Design of Pressure Measuring Cells Using the Unified Material Law

P. Aguirre[*1], F. Figueroa[*2]
[1,2]Sensor Technik Wiedemann GmbH
*Am Bärenwald 6 Kaufbeuren, Bavaria, Germany,
pablo.aguirre@sensor-technik.de, felipe.figueroa@sensor-technik.de

Abstract: Pressure measuring cells are modeled and optimized for better sensor sensitivity using the Solid Mechanics module in Comsol.
More realistic nonlinear effects in sensor signal and a better guess of the mechanical stability of the membrane as its burst pressure can be estimated by defining a realistic parametric nonlinear material model like the "unified material law" (UML).

Burst pressure is estimated here with an iterative algorithm that increases the applied pressure until the computation of stresses drops out and no convergence is possible anymore.

Keywords: Pressure measuring cell, nonlinear elastic/plastic material, UML, burst pressure.

1. Introduction

Pressure sensors (Figure 1) are widely used in the automotive industry. Their main use is the dynamic monitoring of pressure inside combustion engines [1]. Normally these kinds of sensor measuring cells are made of steel and have small sizes up to one centimeter (Figure 2).

Figure 1. Some STW pressure sensors

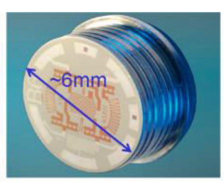

Figure 2. A STW measuring cell

To achieve good signal accuracy, the design of pressure sensors can be improved with FEM calculations of stresses and strains on the measuring cell depending on their geometry and material properties. The geometry is adapted according to a desired nominal pressure level and a limit rule of the stresses defined by the material properties (Figure 3a).

2. Model

A cylindrical measuring cell is modeled in 2D using the Solid Mechanics module in Comsol. The inner diameter (iD), inner contour radius (dR) and membrane thickness (t) are parameterized (Figure 4).

Empirical material properties (Figure 3a) are measured in a laboratory [5] and the relevant parameters like the ultimate tensile strength (Rm) and the Young module (E) as function of the temperature are necessary to parameterize it as an elastic/plastic material with the "unified material law" (UML) [2] [3] equation for steels. This strains based parametric equation (Figure 3b) has been directly (in the analytical form) implemented in Comsol.

Empiric material one-axial stresses measured in a laboratory

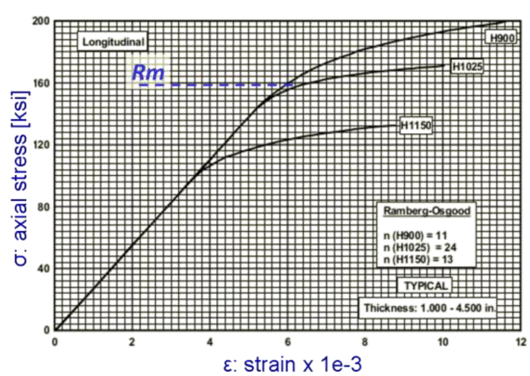

Figure 3a. Typical tensile stress-strain curves at room temperature for various heat treated conditions of 17-4PH stainless steel bar [5].

$$\varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{1,65Rm}\right)^{1/1,65}$$

Figure 3b. UML equation: where Rm is the ultimate tensile strength, ε the strain and σ the axial stress.

3. Use of COMSOL Multiphysics

The FEM problem is defined in Comsol with the stationary solver, where the governing equations for the elasticity study are:

$$-\nabla \cdot \sigma = F_V,$$
$$\sigma = J^{-1} F S F^T$$
$$F = (I + \nabla u)$$
$$J = \det(F)$$
$$S - S_0 = C : (\varepsilon - \varepsilon_0 - \varepsilon_{inel})$$
$$\varepsilon = \frac{1}{2}\left[(\nabla u)^T + \nabla u + (\nabla u)^T \nabla u\right]$$

The cell is fixed on the bottom; pressure loads on the inner surface and an axial symmetry on the membrane center are defined.
Important point in the Study Settings is to activate the option where Comsol includes geometric nonlinearities in the calculation.

