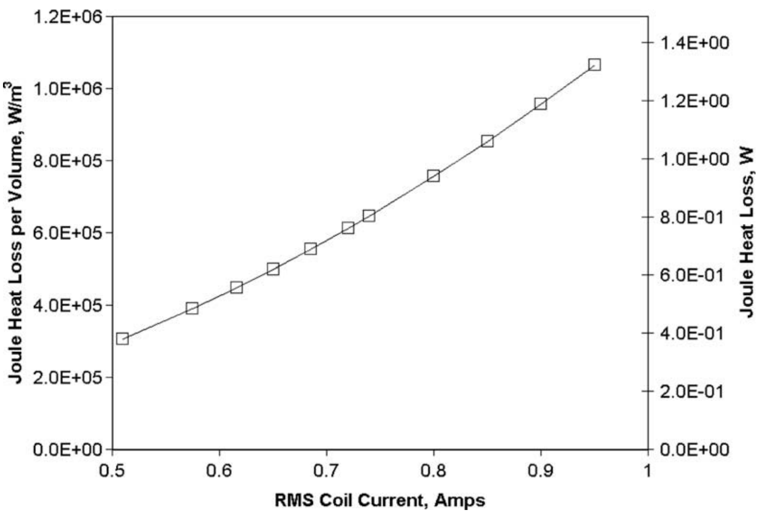


**Fig. 4.** The finite element prediction of the Joule heating within the solenoid coil.

**Table 1**
Material properties used in FEM model

| Property | Insulation polyamide-imide | Copper wire (coil) | Plunger iron powder | Core carbon steel | Air |
|---|---|---|---|---|---|
| $E$ (GPa) | 6.32 | 110 | 200 | 205 | 0 |
| $v$ | 0.38 | 0.343 | 0.29 | 0.29 | 0.5 |
| $k$ (W/m K) | 0.363 | 385 | 76.2 | 44.5 | 0.0313 |
| $\rho$ ($\Omega$m) | $1 \times 10^{15}$ | $1.7 \times 10^{-8}$ | $8.9 \times 10^{-7}$ | $2.49 \times 10^{-7}$ | $1 \times 10^{14}$ |
| $\alpha$ ($\mu$m/m K) | 25 | 16.4 | 12.2 | 11 | 0 |
| $\mu_r$ | 1 | 1 | 100,000 | 1500 | 1 |

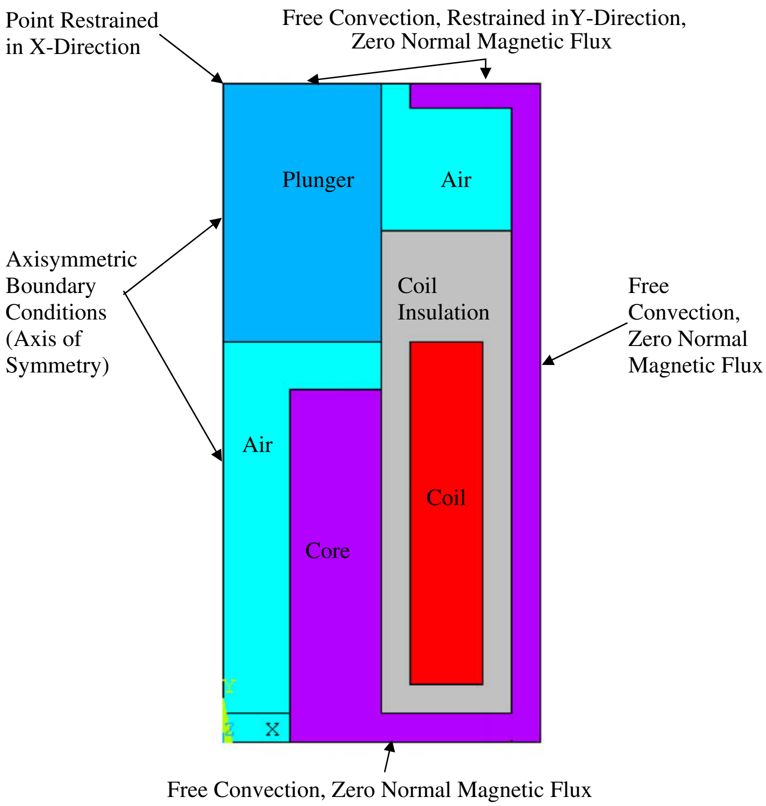


**Fig. 5.** Schematic of the FEM model of the solenoid valve, including boundary conditions.

where $A$ is the surface area, $\Delta T$ is the difference in temperature between the surface and the ambient air, and $h$ is known as the convection coefficient, or the film coefficient. It can be very difficult to analytically predict $h$ for a given geometry. Empirical correlations are therefore often used. For an infinitely long cylinder geometry, Churchill and Chu [12] related $h$ to the Nusselt number, $Nu$, by the equation

$$h = \frac{k \cdot Nu}{D} = \frac{k}{D}\left\{0.60 + \frac{0.387Ra^{1/6}}{\left[1 + \left(\frac{0.559}{Pr}\right)^{9/16}\right]^{8/27}}\right\}^2 \quad (15)$$

where $k$ is the thermal conductivity, $D$ is the diameter of the cylinder, $Ra$ is the Rayleigh number and $Pr$ is the Prandtl number. For this case of a long cylinder, the Rayleigh number is given by

$$Ra = \frac{g\beta(\Delta T)D^3}{\eta\delta} \quad (16)$$

where $g$ is the gravitational constant (9.81 m/s$^2$), $\beta$ is the volumetric thermal expansion coefficient, $\eta$ is the kinematic viscosity and $\delta$ is the thermal diffusivity. For air, some of these properties are given in Table 1, and the rest are given in Table 2. $Pr$ is also taken to be 0.697. For these properties, and diameter of the solenoid (28 mm), the value of $Ra$ calculates to be 78469. Then the predicted value for $h$ is 8.14 W/m$^2$ K.

