Finite Element Simulation and Parameter Optimization of a Flexible Tactile Pressure Sensor Array

by

Shira M. Lee

Submitted to the Department of Mechanical Engineering in Partial Fulfillment of the Requirements for the Degree of

Bachelor of Science
at the
Massachusetts Institute of Technology

June 2005

Signature of Author: ______________________
Department of Mechanical Engineering
May 6, 2005

Certified by: ______________________
Mandayam Srinivasan
Senior Research Scientist
Thesis Supervisor

Accepted by: ______________________
Ernest Cravalho
Chairman, Undergraduate Thesis Committee

Finite Element Simulation and Parameter Optimization of a Flexible Tactile Pressure Sensor Array

by

Shira M. Lee

Submitted to the Department of Mechanical Engineering on May 6, 2005 in Partial Fulfillment of the Requirements for the Degree of Master of Science in Mechanical Engineering

ABSTRACT

A finite element model was developed to optimize design of a flexible tactile sensor. The sensor consists of layers of thin-film copper and PDMS, and the model can be used to determine the effects on sensor sensitivity and durability of variations in material properties and geometry. The model was used to study the effect of variations in copper thickness. Four copper thicknesses, 0.3µm, 0.5µm, 3µm, and 9µm, were analyzed under a range of pressure loads.

The thickness of the copper affected both the stress in the material and the displacement of the copper when a pressure load was applied to the sensor model. The stress in the sensor was highest for 3µm copper, potentially causing decreased durability in this sensor. The separation between the copper strips beneath the pressure load was highest for 9µm copper, so this sensor may have lower accuracy for small loads. Thin copper strips are challenging to manufacture, so the largest but most accurate and durable copper strip thickness, 0.5µm, is recommended from this analysis.

Thesis Supervisor: Mandayam Srinivasan
Title: Senior Research Scientist

Acknowledgements

I am grateful to Dr. Gang Liu for his guidance, teaching, and assistance throughout this project. I also thank Mandayam Srinivasan for supporting the project and Prof. Ian Hunter and Dr. Andrew Taberner for generously allowing me to use their ANSYS software.

Table of Contents

1.0 Introduction and Project Background

Sensors that mimic the sense of touch by providing feedback about forces on their surfaces have an extensive range of applications. If the sensor is thin, flexible and robust, it can be used as a coating for robotic hands so robots can take feedback on the objects they touch. Such sensors can also cover artificial organs used to train surgeons, so the output from the sensors can indicate whether the surgeons should apply more or less force to the organs. One of these tactile sensors is being developed in the MIT Laboratory for Human and Machine Haptics. This sensor is designed to be thin, flexible and robust and to produce an electric signal that indicates the location and magnitude of any forces on the surface.

The sensor is composed of thin-film copper and a silicone-based polymer called polydimethylsiloxane (PDMS). The thin-film copper is deposited in two layers using an evaporation method in a vacuum chamber. There are two different compositions of PDMS used in the sensor, and the copper strips are deposited with polymer A between them and polymer B along the outside (Figure 1). Each copper layer is not continuous, but is instead composed of rows of uniformly spaced parallel strips. The copper strips in one copper layer are laid perpendicular to the copper strips in the second layer, so that each copper strip crosses every copper strip of the opposite layer, at a perpendicular intersection (Figure 2). When a pressure is applied to the surface of the sensor, the copper strips at all nearby intersections (nodes) are brought closer to each other, and the capacitance of the sensor changes. A voltage is applied to one end of each copper strip, and the opposite end is connected to a computer-based signal processing circuit that processes variations in the sensor's electrical signal to indicate the magnitude and location of surface forces.

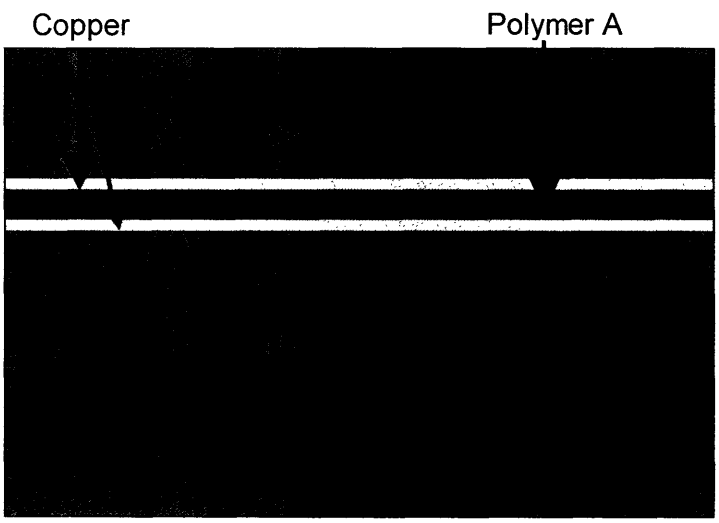

Figure 1: The sensor has five layers, an inner layer of polymer A, two copper strips, and outer layers of polymer B.

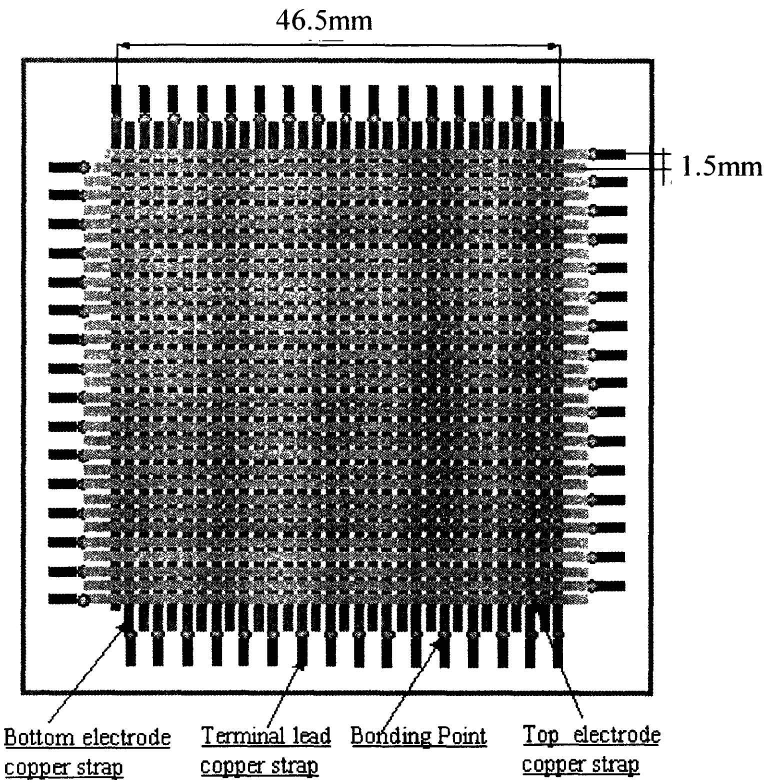

Figure 2: Sensor array with two layers of copper strips. The strips in each layer are parallel but the two layers are perpendicular. The sensor has 32X32 sensor elements and is about 0.3μm thick. *Image courtesy of Dr. Gang Liu.*

To perform as accurately as possible, the sensor must be sensitive to variations in pressure near the range 0MPa to 0.1MPa. The sensor is designed to be used to mimic human skin, and this range of pressures is the range to which human skin is most sensitive.[1] Both the geometry and material properties of the sensor materials can be adjusted to improve the sensor performance. The geometry of the thin-film copper affects its structural and electromagnetic properties. The width and thickness of the copper strips can be varied to adjust the stiffness of the sensor and the sensitivity of its electrical output. The PDMS is formed by blending a base and a catalyst, and varying the ratio of base to catalyst changes the material properties of the polymer. The base-to-catalyst ratio can be varied separately for polymer A and polymer B to adjust their stiffness. The stiffness of these individual polymers and the thickness of each polymer layer can be adjusted, along with the copper geometry, to produce a sensor with desired sensitivity to deformation.

