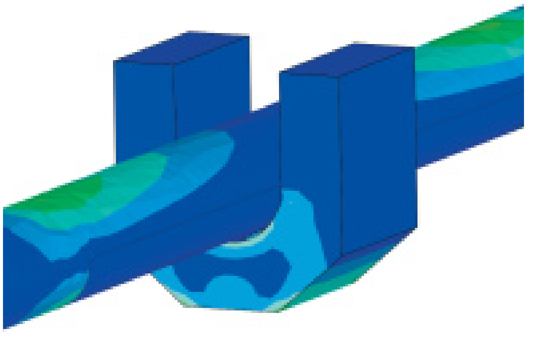
Multiphysics Makes Spinal Surgery Safer

Piezoresistive sensors designed with the help of coupled-field analysis give physicians real-time feedback on applied loads during corrective spinal surgery.

By David Benfield and Walied Moussa, Department of Mechanical Engineering, University of Alberta, Edmonton, Canada
Edmond Lou, Glenrose Rehabilitation Hospital, Edmonton, Canada

Scoliosis is a spinal deformity characterized by abnormal lateral curvature of the spine and axial rotation of the vertebrae. In severe cases, surgery may be required to correct this curvature. In this procedure, special connectors that either hook onto or screw into the vertebrae are inserted along the length of the spine where the curvature needs to be corrected. Surgeons then fit a metal rod into notches in the heads of these hooks and screws to realign the spine until to achieve the required correction.

This alignment process subjects the spine and the hook/screw assemblies to two types of 3-D loads: direct contact forces of the structural elements as they impact one another during the process and



FEA was used to determine contact forces on sensor strips in the surgical device.

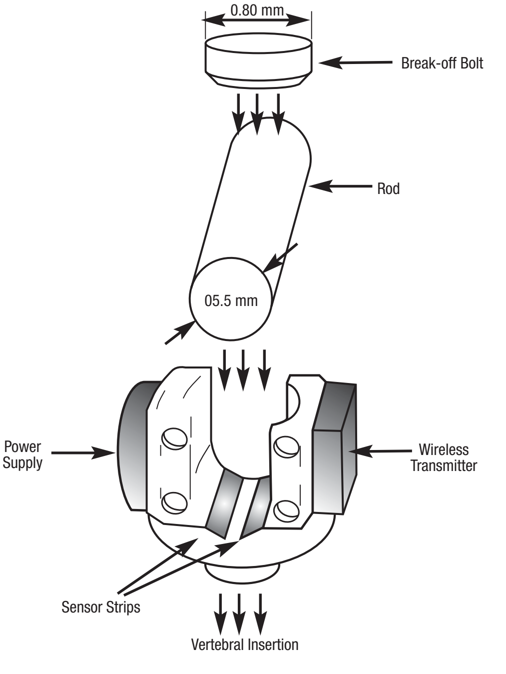


Diagram of hook/screw heads typically used in corrective scoliosis surgery. As seen here, they were modified with two piezoresistive silicon strips, a wireless transmitter and a power supply in order to sense and transmit loading information during surgery.

moment loads created by the leverage of the rods as they are installed. These loads are necessary in realigning the spine, but excess levels can contribute to bone breakage, as well as fatigue failure of the hooks, screws and rods.

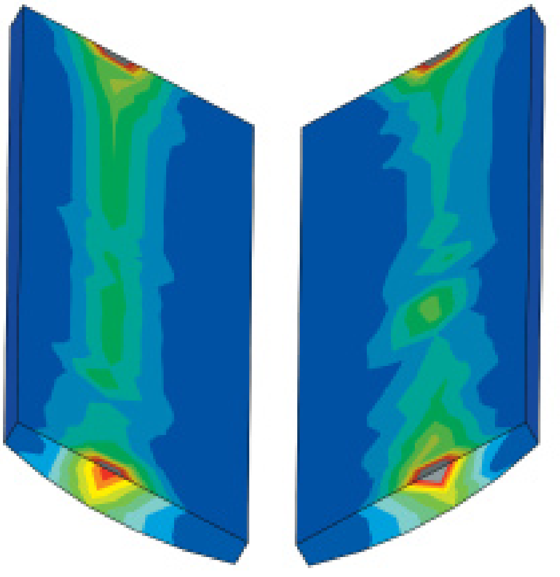
One recent research project at the University of Alberta in Canada is aimed at developing embedded micro-electrical-mechanical systems (MEMS) sensors to measure these forces and moments in real-time during surgery, thus providing physicians valuable feedback during the corrective procedure. For this application, modifications to conventional scoliosis hooks and screws were made. In addition, a power module and a wireless transmitter module were placed on either side of each of the hook/screw connectors, and two silicon sensor strips were placed at the interface between the rod and the hook/screw heads.