**Table 2**
Material properties for air

| Property | Air |
|---|---|
| $\beta$ ($K^{-1}$) | $2.725 \times 10^{-3}$ |
| $\eta$ ($m^2/s$) | $22.8 \times 10^{-6}$ |
| $\delta$ ($m^2/s$) | $32.8 \times 10^{-6}$ |

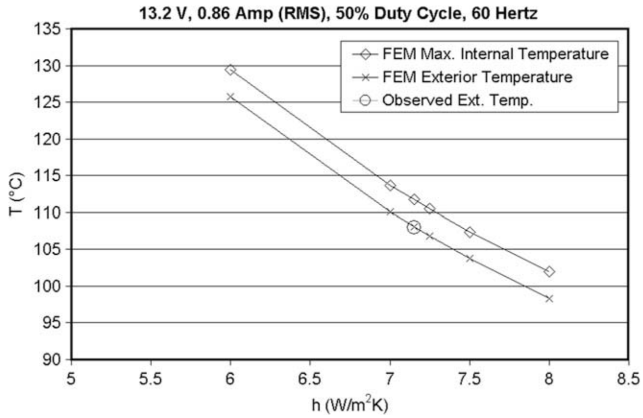


**Fig. 6.** Finite element prediction of the temperature for various convection coefficients.

In addition, a suitable value for $h$ can be found by fitting the external temperature of the solenoid valve predicted by the FEM model to that measured using the test apparatus (see Part 2 of this work). The external temperature extracted from the FEM model was from the same location that the thermocouple contacts the solenoid valve in the test rig (it is about 0.5 cm from the end of the solenoid casing). As shown in Fig. 6, the fit results in a value of 7.15 $W/m^2$ K, which is a very similar value that was calculated from the empirical prediction above (Eq. (15)). This fit was performed for the case of a solenoid valve running at 13.2 V, 0.86 A (RMS), 50% duty cycle and 60 Hz.

Using the fit value of $h = 7.15$ $W/m^2$ K, the predicted external temperature can be compared to a wide range of experimental results for various excitation voltages. The results are used to design and select the conditions used to test the solenoid valves.

## 3. Results and discussion

A thermal, mechanical and electrical analysis of the solenoid valve using the Ansys™ finite element package has been performed and the results are presented below. The FEM model solves the coupled mechanical, thermal and electromagnetic physics of the problem. The results shown below are for the same solenoid valves tested in the experimental portion of this work (see Part 2) and the physical properties used to model this case are given in Tables 1 and 2.

The multi-physics model of the solenoid valve can produce predictions for a wide range of quantities, such as temperature, strain, stress and Joule heat. Some of these results are presented below for conditions similar to those expected in the solenoid valve application and during testing. One will also find that they produce the expected trends that would result from simple approximations and predictions of solenoid behavior.

The finite element predicted temperature distribution of the solenoid valve coil is shown in Fig. 7. As expected, the highest temperatures are found within and around the coil. The plastic encapsulating the coil is a poor conductor and so it causes the heat to build up near the coil. As will be shown in Part 2 of this work, this build up of heat can cause the wire insulation to fail and the plastic to melt, causing failure of the solenoid valve. It can also be observed that the minimum temperatures are located near the top of the solenoid coil, as that is where much of the heat is conducted into the valve portion of the part and convected away. Since an averaged model is used to consider the solenoid coil, the local temperature distributions in each wire of the solenoid coil are not seen (individual wires are not modeled). Therefore, the local temperatures in the wire might be larger than the 'averaged' temperatures predicted by the finite element model and shown in Fig. 7. To examine this possibility, later in this section a localized model of the coil wires will also be presented.

The temperature distribution shown in Fig. 7 will of course cause thermal expansion in the materials. The expansion and strain will not match between the various materials and parts which will then cause stresses to form (see Fig. 8). These stresses could be substantial and could also influence the failure and reliability of the solenoid valve. It appears that the highest stresses occur within the casing of the solenoid. However, there are also significant compressive stresses within the solenoid