1.1 Use of Finite Element Analysis for the Sensor

Finite element analysis (FEA) is a powerful tool for investigating the effects of variations in physical parameters on the behavior of a structure. The analysis is most useful when parameter variations are difficult, expensive, or too numerous to test efficiently by experimentation. FEA involves building a software model of the geometry and loading of a

[1] Data provided by Dr. Gang Liu

structure. The structure is then divided into finite elements (meshed), and the state of any mechanical, electrical, or thermal parameters of interest are determined across each element. If the elements are small enough, the finite solution is an accurate but efficient method of approximating the solution for an infinite number of points across the structure.

Optimization of sensor design involves variation of numerous parameters, including the geometry of each layer and the properties of each material, under a range of expected loads. Optimization through experimentation alone would be extremely time-consuming, as numerous combinations of small variations in geometry, material type, and load would have to be carefully manufactured and then tested. Finite element analysis can be used to test many combinations quickly, by varying the parameters in a model and comparing the analysis results. Once an optimized set of parameters is determined with FEA, these parameters can be tested experimentally to confirm the accuracy of the model optimization and to make slight adjustments in the sensor.

A variety of finite element analysis software packages are commercially available. All analysis for this project was done using ANSYS 9.0 University Research Edition.

1.2 Goal for Current Analysis

The copper strips in the sensor are thin-film copper several hundred nanometers thick. Changes in the thickness of the copper affect the stiffness of the strips and the overall stiffness of the sensor. The stiffness of the sensor changes the deformation between copper strips resulting from pressure loads. The deformation, in turn, determines the electrical output of the capacitive sensor. Thicker copper strips may increase the stiffness of the sensor and decrease its sensitivity, but sensors with thick strips are much easier to fabricate. Ease of fabrication must be balanced with sensitivity, and finite element analysis can be used to determine the range of copper thicknesses that can produce the desired sensor sensitivity.

Finite element models of a segment of the sensor with four different thicknesses of copper were built and meshed. The models were analyzed under a range of pressure loads to determine their sensitivity. The deformation in each model was analyzed quantitatively to determine which copper thicknesses would produce distinct deformations according to the magnitude of the load applied. The capacitance of the sensor is directly related to the separation between the copper strips, so the separation was determined as a function of pressure magnitude for each copper thickness. The thickness of the copper also affects the stress in the copper strips. If the axial stress reaches the yield stress of the material, the durability of the sensor decreases. The axial stress along the copper strips was determined as a function of pressure magnitude and copper strip thickness.

The models built for analysis of copper strip thickness can be easily modified to test the effect of changes in material properties of any of the three materials by varying the material property data and re-solving the model. The models can also be modified to test the effect of changes in geometric parameters besides copper thickness, such as copper width and polymer thickness. The geometric and material parameters can be varied individually, to isolate the effect of one parameter, or can be varied in combination to optimize the complete sensor design. The models built for this analysis form a foundation for extensive future analysis.

1.3 Sensor Structure and Parameters

All geometric and material properties except the thickness of the copper were set according to sensor prototypes currently being developed at MIT. The model is a two-dimensional cross-section of the sensor. The cross-section is cut along one copper strip in the bottom layer so that it bisects the perpendicular strips in the top layer. The cross-section was selected to be long enough to bisect three top strips, so the model includes three copper nodes and the space between each node.

The dimensions of the model are shown in Figures 3 and 4. Each copper strip is 1mm long, and the space between each strip is 0.5mm. The bottom layer of PDMS, polymer B, is 200µm thick. The bottom layer of copper strips is deposited on top of this outer polymer, and then a layer of polymer A 20µm thick is laid on top of the copper. The top layer of copper strips is then laid on top of this middle polymer layer, and a top layer of polymer B 100µm thick is used to cover the top of the copper. Four thicknesses (th_{CU}) of the copper strips were tested: 0.3µm, 0.5µm, 3µm, 9µm.

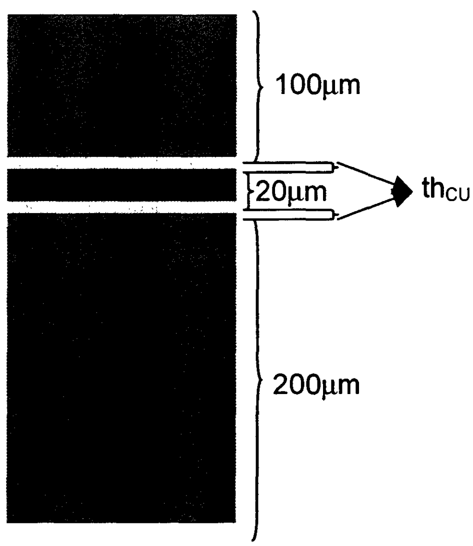

Figure 3: Vertical dimensions of the sensor model. The thickness of the copper layers is varied in each model between 0.3µm and 9µm.

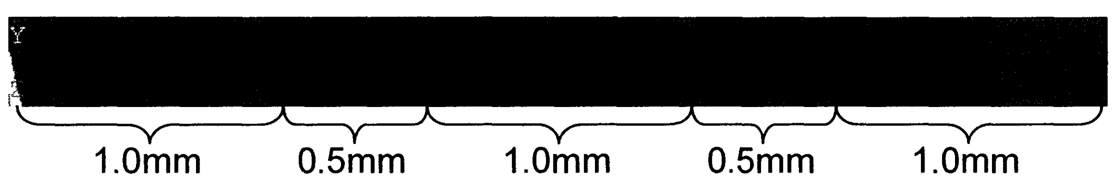

Figure 4: Horizontal dimensions of the sensor model. Each copper node is 1.0mm wide and the space between each node is 0.5mm.

1.3.1 Material Properties

The material properties of each material in the sensor are listed in Table 1. The dimensions are all in the μMKSv (microMeter-Kilogram-Second-Volt) system of units, a system developed to be used in nano-scale electromechanical devices. The conversion from MKS units to μMKSv is included in Appendix A.

Table 1: Material Properties used in the sensor model[1]

Mechanical Parameter	**μMKSv Unit and Value**
Young's Modulus	MPa=kg/(μm)(s)^2
Copper	1.10E+05
PDMS, middle	0.0682
PDMS, outer	0.2029
Poisson's ratio	(constant)
Copper	0.343
PDMS	0.499
Thermal Parameter	
Conductivity	pW/(μm)K=(kg)(μm)/(K)(s)^3
Copper	3.85E+08
PDMS	1.50E+05
Specific Heat	pJ/(kg)K=(μm)^2/(K)(s)^2
Copper	3.85E+14
PDMS	1.46E+15
Electrical Parameter	
Conductivity	pS/μm=(pA)^2(s)^3/(kg)(μm)^3
Copper	58823529.41
PDMS	2.50E-14
Resistivity	TΩμm=(kg)(μm)^3/(pA)^2 (s)^3
Copper	0.017
PDMS	4.00E+19
Relative Permittivity	(constant)
Copper	9.99E+59
PDMS	2.75
Permittivity of free space	pF/μm
	8.85E-06
Permeability of free space	TH/μm

[1] Amrani, Liu & Aluru. "Re-configurable Fluid Circuits by PDMS Elastomer Micromachining;" 12th International Conference on MEMS, MEMS 99, pp.222-227, Orlando, FL, 1998.
"Copper, Cu; Annealed," MatWeb.com, The Online Materials Database; Automation Creations, 1996.
CRC Handbook of Chemistry and Physics, CRC Press, 2004.
Polymer Data Handbook, Oxford University Press, 1999.

		1.26E-24

The stiffness of each polymer was determined experimentally from polymers used in sensor prototypes. Experiments were conducted to measure increments in stress and strain for uniaxial compression and then release of the compression for each polymer sample. The stress-strain curves are all convex up, but there is some hysteresis between the curves generated in push and release. The polymers were approximated as linear materials to simplify the model, and the slope of the linear approximation of the push curve was used as the estimate of Young's Modulus. The stress-strain curves for each polymer are shown in Figures 5 and 6. Both polymers are from the Dow Coring Corporation; the middle polymer is brand name HS IV, and the outer polymer is brand name Sylgard 184.

