

Reliability and Life Study of Hydraulic Solenoid Valve - Part 1 - A Multi-physics Finite Element Model

S. V. Angadi, R. L. Jackson, Song-yul Choe, G. T. Flowers and J. C. Suhling
Auburn University

Young-Kwon Chang and Jung-Keol Ham
Korea Testing Laboratory

Copyright © 2009 SAE International

ABSTRACT

A comprehensive multi-physics theoretical model of the 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.

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 Fig. 1 and Fig. 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, commercial dishwashers, etc. 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[1].

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 experimental test set up, experimental methodology, and the accompanying results can be found in the Part 2 paper [1].

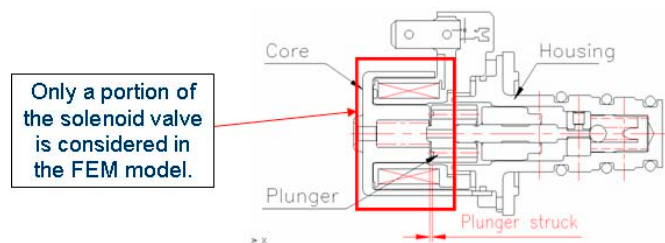


Fig. 1. Schematic of solenoid valve and portion considered in model.

Most theoretical models of solenoid valves are designed to consider their dynamic characteristics so that a control scheme can be designed and optimized [2-10]. Most of these previous works do not consider the coupled thermo-mechanical 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 [11, 12]. 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.

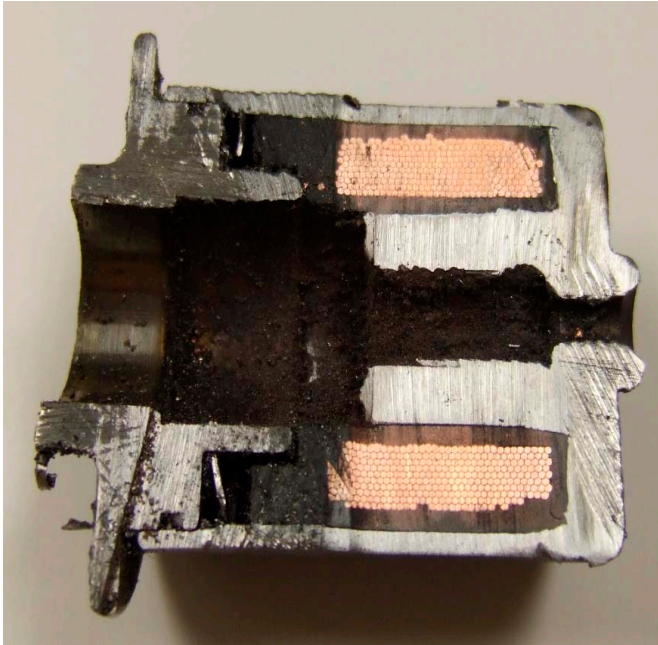


Fig. 2. Cross section of an unfailed solenoid valve.

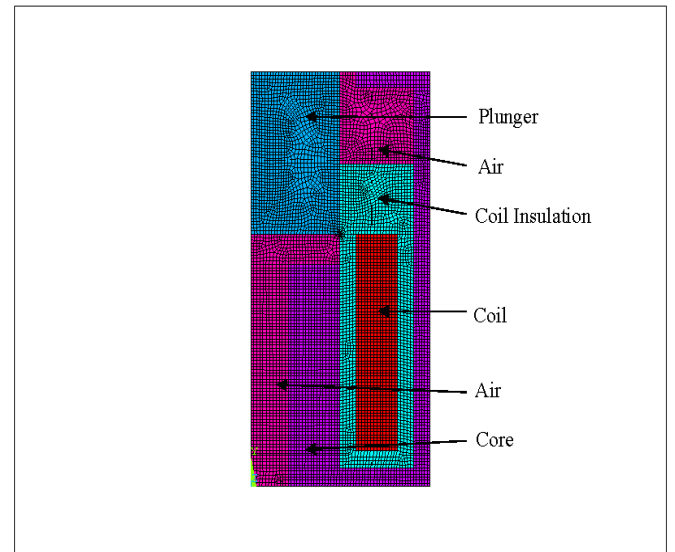


Fig. 3. A plot of the finite element mesh used to model the solenoid valve.