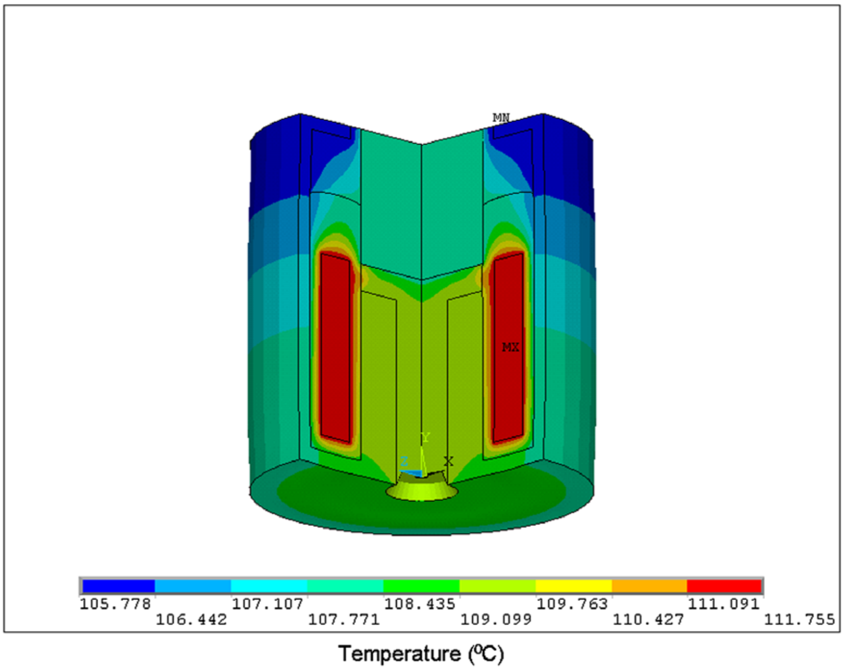


**Fig. 7.** The predicted temperature distribution in the solenoid cross-section for 0.86 A of current.

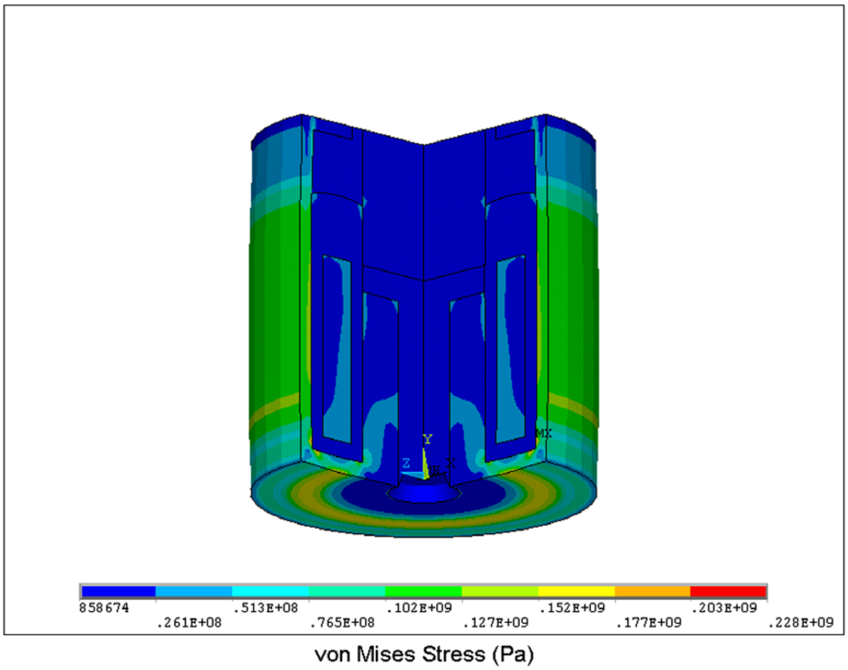


**Fig. 8.** The predicted von Mises stress distribution in the solenoid cross-section for 0.86 A of current.

coil that will cause the wire and insulation to press against each other. Combined with high temperatures, this could cause the insulation to squeeze out from between the wires and result in shorting and failure (as is shown experimentally in Part 2).

As mentioned previously, the heat generated and the temperature distributions will result in thermal deflections and stresses. The finite element predicted deflections in the cross-section are shown in Figs. 9 and 10. The deflections in the *x*-direction are maximum near the outer radius of the solenoid valve casing and appear to reach a value of almost

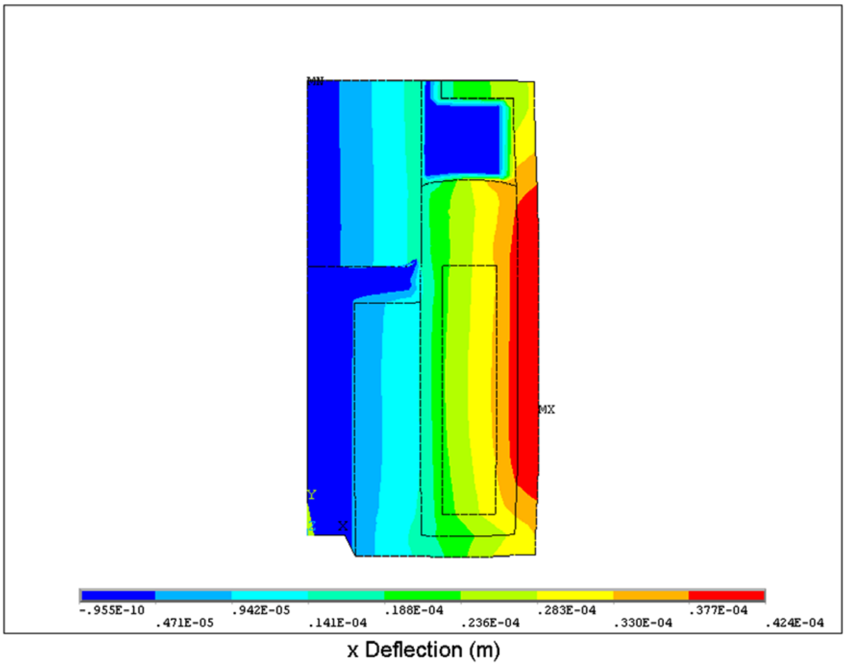


**Fig. 9.** The finite element model predictions of the deflections in the *x*-direction within the solenoid valve resulting from thermal expansion (for 1 A).