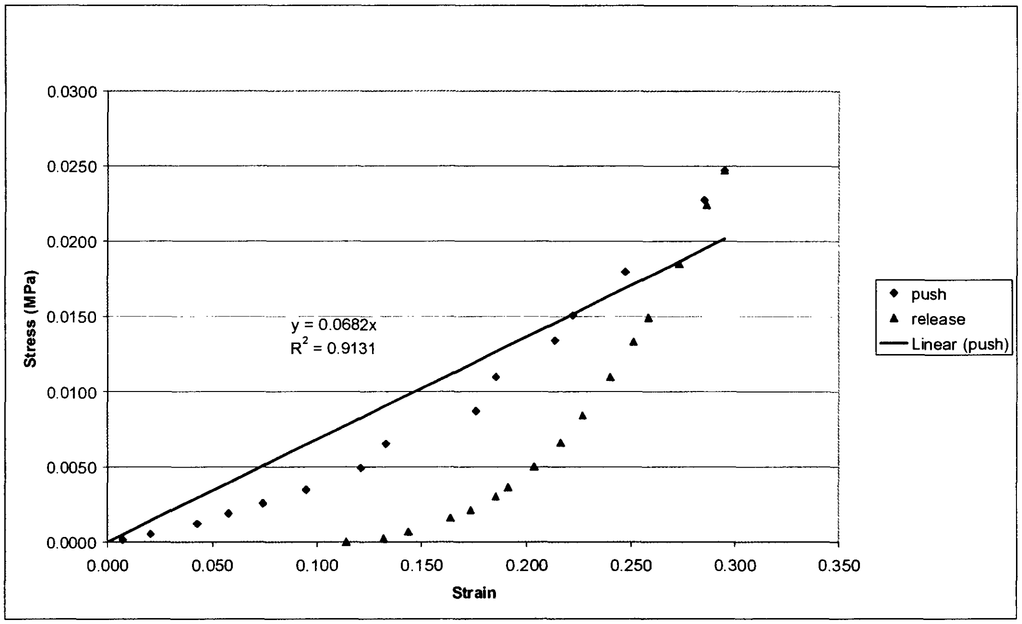

Figure 5: Experimental uniaxial compression data for the middle layer of PDMS (polymer A). A linear approximation of the push data was used to determine a value of 0.0682MPa for Young's Modulus of polymer A. *Data supplied by Dr. Liu.*

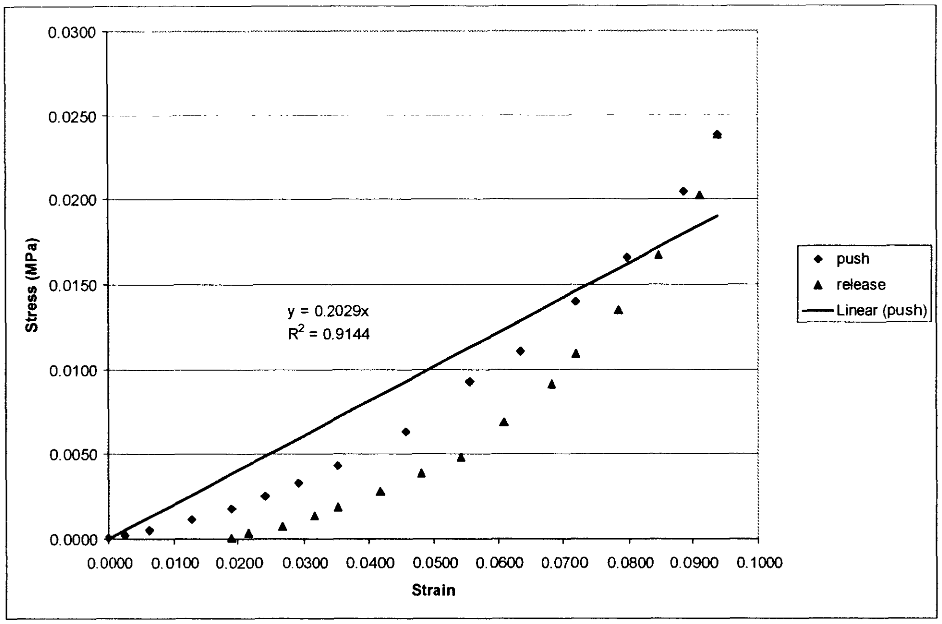

Figure 6: Experimental uniaxial compression data for the outer layers of PDMS (polymer B). A linear approximation of the push data was used to determine a value of 0.2029MPa for Young's Modulus of polymer B. *Data supplied by Dr. Liu.*

For all structural analysis, the bottom line of the sensor was fixed for all degrees of freedom, because in practice the sensor is fixed along the bottom surface. A uniformly distributed pressure load was applied along the top line of the top polymer layer over the center node of copper, to simulate distributed pressures on the top surface of the sensor.

The mechanical parameters of greatest interest were the vertical (y) component of displacement, especially the minimum distance between the copper strips, and the horizontal (x) component of stress. The distance between the copper strips determines the capacitance of the sensor, so the vertical displacement of each strip is directly related to the electrical output of the sensor. The stress in the copper strips affects their durability, especially their likelihood to fail or fatigue during use.

2.0 Finite Element Analysis

Finite element modeling consists of four primary steps. First, a solid model of the geometry is constructed. For the sensor, a two-dimensional cross-section was drawn across three nodes (intersections of the upper and lower copper strips). The dimensions of this solid model were set according to Figures 3 and 4. Four models were built, one for each thickness of copper.

The computing power and time required for FEA increases with the number of elements in a model. An effective method of minimizing the number of elements is to cut the model along any plane of symmetry, apply symmetry boundary conditions to all planes of symmetry, and analyze the smallest unique portion of the model. The sensor model is symmetric about a vertical line down the middle of the center node, so only the left half of the sensor model was meshed. A symmetry boundary condition was enforced along the middle vertical line of the center node.

The second step in building a finite element model is to divide the model into finite elements. This is called meshing. The type and shape of element affects the accuracy and complexity of the solution. For the sensor model, planar, rectangular elements with eight nodes were chosen. The division of each area into elements was controlled manually to produce the desired number of elements in each layer of the model. To analyze variation in parameters across the copper and the middle layer of polymer, for example, there had to be multiple elements across the thickness of each of these thin layers. The horizontal divisions for these thin layers were also controlled, to produce elements with moderate aspect ratios. Elements with moderate aspect ratios produce more accurate solutions than long thin elements. At the same time, the number of elements needed to be minimized to allow the computer to process the analysis. The mesh of the sensor model is shown in Figures 7-10.

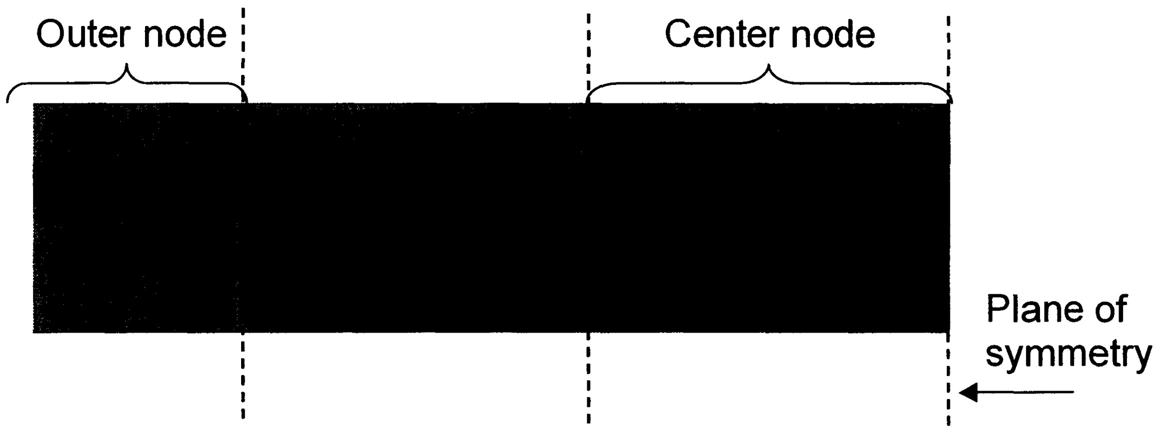

Figure 7: Mesh of a segment of the sensor, with copper 0.5μm thick.