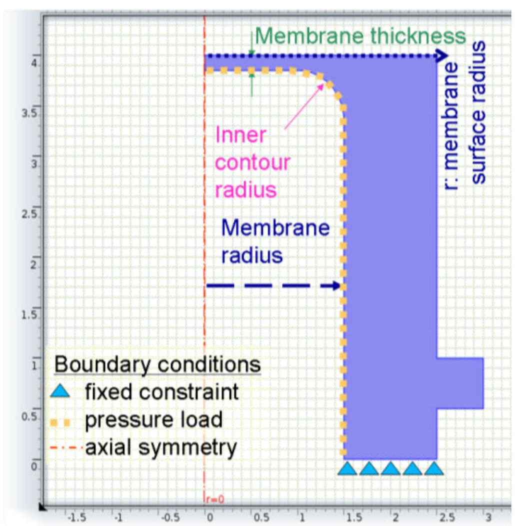

Figure 4. 2D cell model in cylindrical coordinates. Axial symmetry in the membrane center is defined.

After defining a constant maximal limit of allowed stress – normally multiplied by a safety factor based on a material fatigue norm [4] (Figures 3 & 7) – and varying the geometry parameters (Figure 4) in a Parametric Sweep Study, it is possible to optimize the difference between the maximal and minimal strains on the membrane surface (Figure 5).

4. Results

A cut line plot of the strains on the membrane surface (Figure 5) as a function of the distance to its center, shows the surface regions (Figure 6) which are compressed and expanded, allowing to determine the best places to allocate the strain-gauges for maximal signal sensitivity.

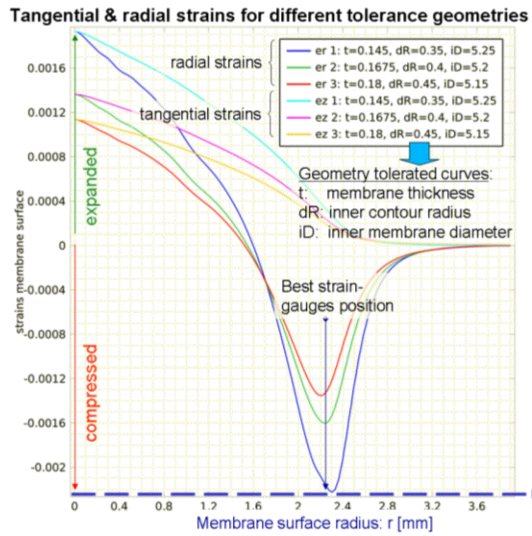

Figure 5. Calculated strains on the outer membrane surface along the radius cut line curve. The extreme values are found at r = 0 [mm] and r = 2.3 [mm]. Allocating the strain gauges in these places gives the maximal signal for that geometry.

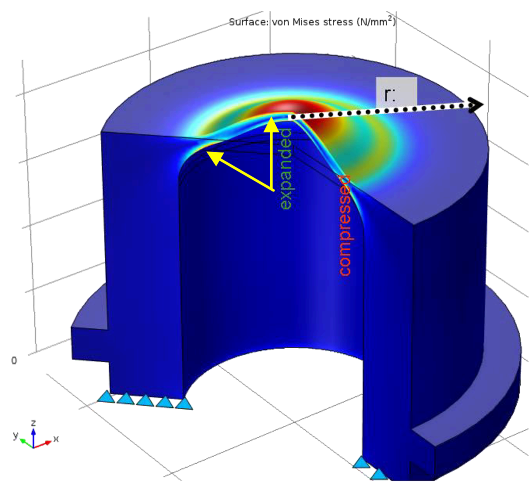

Figure 6. Von Mises stresses [MPa]. 3D FEM reconstruction cell solid. Red spots show where the maximal expanded zones in the membrane are situated.

Note that red spots are found on the membrane center outer surface and also on the inner cutter radius surface (Figure 6). A cut plane through these surfaces (Figure 7) shows that the maximal stress in the inner contour surface of the cell is more critical (bigger) than in the center of the outer membrane surface (Figure 8).