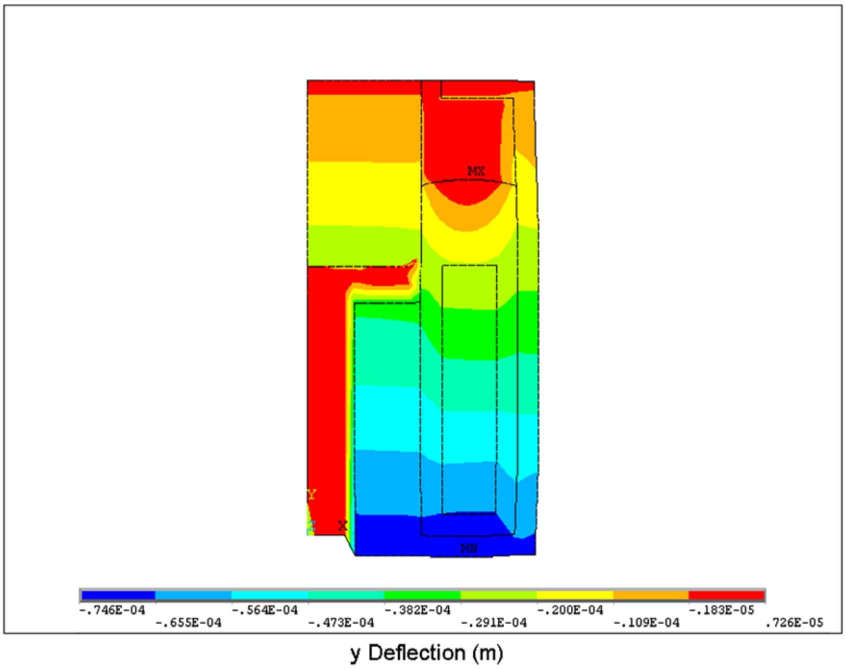


**Fig. 10.** The finite element model predictions of the deflections in the *y*-direction within the solenoid valve resulting from thermal expansion (for 1 A).

0.05 mm (see Fig. 9). The deflections in the *y*-direction are smaller than the deflections in the *x*-direction and also appear to maximize at different locations (see Fig. 10). The maximum deflections in the *y*-direction occur near the bottom of the solenoid valve. Of course, these deflections are relative since they depend on which points were fixed in the model. As discussed in Section 2 on the modeling methodology, the axis of symmetry restricts the deflection in the *x*-direction, while the cross-section is fixed in the *y*-direction along the top surface.

Preliminary tests of the solenoid valves were then run to verify the effectiveness of the finite element model in predicting the solenoid valve operation (see Part 2 of this work for details of the experimental setup). The solenoid valve was run at

60 Hz, under a constant duty cycle, a constant peak voltage, a constant peak current, and in ambient air. The tests were run until the solenoid valve reached a 'steady-state' temperature as measured by the external thermocouples (see Part 2 paper). For various values of applied current the 'steady-state' temperatures were recorded (see Fig. 11). Note that these temperatures are for the thermocouple touching the external surface of the solenoid valve and therefore to make a consistent comparison, the temperature from the finite element model located at nodes corresponding to the location of the thermocouple must be used. As shown in Fig. 11, the experimentally measured (observed) and finite element predicted temperatures agree very well. It can also be observed that the maximum temperature predicted by the finite element model is not significantly higher than the external temperatures. This suggests that the externally measured temperature is sufficient for evaluating solenoid valve performance. This will also be confirmed through a localized finite element model of the solenoid valve wires (see later section) and by measuring the temperature directly from the solenoid coil (see Part 2 paper).

Since the finite element model appears to be able to make good predictions of the solenoid valve performance, we can now use it to generate results over a wider range of operating conditions. As shown in Fig. 12, the maximum von Mises stress in the solenoid core and coil for various applied currents were recorded. The stresses in the core are significantly higher than the coil, but the steel material of the core is also much stronger. The yield strength of the copper wire making up most of the solenoid coil is approximately 200 MPa and the yield strength of the core steel is approximately 480 MPa. As the current is increased, the stress does increase, but for the conditions shown the stresses do not reach the yield strengths. It is therefore believed that the failure of the solenoid valves is not sourced only from thermally induced stresses. However, the stresses are high enough that in combination with high temperatures, they could cause damage to the wire insulation.

### 3.1. Local wire model

A finite element model of the local wires within the solenoid coil is also constructed. The purpose of this localized model is to determine if the local temperature of the wires is significantly higher than the average temperature predicted by the larger scale solenoid valve model. To create a local wire model the hexagonal symmetry of the wire coil structure is employed (See Fig. 13). This cross-section is then meshed as shown in Fig. 14. It is assumed that all the boundaries follow the laws of symmetry, so that they have zero normal deflections and zero heat conduction. However, the temperatures of the locations where the insulation contacts the edges are held constant. Then a current is applied to each of the wires and the temperature distribution and deflections of the cross-section are predicted.