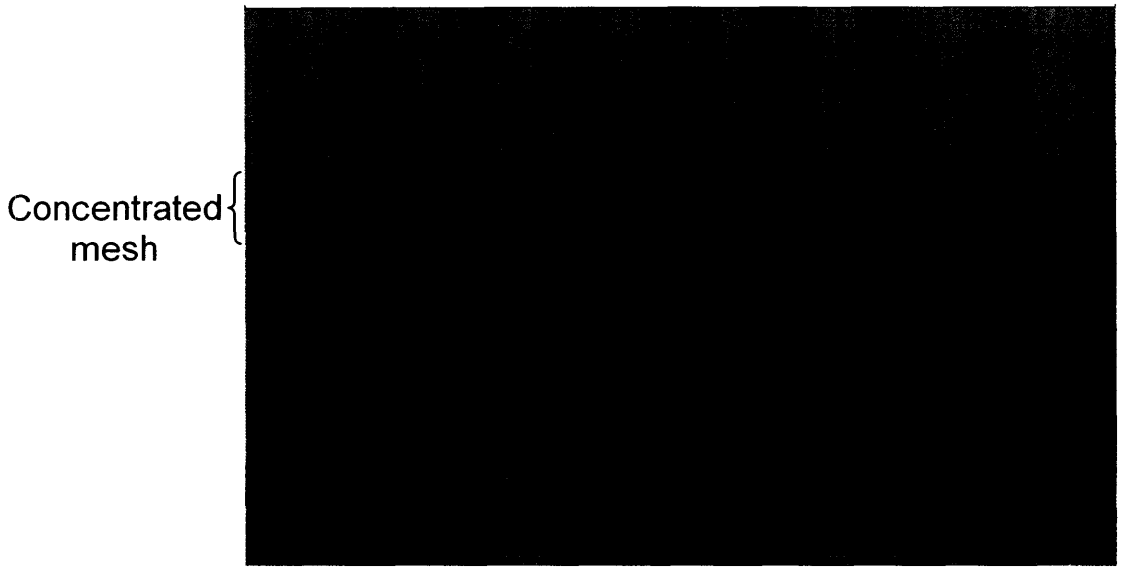

Figure 8: Mesh of the center copper node with 0.5μm thick copper; the elements closest to the copper and middle polymer layer are smallest, to accurately model stress and displacement distributions near these thin areas.

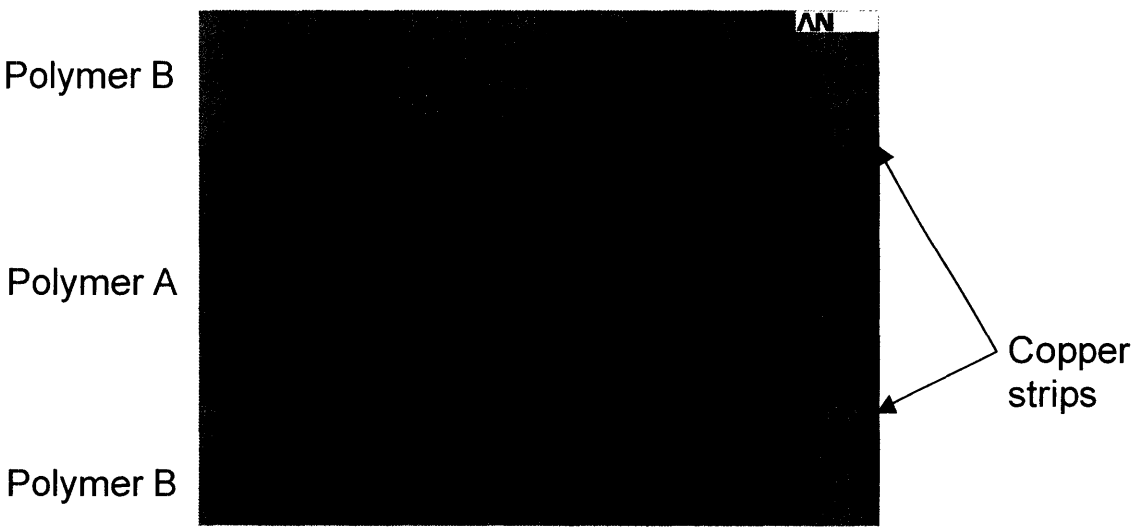

Figure 9: Mesh of the sensor with 0.5μm copper strips. The magnification shows the middle layer of polymer with the thin copper strips above and below it and the outer polymer surrounding the copper.

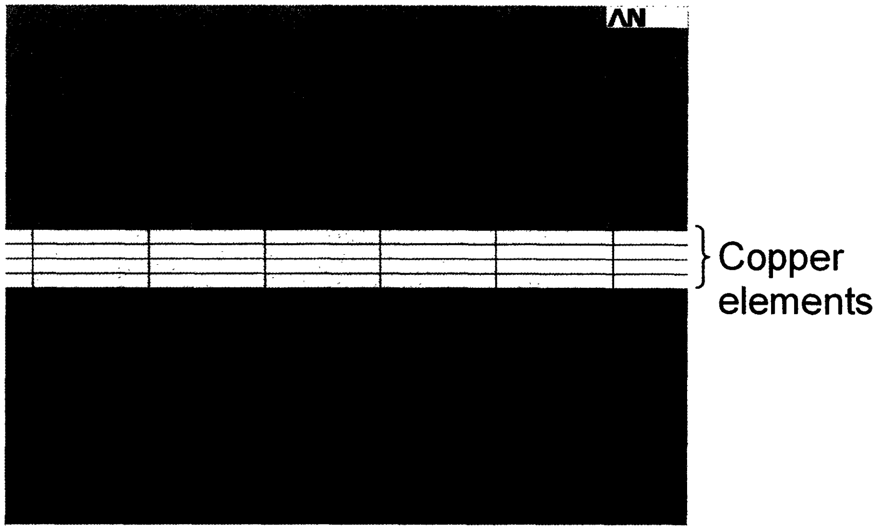

Figure 10: Mesh of the copper model with 0.5μm copper strips. The magnification shows the mesh within the copper strips, which has multiple layers to allow variation in stress and displacement across the thickness of the copper.

In the third step of the analysis, mechanical or electric loads were applied to the finite element model. The mechanical loads used on the sensor included a constraint on the bottom line for all degrees of freedom. The second constraint of interest was a pressure on the top line of the model representing a pressure on the upper surface of the sensor. A distributed pressure load was applied to the top of the top layer of polymer over the center node of the model. All mechanical forces are shown in Figure 11. Five distinct analyses were conducted to solve each model for five different pressure magnitudes, 0.02 MPa, 0.04MPa, 0.06 MPa, and 0.08 MPa.