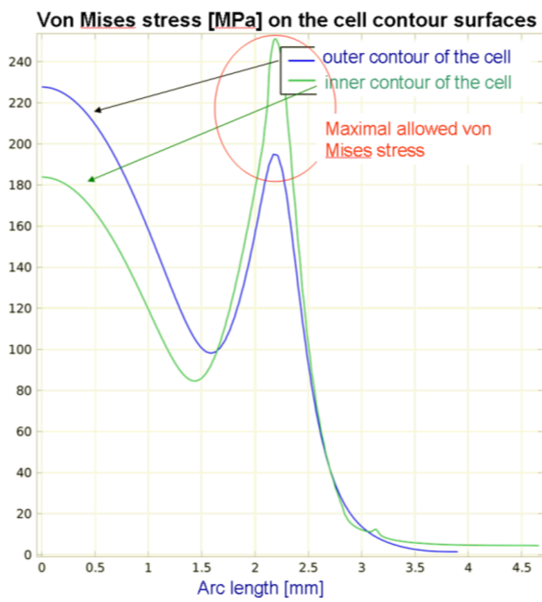

Figure 7. Calculated von Mises stresses along the cut plane on the inner and outer surface

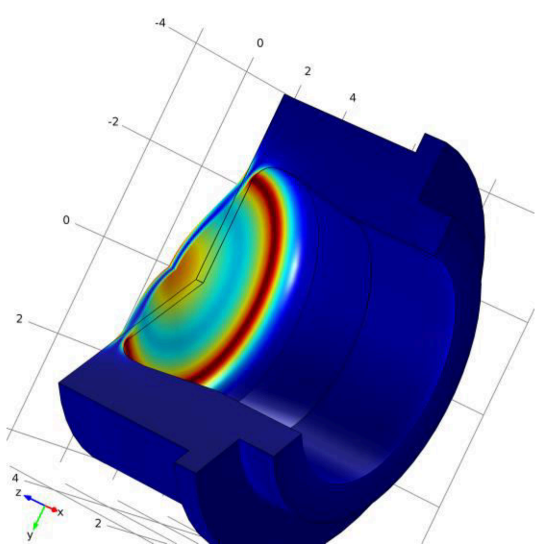

Figure 8. Von Mises stresses on the inner surface. Dark red ring shows the maximal von Mises stress.

5. Conclusions and extended results

Signals are improved by changing the defined geometry parameters according to a maximal stress design rule. The design with FEM tools allows us to analyze effects of tolerances in the geometry, affecting signal and mechanical stability of the sensor. Therefore, nonlinear effects (Figure 9) under mechanic tolerances have been also studied.

These nonlinear effects in signal are dependent on the applied pressure and can be calculated from the gained signal of strain-gages on the membrane surface by defined material thin-film properties. Statistical variables (mean, min. and max. values) are defined for each geometry parameter (t, dR and iD) in order to estimate the expected nonlinearity curves.

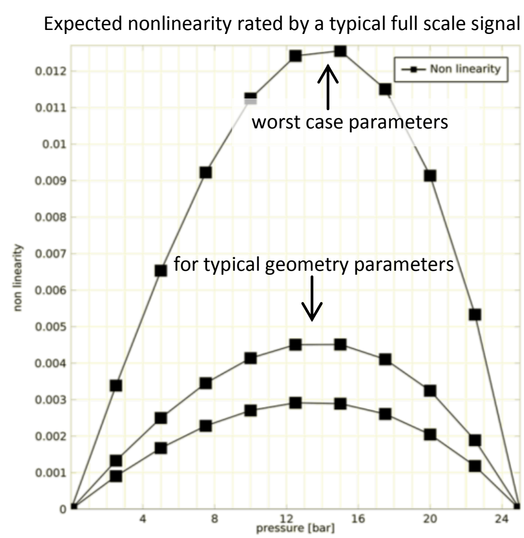

Figure 9. Nonlinearity estimation with statistical variation of the modelled geometry.

The burst pressure has been effectively estimated (Figure 10) with Comsol using UML [4] and a test-fail convergence algorithm, that stops when the equilibrium between loads and stresses by deformation cannot be compensated anymore.

Due to a defined limit of axial stress in the material model, the elastic/plastic relation results in a huge increment of material deformation by small increments of the applied pressure. Therefore red spots of stresses (Figures 11 & 12) became bigger and bigger.

The red spots have reached the maximal allowed stress of the material. The material is not elastic anymore and flows in plasticity.

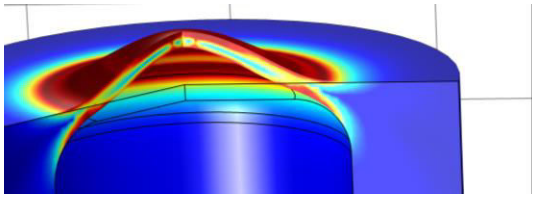

Figure 11. Red spots of stress became bigger and bigger by small increments of applied pressure.