The local solenoid wire model is then used to predict the local temperatures for the case of solenoid valve powered with a RMS current of 1 A and 80 °C ambient temperature, which as will be shown in the following section, results in a peak temperature of 210.8 °C. The resulting temperature distribution is shown in Fig. 15. The predicted rise in the local temperature is only 1.5 °C. Therefore, it appears that local temperature rise should not be an important effect when considering solenoid valve performance and reliability. However, as will be shown next, that does not appear to be the case for the predicted deflections.

The rise in temperature of the wire and insulation from room temperature will cause a significant amount of thermal expansion and deflection, as shown in Fig. 16. Although the wires appear to be overlapping due to the deflection shown in Fig. 16, they are not, it is just because the deflections are exaggerated for clarity. Nonetheless, the deflections are large (as much as 0.44 mm). These deflections should compress the relatively weak polyamide-imide insulation very easily, especially since the temperatures are also very high. This will result in the possible failure of the insulation and shorting between the solenoid coil wires. This mechanism is believed to be one of the failure mechanisms occurring in the accompanying tests of the solenoid valves (see Part 2 paper).

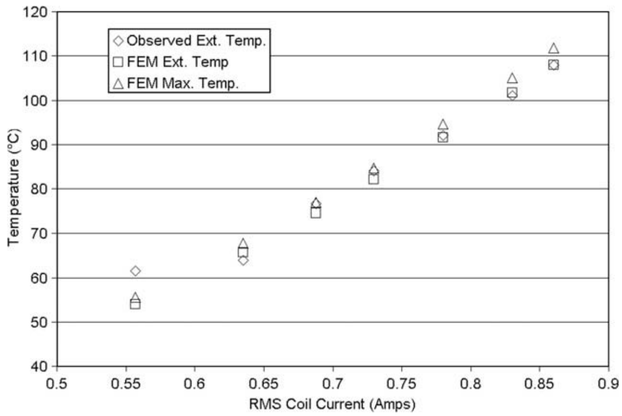


**Fig. 11.** Comparison of the theoretically predicted and experimentally measured solenoid temperature.

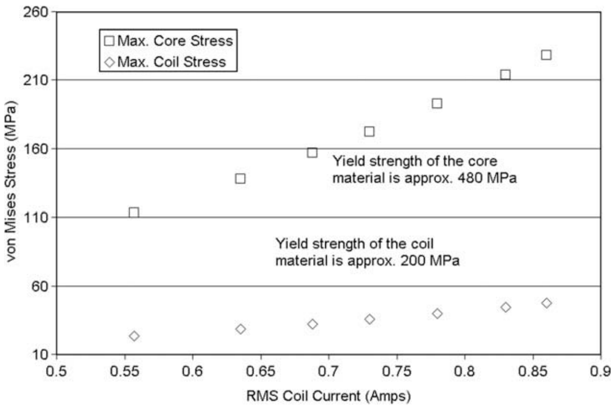