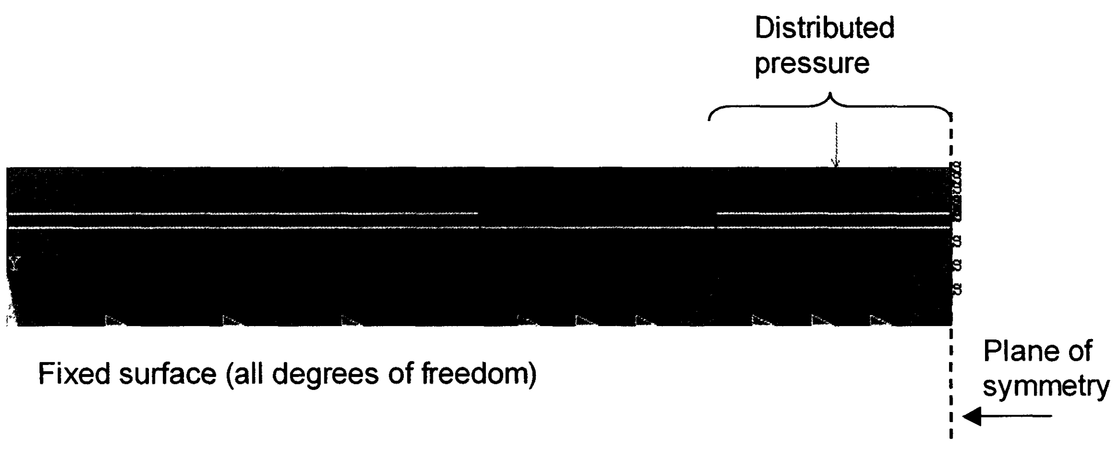

Figure 11: Mechanical loads on the sensor model, including pressure on the top surface of the center node, symmetry at the middle of the center node, and zero displacement along the bottom surface.

The final step in FEA is solving the model and processing results. Each model was solved to determine the magnitude of vertical (y) displacement and horizontal (x) stress throughout the sensor after application of each load magnitude. The analysis was static because the desired information corresponded to the final state of the sensor after application of the load. For each solution, contour plots of the vertical displacement and horizontal stress were produced throughout the sensor, with magnified plots of specific areas of interest. The magnified plots included the point of maximum horizontal stress, the vertical displacement of the copper strips at the center of the pressure load, and the displacement and stress at the edge of the node under pressure.

A more detailed list of analysis steps for this model, including specific ANSYS commands, is included in Appendix B.

2.1 Primary Assumptions

A primary goal of this project was to develop a finite element model that was functional, efficient, and could be modified to analyze the effect of a large variety of parameters. Several assumptions were made to simplify the model to allow it to be processed efficiently in ANSYS. Some of these assumptions can be modified in future analyses to reflect the more complex (often nonlinear) behavior of the materials in the sensor.

For this analysis, the material properties of copper were assumed to be the material properties of bulk copper. Nano-scale copper has material properties that depend on its geometry, but these properties are not well understood. It is necessary to do experimental materials testing of specific copper geometries to determine the nano-scale properties. The initial FEA was used to narrow the range of copper geometries to test experimentally. Once such testing is done to determine the material properties of copper for the thicknesses most likely to be used in the sensor, this more accurate material data can be input to the model.

For electrical analysis, the copper was assumed to be a perfect conductor with a relative permittivity of infinity. However, the maximum value of relative permittivity allowed in ANSYS is 1e60, so this value was used for copper's relative permittivity instead of infinity.

The PDMS has a Poisson's ratio of approximately 0.5, so the maximum value allowed in ANSYS, 0.499, was used for both PDMS variations. The stiffness of both PDMS variations was calculated from a linear approximation of nonlinear experimental data for uniaxial compression. Poisson's ratio and all electrical properties were assumed to be standard for PDMS and to not differ between the two PDMS variations, polymers A and B, used in the sensor. Experimental testing of these properties for each PDMS variation is necessary if greater accuracy is desired. Before conducting these experimental tests for numerous PDMS variations, the stiffness of the PDMS for an optimized sensor can be determined using the model. Experimental testing can then be conducted to determine the base-to-catalyst ratios that produce the desired stiffness. PDMS compositions that most closely produce the desired stiffnesses can be experimentally tested to determine their actual stress-strain curves, these curves can be used in new FEA, and optimized polymers can be developed through iterations of this cycle. Analysis of PDMS behavior with cyclical loads should also be analyzed and tested because of the hysteresis apparent in the stress-strain curves between push and release. This hysteresis should be incorporated into material models for simulations of sensor loading and unloading so that the sensor behavior can be optimized for the repetitive loading it will encounter in applications.

3.0 Results

The most important parameters investigated using FEA were the vertical displacement of the copper strips relative to each other and the magnitude of horizontal stress in the copper. Contour plots were generated using ANSYS to display these parameters for each model.

The model consists of three copper nodes with a pressure applied across the center one, but the model is symmetric about a vertical line down the middle of the center node. All results are shown with the left half of the sensor only; the right half is a mirror image of the contours in the figures. All figures are drawn to scale and all units are μMKSv (microMeter, Kilogram, Second, Volt).

3.1 Displacement

The vertical displacement of the sensor with 0.5μm thick copper under a load of 0.06MPa is shown in Figure 12. Similar contour plots of vertical displacement were produced for loads of 0.02MPa, 0.04MPa, 0.06MPa, and 0.08MPa for copper strips of thicknesses 0.3μm, 0.5μm, 3μm, and 9μm. All results are shown in Appendix C. As shown in Figure 12, the highest magnitude of displacement occurs directly under the pressure load. The highest magnitude of displacement is negative because of the downward pressure force, so ANSYS designates this maximum (negative) displacement as MN. The greatest upward displacement is designated as MX in the figures because it is the greatest displacement in the positive y-direction.

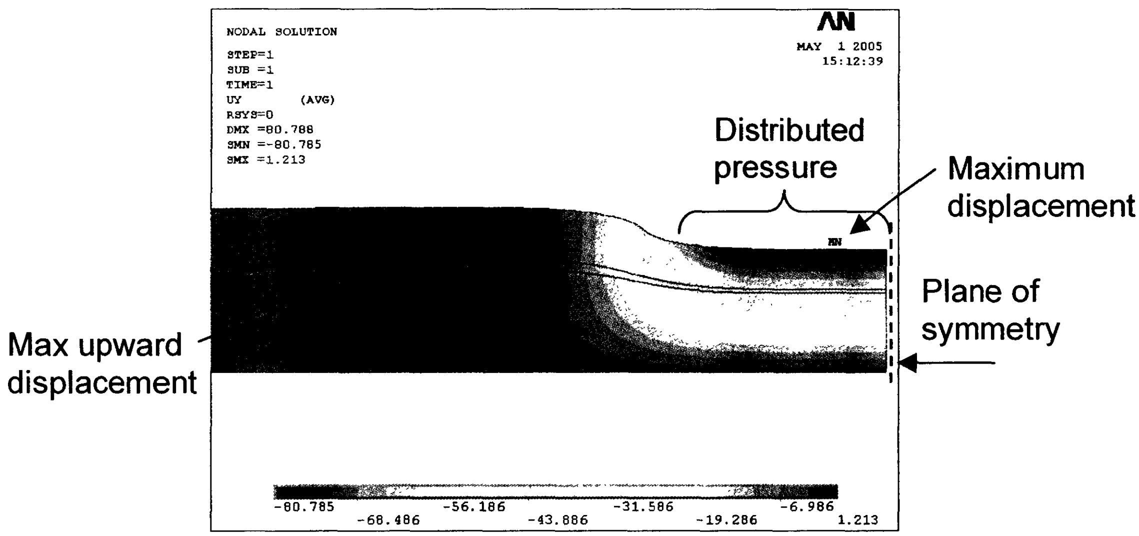

Figure 12: Vertical displacement (μm) of the sensor with 0.5μm copper under a load of 0.06MPa.

The capacitance of the sensor is determined by the distance between the copper strips, so the primary locations of interest for vertical displacement were at the center of pressure and at the minimum distance between the copper strips. For some of the models, the minimum separation of the copper strips occurred at the edge of the copper node, while for others it occurred at the center of pressure. A magnified image of the vertical displacement at the center of pressure for the sensor with 0.5μm thick copper under a load of 0.6MPa is shown in Figure 13. The position of the copper strips is known before application of the load, so the vertical displacement can be used to determine the distance between the strips after deformation.