This plasticity affects the whole red spots. Just one iteration before the computation stops (Figure 12), it is possible to guess where first the membrane busts.

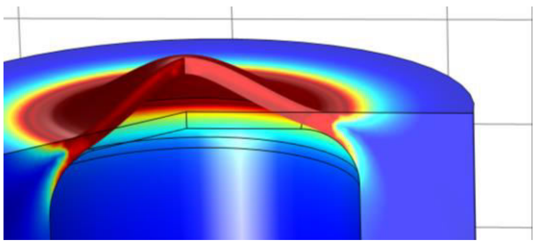

Figure 12. Just one iteration before computation drops out and equilibrium between inner material forces and applied pressure is not possible anymore.

Material flows in free plasticity and the deformation has no stop. Comsol Solver stops with singular matrices error. Note how the material of the membrane is displaced (Figure 13 and 14) that is because the equilibrium between inner material forces and applied force is not there anymore.

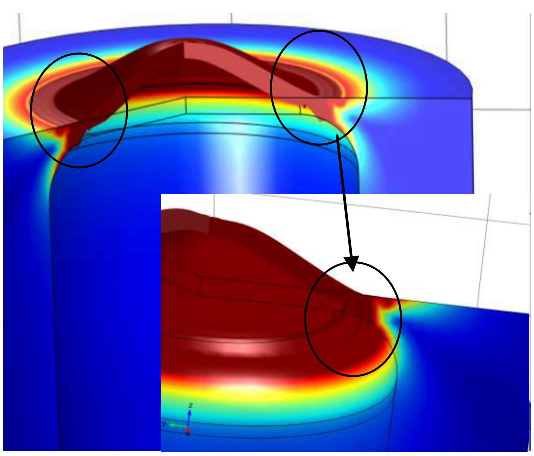

Figure 13. Stress plot by the iteration when the computation drops (3D)

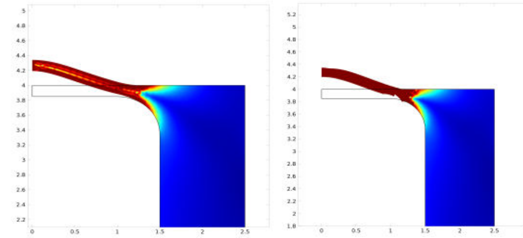

Figure 14. Stress results by iterations of Figures 12 and 13 when the computation drops (2D view)

The iteration algorithm can be better represented by a deformation load diagram (Fig. 15).

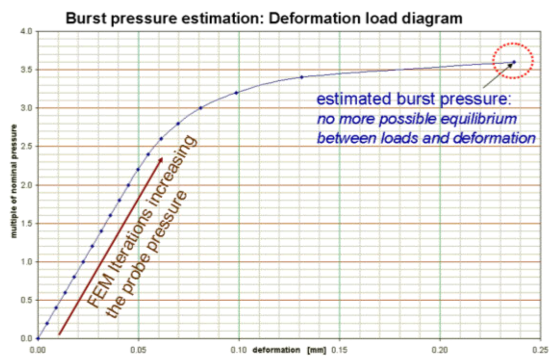

Figure 15. Burst pressure estimation using iterative convergence procedure until a FEM calculation loses the equilibrium between membrane deformation and stresses.

8. References

[1]: Mohapatra G.,
Design and Implementation of Diaphragm Type Pressure Sensor in a Direct Tire Pressure Monitoring System (TPMS) for Automotive Safety Applications, International Journal of Engineering Science and Technology (IJEST), Vol. 3 No. 8 August 2011

[2]: Korkmaz, S.
Master's Thesis, Bauhaus University Graduate School of Structural Engineering, 2008

[3]: Bäumel, A., Seeger, T.,
"Materials Data for Cyclic Loading", Supplement 1, Elsevier. Amsterdam, 1990

[4]: Smith, N. K., Watson, P., Topper T.H.
A Stress-Strain Function for the Fatigue of Metals,, Journal of Materials, Vol. 5, No 4, 1970

[5]: Department of Defense U.S.
Handbook "Metallic Materials and Elements for Aerospace Vehicle structures". MIL-HDBK-5J Page 2-208, January 2003