**Fig. 12.** The maximum stresses predicted by the finite element model.

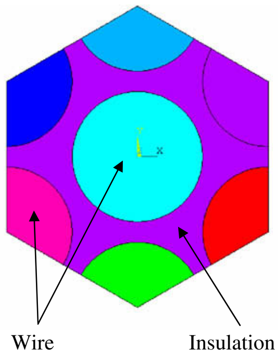


**Fig. 13.** Schematic of the hexagonally symmetric local solenoid wire model.

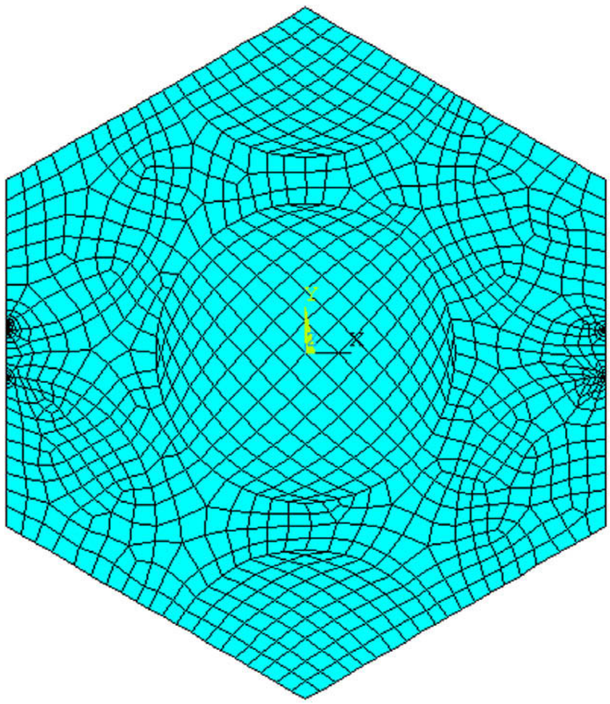

**Fig. 14.** The finite element mesh of the local solenoid wire cross-section.

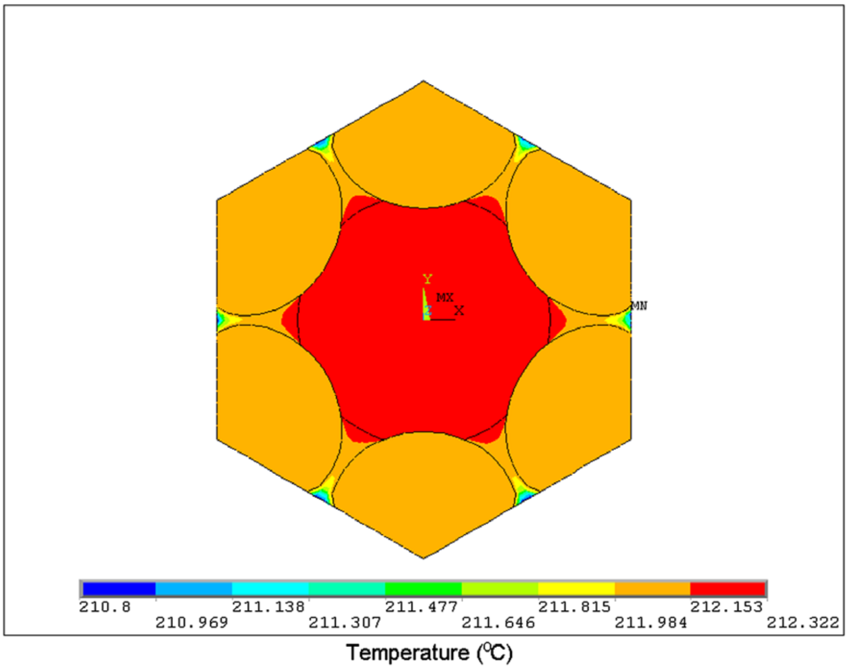


**Fig. 15.** Finite element predicted local solenoid coil wire temperature distribution.

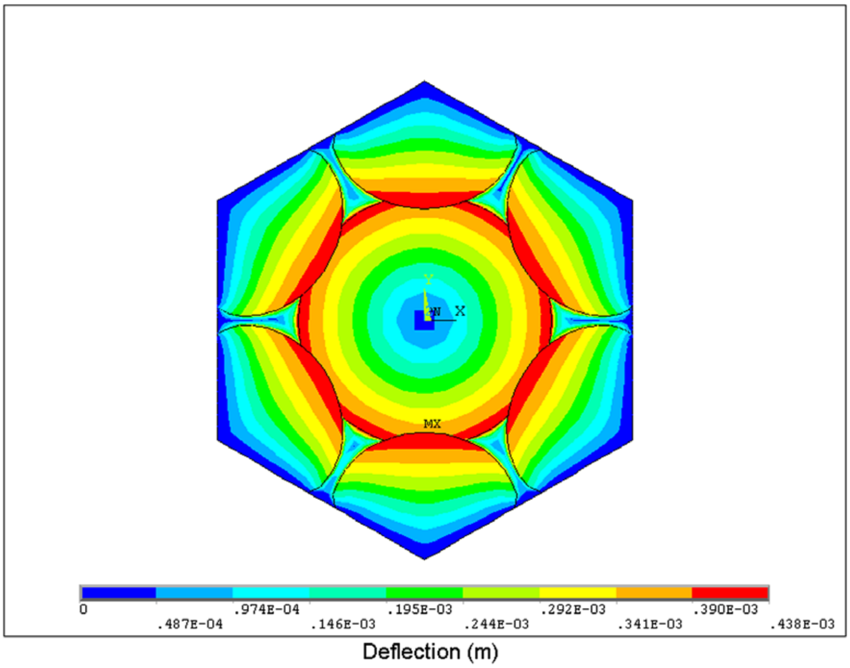


**Fig. 16.** The finite element model predictions of the local deflections within the solenoid coil resulting from thermal expansion (for 1 A).

### *3.2. Theoretical design of reliability test*

The finite element model can now be used to make theoretical predictions of solenoid valve operational parameters which affect reliability, such as temperature, stress and strain. These predictions are useful to then design the test loads and conditions which may cause failure in the solenoid valve. First, to increase the severity of the test and to model realistic temperatures seen in the transmission, the air temperature in the model is increased to 80 °C. It was observed that when the

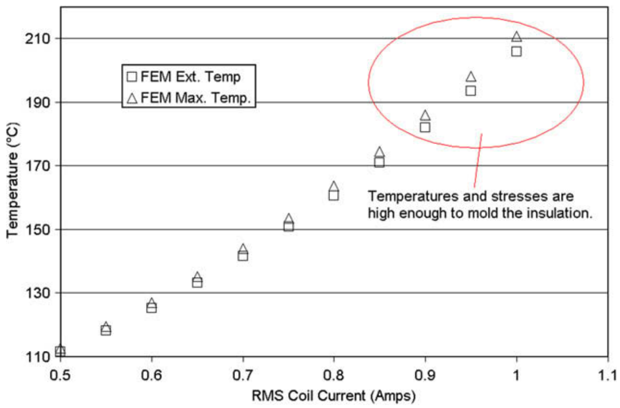


**Fig. 17.** The FEM predicted temperature in the coil and on the external surface of the solenoid valve.

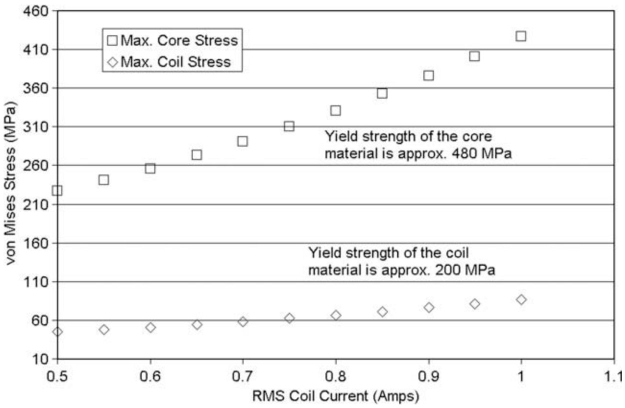


**Fig. 18.** The maximum stresses predicted by the finite element model.

solenoids were run at room temperature it was difficult to obtain failure. Changing the ambient temperature to 80 °C can be accommodated by making a simple change in the model boundary conditions.