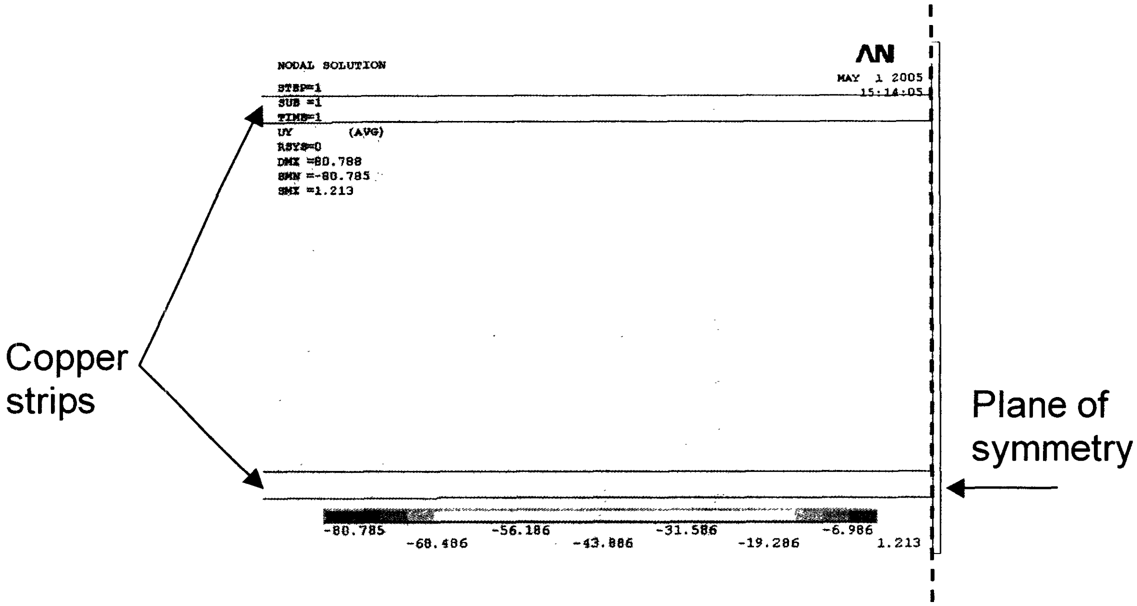

Figure 13: Vertical displacement (μm) of the 0.5μm thick copper strips under a load of 0.06MPa. The vertical line on the far right is the plane of symmetry and the center of the pressure load.

The 0.5μm thick copper has smaller strip separation at the center of pressure than at the node edge for every load magnitude. The vertical displacement at the edge of the copper node for 0.5μm thick copper under a load of 0.6MPa is shown in Figure 14. For the sensor with 9μm thick copper, however, the smallest strip separation is located at the node edge for all load magnitudes, rather than at the center of pressure. This does not occur because the 9μm strips deform more, but because pressure on the strips deforms the polymer between the strips at the edge of the copper node. The vertical displacement at the edge of the copper node for 9μm thick copper under a load of 0.06MPa is shown in Figure 15.

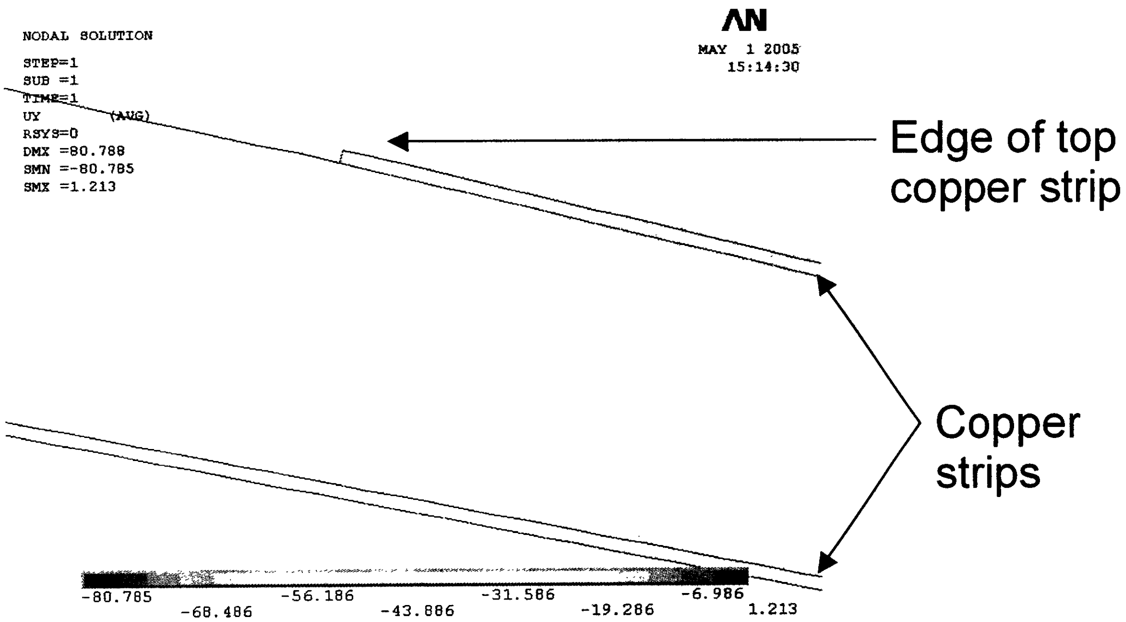

Figure 14: Vertical displacement (μm) of 0.5μm-thick copper under a weight of 0.06MPa, at the edge of the center copper node.

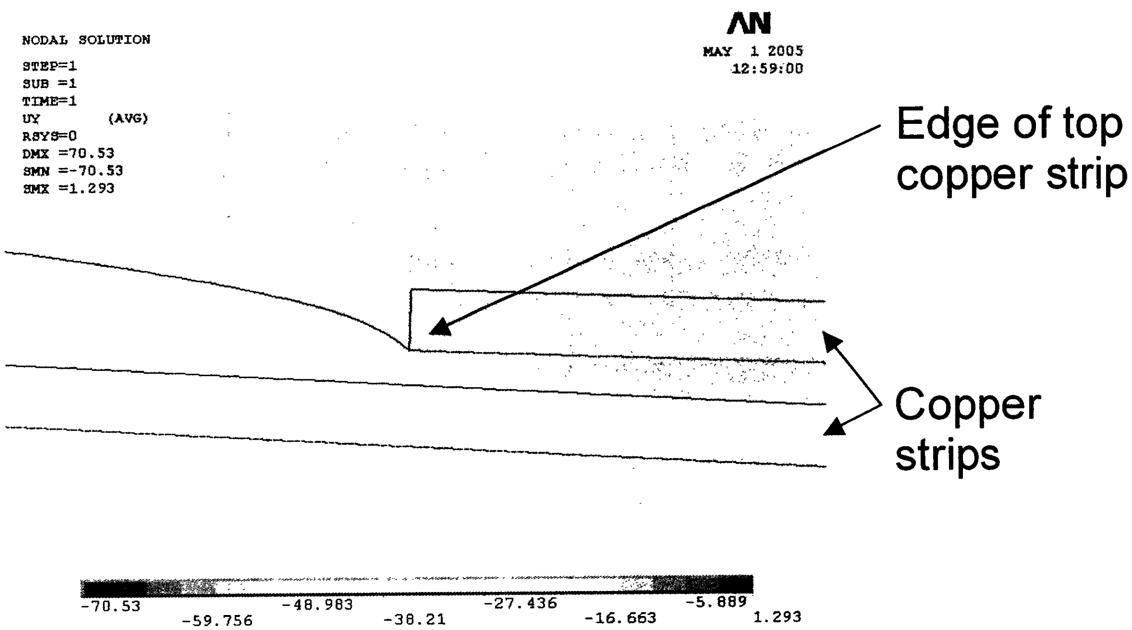

Figure 15: Vertical displacement (μm) of 9μm thick copper under a weight of 0.06MPa, at the edge of the center copper node. The edge of the copper strip is the location of the shortest distance between the copper strips.

The separation between the copper strips at the center of pressure (middle of the center node) and edge of the center node are listed in Table 2, along with the location of the minimum distance between copper strips.