MODELING METHODOLOGY

A multiphysics (thermal, mechanical and electro-magnetic) model of the solenoid valve (SV) is developed in the finite element package AnsysTM (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, θ, z) as:

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

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

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

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

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

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

where E is the elastic modulus, ν is Poissons Ratio, σ is the normal stress, τ is the shear stress, ϵ is the normal strain, γ is the shear strain, α is the thermal expansion coefficient, and ΔT 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

$$\gamma_{r\theta} = \gamma_{\theta z} = \tau_{r\theta} = \tau_{\theta z} = u_{\theta} = 0 \quad (7)$$

$$\epsilon_r = \frac{\partial u_r}{\partial r} \quad (8)$$

$$\epsilon_{\theta} = \frac{u_r}{r} \quad (9)$$

$$\epsilon_z = \frac{\partial u_z}{\partial z} \quad (10)$$

$$\gamma_{rz} = \frac{\partial u_r}{\partial z} + \frac{\partial u_z}{\partial r} \quad (11)$$

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 θ is:

$$\frac{\partial^2 T}{\partial r^2} + \frac{\partial^2 T}{\partial z^2} + \frac{1}{r} \frac{\partial T}{\partial r} + \frac{1}{r^2} \frac{\partial^2 T}{\partial \theta^2} = \frac{1}{k} q'(r, \theta, z) \quad (12)$$

where $q'(r, \theta, z)$ is volumetric heat generation, k is the thermal conductivity, 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. In addition, the thermal contact resistances between the surfaces of the solenoid are neglected because they should be small in comparison to the bulk resistances. 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 multiphysics 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:

$$q' = \frac{Q}{V} = \frac{I^2 R}{V} = \frac{I^2 \rho L}{A^2 L} = \frac{I^2 \rho}{A^2} \quad (13)$$

where Q is the power of the 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 ρ is the electrical resistivity. As expected, the total heat generated increases with the current squared (see Fig. 4). The heat generated can reach 1 Watt 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.

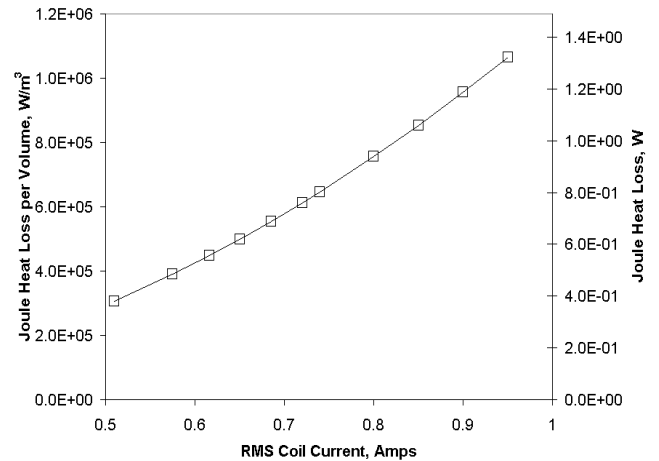


Fig. 4. The finite element prediction of the Joule heating within the solenoid coil.

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 multiphysics) 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.

Multiphysics modeling of a SV using AnsysTM 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. All of the symbols are defined earlier, except μ_r , which is the relative permeability. Not that Air has been assigned an elastic modulus of zero and a Poisson's ratio of 0.5 because it is a gas.

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
ν	0.38	0.343	0.29	0.29	0.5
k (W/mK)	0.363	385	76.2	44.5	0.0313
ρ (Ωm)	$1 \cdot 10^{15}$	$1.7 \cdot 10^{-8}$	$8.9 \cdot 10^{-7}$	$2.49 \cdot 10^{-7}$	$1 \cdot 10^{14}$
α ($\mu\text{m/mK}$)	25	16.4	12.2	11	0
μ_r	1	1	100000	1500	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.

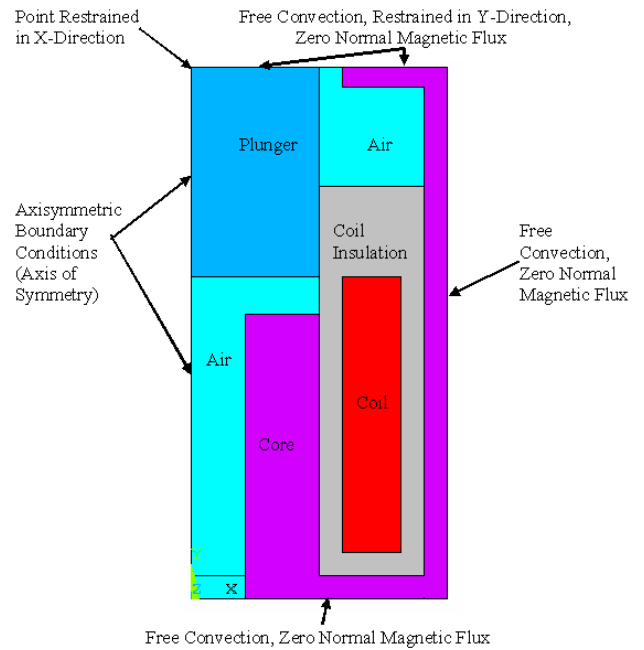


Fig. 5. Schematic of the FEM model of the Solenoid Valve, including Boundary Conditions.

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):

$$q = hA\Delta T \quad (14)$$

where A is the surface area, ΔT 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 [13] 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

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