The finite element model is then used to generate results over a wide range of applied currents (see Figs. 17 and 18). Based on the FEM predictions, a test procedure can be designed to bring the temperature of the solenoid valve coil near to the critical operating temperature of the polyamide-imide insulation (approximately 200 °C [13]) (see Part 2 paper).

The applied RMS current was varied between approximately 0.5 and 1 A and the external and maximum temperatures were recorded (see Fig. 17). Once again the temperature increase with current was expected. Also, the difference between the maximum temperature and the external temperature that would be measured by the thermocouple is very small. Based on these predictions, the temperatures in the coil can rise to values that are above what is recommended for the insulation. In addition, at these temperatures, it is expected that the stresses within the solenoid coil wire should be very high due to thermal expansion (see earlier section on the local wire model).

The stresses under these elevated conditions (80 °C ambient temperature) are predicted to be significantly higher than the room temperature case (see Figs. 12 and 18). However, it should be remembered that these predicted stresses are expected to be heavily influenced by the machining tolerances of the parts. Still, these high stresses, especially within the coil, are expected to influence and assist the failure of the solenoid valve. These stresses in combination with the elevated temperatures will essentially form and mold the insulation present in between the wires. Signs of these high compressive stresses have been seen in all the run solenoid valves, even the ones that did not fail (see Part 2). The wires in the solenoid coil appear to change from a round cross-section to a hexagonal shape due to the high compressive stresses.

The experimental test procedure used in the current analysis was devised based on the elevated temperatures and stresses seen at certain applied currents when the ambient temperature is raised. To further evaluate solenoid performance under these conditions the finite element predicted temperature and stress distribution for the most severe case modeled (1 A) are presented (see Figs. 19 and 20).

In Fig. 19, the predicted temperature distribution for 1 A and 80 °C ambient temperature is shown. Note that the temperature is only about 5 °C higher in the coil than on the outer surface where the thermocouples are located in the test apparatus (see Part 2 paper). The maximum temperature is located in the coil where the Joule heating is present. The plastic material

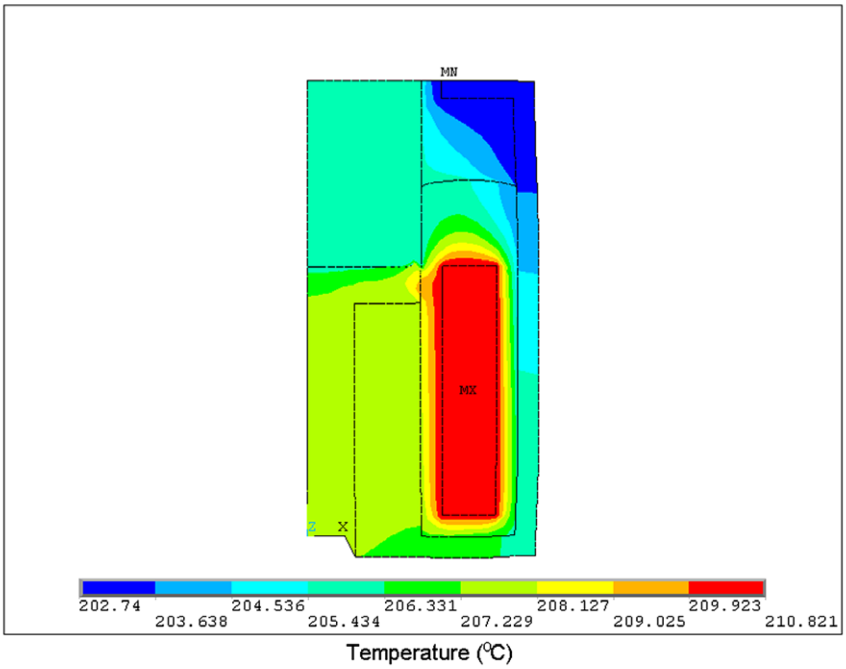


**Fig. 19.** The predicted temperature distribution in the solenoid cross-section for 1 A of current and 80 °C ambient temperature.