Table 2: Location of minimum distance between copper strips

Copper thickness (μm)	Pressure (MPa)	Separation at pressure center (μm)	Separation at node edge (μm)	Location of minimum separation
9	0.02	17.61	15.21	Edge of node
9	0.04	12.82	10.42	Edge of node
9	0.06	9.23	5.64	Edge of node
3	0.02	15.80	15.80	Equal at edge and center
3	0.04	11.60	11.60	Equal at edge and center
3	0.06	7.40	7.40	Equal at edge and center
3	0.08	3.20	3.20	Equal at edge and center
0.5	0.02	15.90	17.27	Center of pressure
0.5	0.04	11.80	14.53	Center of pressure
0.5	0.06	7.70	11.80	Center of pressure
0.5	0.08	3.60	9.07	Center of pressure
0.3	0.02	15.90	17.26	Center of pressure
0.3	0.04	11.79	14.53	Center of pressure
0.3	0.06	7.69	11.79	Center of pressure
0.3	0.08	3.58	9.06	Center of pressure

The capacitance of the sensor depends on the average distance between the copper strips at each node. This separation distance is affected by the separation at the center of pressure and at the edge of the node, but the electric field spreads out at the edge of the node so the separation distance at the edge has less influence on capacitance than the separation at the node's center. The separation at the center of the node was used as a standard of comparison for vertical displacement. This center separation is plotted in Figure 16 as a function of the thickness of the copper strips and the magnitude of the pressure load. The distance between the copper strips has low dependence on the thickness of the strips for the three thinnest strips, though the dependence increases with pressure magnitude. The differences between the three thinnest copper thicknesses are less than 1% for the 0.02MPa load, rising to only 11% for the 0.08MPa load. The difference between the copper separation for the 9μm thick copper and the 3μm thick copper for the 0.06MPa load is much higher, about 24%. (Accurate results were not achieved for the 0.08MPa load on the 9μm copper.)

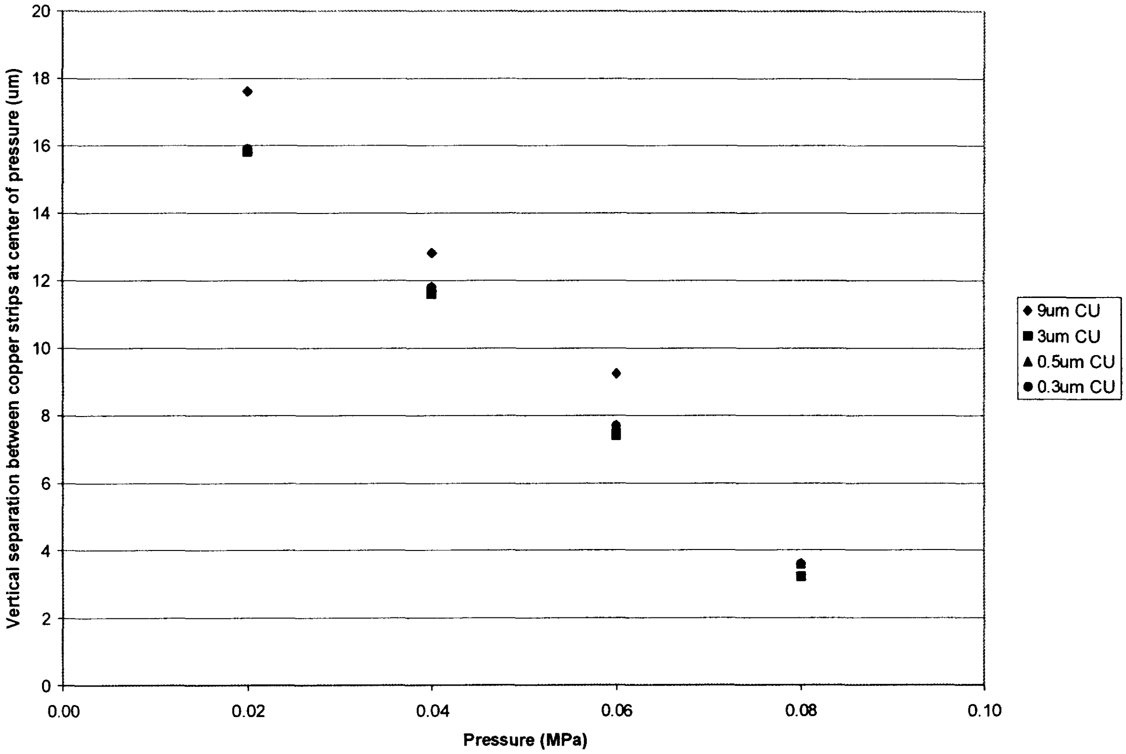

Figure 16: Distance (μm) between the two layers of copper strips at the center of pressure, as a function of the magnitude of the pressure load and the thickness of the copper strips.

3.2 Stress

The horizontal component of stress was plotted throughout the sensor, and the image generated for the 0.5μm copper under a load of 0.06MPa is shown in Figures 17 and 18. The stress state throughout the polymer layers is approximately uniform, but the magnitude of stress in the copper increases where the copper is deformed. The maximum magnitude of horizontal stress is indicated by MX for the maximum positive stress and MN for the maximum negative stress. Figure 17 shows the location of the maximum magnitude of horizontal stress, marked by MX.

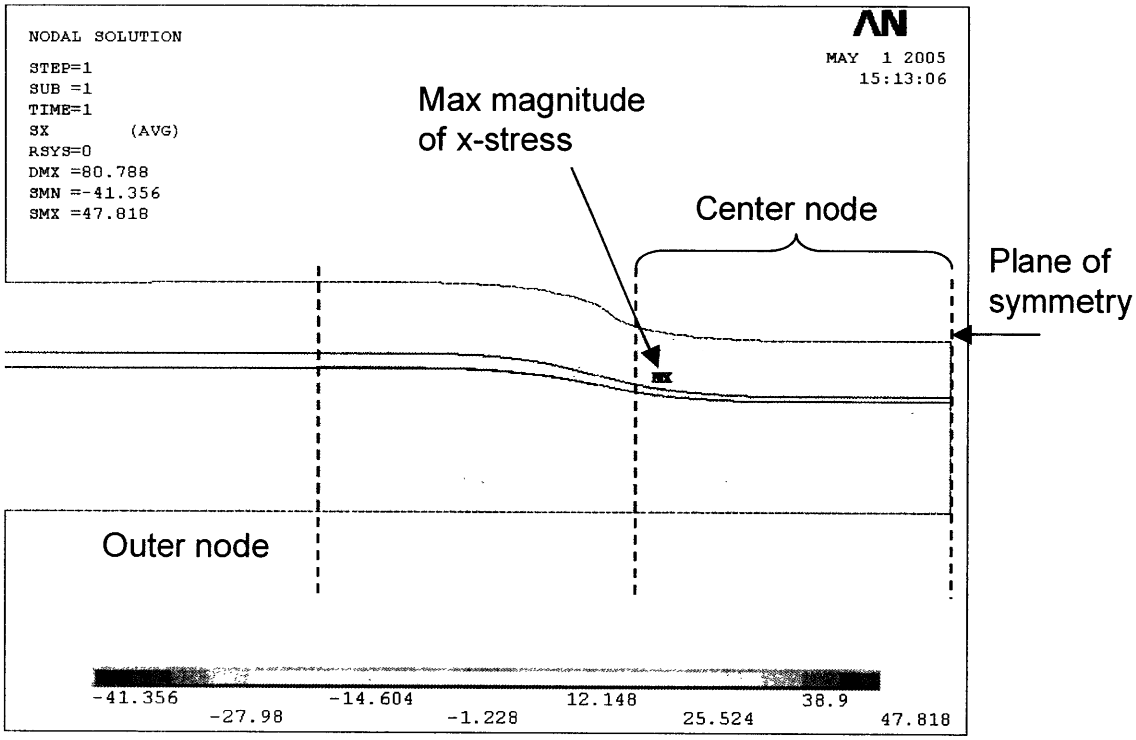

Figure 17: Horizontal component of stress (MPa) in sensor with 0.5μm thick copper strips under a weight of 0.06MPa. The location of the maximum stress magnitude is marked by MX and is located near the edge of the center node.