To detect 3-D forces and moments, each sensor strip must detect loads at more than one location along its length, so each strip has two sensor pads evenly spaced along the contact line of the corrective rod. Each pad consists of a deformable membrane onto which are mounted four piezoresistive gauges that are sensitive to contact forces in shear and normal directions. When combined, electrical output from the gauges on both strips indicate the 3-D forces and moments applied during surgery.

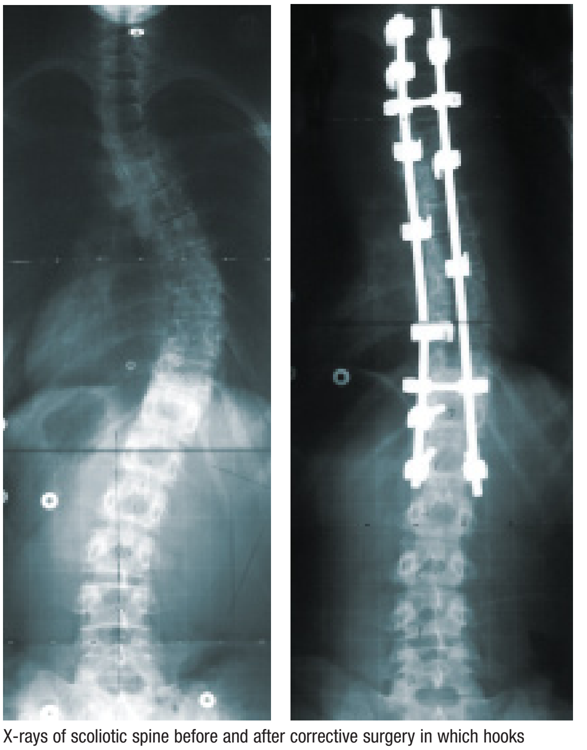
Multiphysics analysis using ANSYS Mechanical software was used to study three aspects of this configuration:

- Contact analysis used to predict loads transmitted between the rod and the sensor strips
- Structural analysis used to determine subsequent deformations of the anisotropic silicon membranes
- Piezoresistive analysis used to determine the output voltages from each piezoresistive gauge

ANSYS Mechanical technology provides coupled-field elements and multiphysics solution tools that enable all three regimes of this problem to be solved simultaneously in a single multiphysics solution. Using models that incorporate the element types and sizes for MEMS devices (see sidebar pages 8 and 9), output voltages from the sensor array were accurately predicted. Empirical data from other studies [7, 8] was used to determine a range of forces and moments that would be applied to each hook or screw.