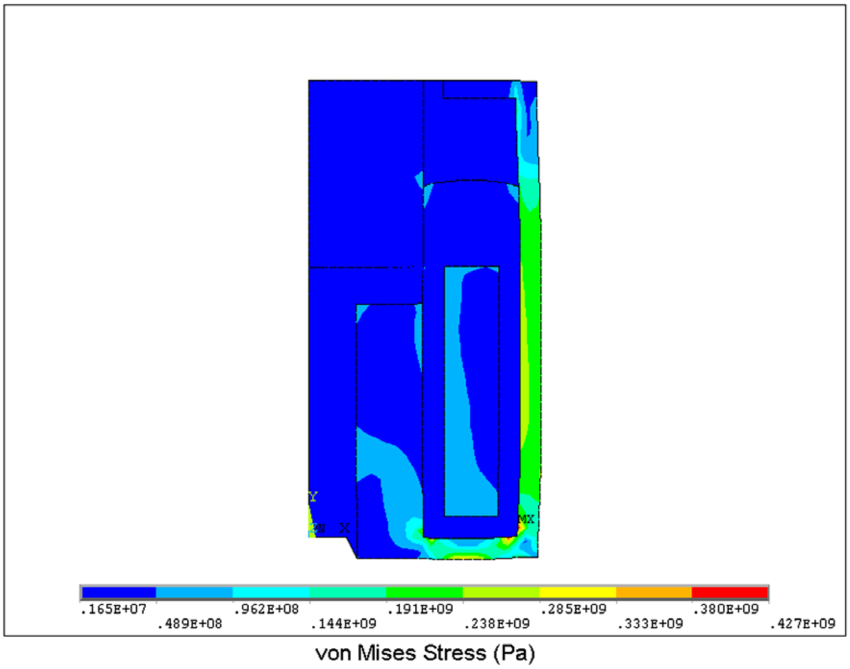


**Fig. 20.** The predicted von Mises stress distribution in the solenoid cross-section for 1 A of current and 80 °C ambient temperature.

around the coil is also heated considerably, which is also noticed in the experimental results. During an experiment, the plastic material surrounding the coil actually melts and exits the solenoid valve when failure occurs (see Part 2 paper).

The predicted von Mises stress distribution for this same extreme case is shown in Fig. 20. The stresses are greatest in the steel core and outer casing of the solenoid valve. However, the stresses within the coil are still elevated and could cause the insulation to fail under compression.

In summary, the finite element model results suggest that the solenoid valve will fail due to a thermal–mechanical failure of the insulation between the solenoid coil wires. It is expected that if a high enough ambient temperature and RMS coil current is applied then the temperatures and compressive stresses will be sufficient to soften or melt the insulation enough to move it from between the copper wires. The copper wires would then short out and cause the resistance in the coil to drop. This would result in the solenoid coil magnetic force being reduced and perhaps fail to actuate the valve. These theoretical predictions will be experimentally confirmed in Part 2 of this work.

## 4. Conclusions

A multi-physics finite element model of the solenoid valve is constructed that is able to make predictions of the stresses, strains and temperatures within the solenoid valve. The results suggest that under slightly elevated but realistic operating conditions, very high temperatures and stresses can occur. These high compressive stresses are caused by thermal expansion of the solenoid valve components. The compressive stresses can then in combination with the high temperatures degrade and mold the insulation between the solenoid coil wires. This will result in shorting between the coil wires that will decrease the electrical resistance of the solenoid valve and also cause eventual failure. The finite element model is also used to confirm that the temperature within the coil is only a few degrees different from the temperature on the external surface of the solenoid valve that is measured experimentally.

## Acknowledgements

This work was supported by the Center for Advanced Vehicle Electronics, a National Science Foundation Industry/University Cooperative Research Center.

## References

[1] Jeong H-S, Kim H-E. Experimental based analysis of the pressure control characteristics of an oil hydraulic three-way on/off solenoid valve controlled by PWM signal. J Dynam Syst Measur Control: Trans ASME 2002;124:196–205.
[2] Tseng C-Y, Lin C-F. Characterisation of solenoid valve failure for electronic diesel fuel injection system of commercial trucks. Int J Heavy Vehicle Syst 2006;13:180–93.
[3] Ferreira JA, De Almeida F Gomes, Quintas MR. Semi-empirical model for a hydraulic servo-solenoid valve. Proc Inst Mech Eng Pt I: J Syst Control Eng 2002;216:237–48.
[4] Schultz A. Design of valve solenoids using the method of finite elements. In: Presented at power transmission and motion control, Bath, United Kingdom; 2005.
[5] Sung DJT, Lee T-T. Model reference adaptive control of a solenoid valve controlled hydraulic system. Int J Syst Sci 1987;18:2065–91.
[6] Szente V, Vad J. Computational and experimental investigation on solenoid valve dynamics. In: Presented at IEEE/ASME international conference on advanced intelligent mechatronics, Como; 2001.
[7] Tao G, Chen HY, He ZB. Optimal design of the magnetic field of a high-speed response solenoid valve. J Mater Process Technol 2002;129:555–8.
[8] Wang S-M, Miyano T, Hubbard M. Electromagnetic field analysis and dynamic simulation of a two-valve solenoid actuator. IEEE Trans Magnet 1993;29:1741–6.
[9] Xu M, Tang X. Time constant and magnetic force of an electrohydraulic seat valve solenoid. In: Presented at FLUCOME, San Francisco, CA, USA; 1991.
[10] Li J-S, Hu W-H, Lin H-T, Li H-P. Simulation analysis on static performance of EP solenoid valves based on ANSYS. China Railway Sci 2005;26:72–5.
[11] Elmer K, Henthorn K, Skellern P. Self heating effects on the dynamics of a proportional control valve. Measur Control 1998;31:101–4.
[12] Churchill SW, Chu HHS. Correlating equations for laminar and turbulent free convection from a horizontal cylinder. Int J Heat Mass Transfer 1975;18:1049–53.
[13] Mark JE. Polymer data handbook. New York: Oxford University Press; 1999.