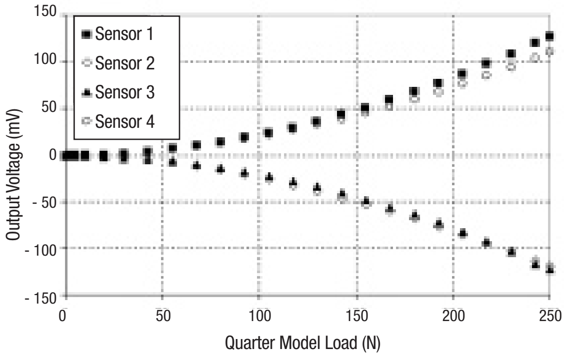


Contact stresses induced on sensor strips in a full hook trial

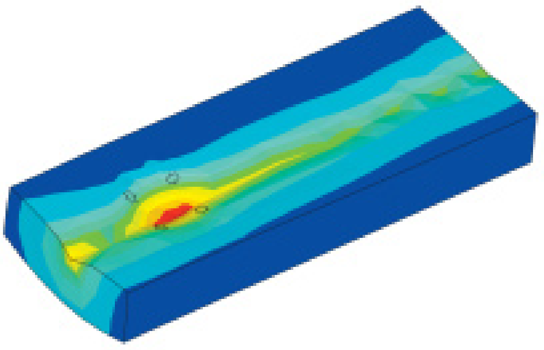


X-rays of scoliotic spine before and after corrective surgery in which hooks and screws are inserted into the vertebrae

Images courtesy Glenrose Rehabilitation Hospital.



Simulated sensor voltage outputs for the normal loading scenario



Numerical results for membrane deformation (μm units)

A full hook model with a rod contacting the sensor strips was built to determine the force distributions on the sensor strips for different load conditions. Using these force distributions, a symmetrical quarter-strip model was created to determine the membrane deformations and subsequent voltage outputs from the piezoresistive gauges. An excitation voltage of 3 volts was used to represent the power output characteristics of the power module.

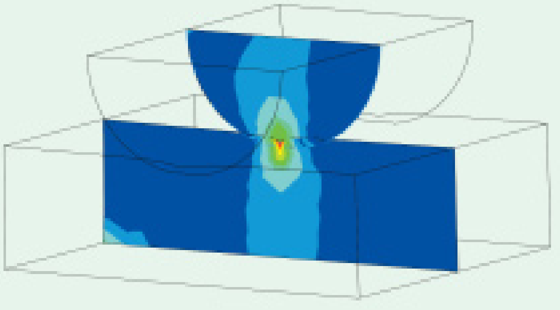
The flexibility of the FEA software allowed the voltage outputs for all 16 piezoresistive gauges to be accurately determined for the applied forces and moments. Voltage data from this analysis allowed calibration equations for the full sensor array to be developed utilizing simple superposition principles. These calibration equations enabled the 16 voltage outputs from each hook/screw assembly to be converted back into force or moment information representing loads that surgeons apply to the spine.

The numerical model is a powerful tool in creating these calibration equations, as it allows the problem to be broken down into symmetrical portions that are more easily analyzed than a full model. Moreover, ANSYS FEA simulation also can accurately predict sensor outputs,

Selecting and Sizing Finite Elements

ANSYS Mechanical software was utilized in each of three types of analyses and to establish appropriate parameters for the multiphysics simulations. This included using theoretical solutions to typical micro-electrical-mechanical system (MEMS) problems or simplified versions of the strip model to select the best element types and sizes for obtaining an accurate, fast solution. Element types and sizes were determined as follows:

Contact analysis of loads. The contact problem was the first to be evaluated for appropriate finite element (FE) parameters. A simple model of a cylindrical surface contacting a flat surface was developed, and the results of this model were compared to theoretical values obtained using standard Smith–Liu equations for determining contact stress distributions [1, 2]. At this point in the analysis, material properties are assumed to be linear for the flat silicon surface and for the stainless steel rod. The results of this analysis determined that when using a contact pair model (composed of ANSYS CONTA174 and TARGE170 elements) applied to 3-D solid elements, the mesh size in the contact area should be finer than one-half the theoretical contact width. In this analysis, the theoretical contact width is approximately 400 μm, so the optimal mesh size was determined to be smaller than 200 μm in order to obtain accurate analysis results.



Contact stresses produced in a trial analysis using a simple model of a cylindrical surface contacting a flat surface

Structural analysis of membrane deformation. Next, parameters were determined for accurately predicting deformations of the anisotropic silicon membrane produced by the contact forces calculated in the previous problem phase. For this analysis it was found that deflections predicted using ANSYS SOLID187 thus avoiding complicated testing of sensor components and related packaging.

Through preliminary manufacturing trials, it has been found that the manufactured devices have similar performance curves to the simulated models. Predicting the performance of a complex sensor array using multiphysics analysis without ever building a device is a significant advantage in terms of costs and time during MEMS product development. ■

References

[1] Shigley, J. E.; Mischke, C. M.; Budynas, R. G., “Contact Theory,” *Mechanical Engineering Design*, 2004, 7 Ed., McGraw-Hill: Toronto, pp. 161-166.

[2] Smith, J. O.; Liu, C. K., “Stresses Due to Tangential and Normal Loads on an Elastic Solid with Application to Some Contact Stress Problems,” *Journal of Applied Mechanics*, 1963, Vol. 71, pp. 157-166.

[3] Timoshenko, S.; Woinowsky-Krieger, S., *Theory of Plates and Shells*, 1959, McGraw-Hill: Toronto.

[4] Bao, M.; Wang, Y., “Analysis and Design of a Four-Terminal Silicon Pressure Sensor at the Centre of a Diaphragm,” *Sensors and Actuators*, 1987, Vol. 12, pp. 49-56.

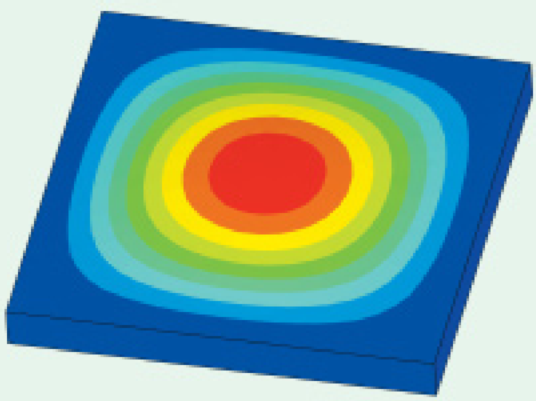
[5] Kanda, Y., “Optimum Design Considerations for Silicon Pressure Sensors using a Four-Terminal Gauge,” *Sensors and Actuators*, 1983, Vol. 4, pp. 199-206.

[6] Bao, M.; Qi, W.; Wang, Y., “Geometric Design Rules of Four-Terminal Gauge for Pressure Sensors,” *Sensors and Actuators*, 1989, Vol. 18, pp. 149-156.

[7] Lou, E.; Hill, D. L.; Raso, V. J.; Moreau, M. J.; Mahood, J. K., “Instrumented Rod Rotator System for Spinal Surgery,” *Medical & Biological Engineering & Computing*, 2002, Vol. 40, pp. 376-379.

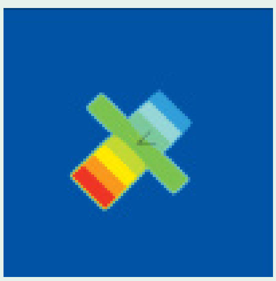
[8] Duke, K. K., “The Design of Instrumentation for Force Measurement During Scoliosis Surgery,” *Mechanical Engineering*, 2001, University of Alberta, Edmonton, pp. 114.

elements and a mesh size finer than 2.5 percent of membrane edge length were in agreement with calculations for a typical square-membrane problem, as defined by Timoshenko [3]. For the 1 mm x 1 mm membranes used in this analysis, this corresponds to a maximum mesh size of approximately 25 μm. Fortunately, solutions produced by ANSYS Mechanical software are not subject to the thin-membrane limitation of theoretical solutions, which may not yield accurate results when membrane thickness is larger than 5 percent of membrane edge length.



Deflection of the 1 mm x 1 mm silicon membrane is predicted by structural analysis.

Piezoresistive analysis to determine output voltage. FE parameters for the piezoresistive part of the analysis were next determined for the four-terminal gauge subjected to uniaxial stress. The theoretical solution to this problem is outlined [4–6]. Conclusions from this literature agree with piezoresistive theory: that p-type silicon devices have maximum sensitivity when uniaxial stress is applied perpendicular to the device surface with the long axis of the four-terminal gauge at a 45-degree angle to this direction. The required mesh size for a 100 μm x 50 μm four-terminal sensor modeled with SOLID227 elements was also evaluated, leading to a determination that a 10 μm mesh is required to numerically simulate voltage outputs that are consistent with piezoresistive theory.



Voltage distribution of a simple four-terminal sensor under uniaxial stress