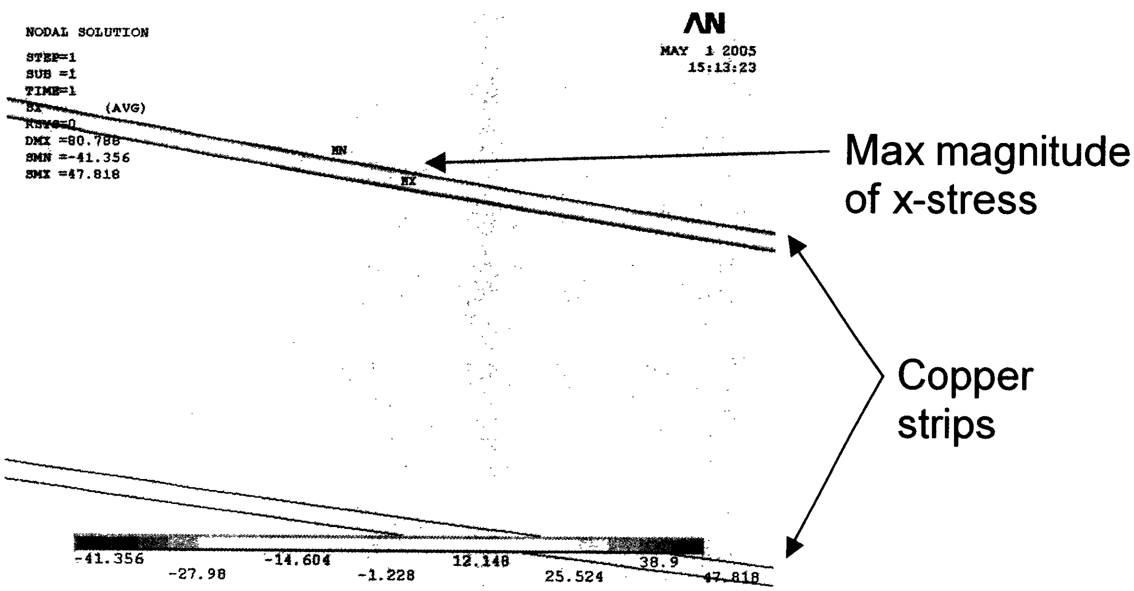

Figure 18: Horizontal component of stress (MPa) in sensor with 0.5μm thick

copper strips under a weight of 0.06MPa, magnified at the point of maximum stress magnitude.

The maximum and minimum of stress values in each model were located in the sensor and recorded. These values are plotted as a function of copper thickness and pressure magnitude in Figure 19. The increase in stress as a function of pressure load is perfectly linear for each thickness of copper, as is expected with perfectly linear materials. The percent difference between stresses in different copper geometries under a given pressure load is the same for every load magnitude.

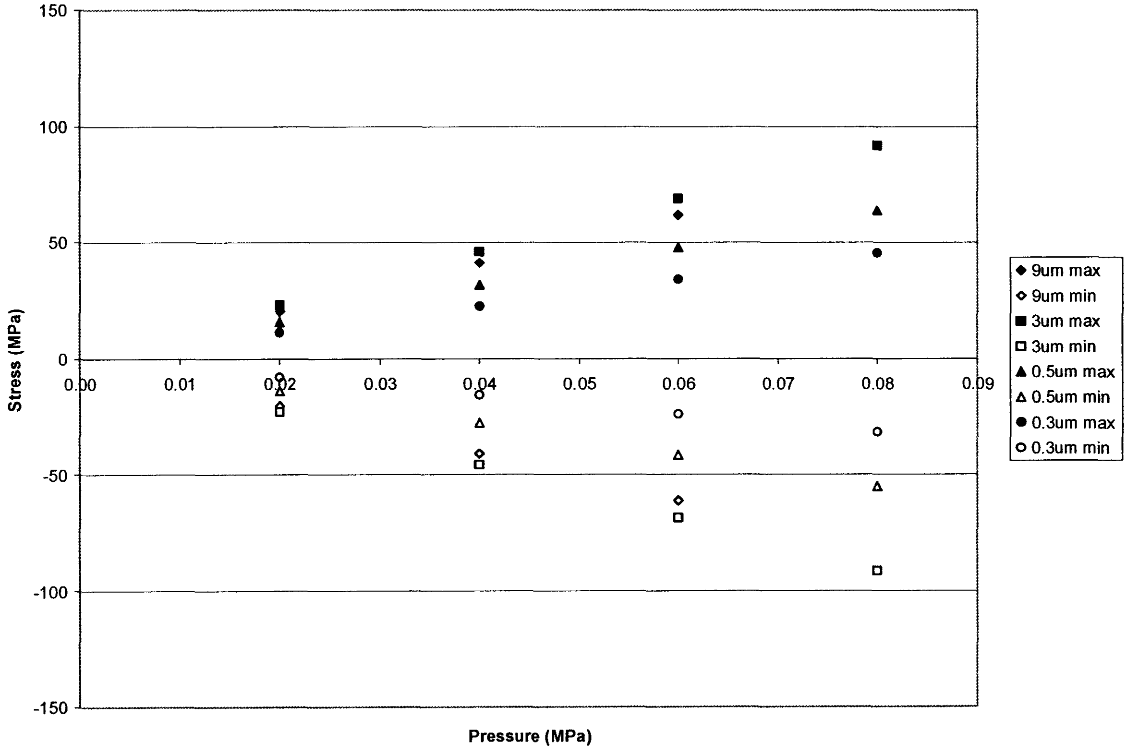

Figure 19: Horizontal stress in the sensor plotted as a function of pressure magnitude for each thickness of copper. For each geometry, the increase in stress with pressure magnitude is linear (zero measurable error).

The location of maximum and minimum displacement and stress varied with different thicknesses of copper. The deformed sensor is divided into regions in Figure 20. The location of the displacement and stress extremes are listed in Table 3 according to the letter in Figure 20 to which their location most closely corresponded. The maximum and minimum stress are always located in close proximity to each other, so they are listed as one entity.

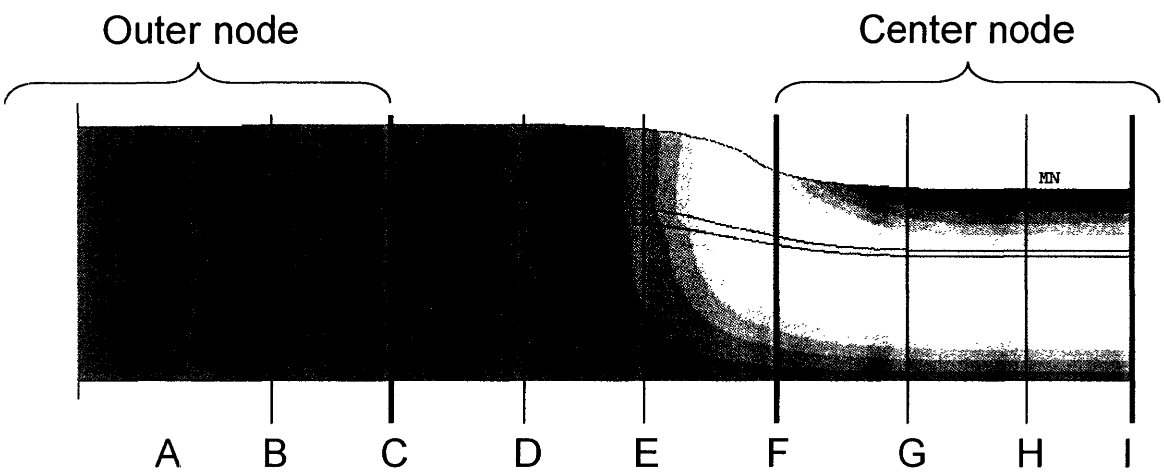

Figure 20: Division of the model into regions for description of location of extreme values of displacement and stress. This model is the vertical displacement of the sensor with 0.5μm thick copper strip under 0.06MPa. The max upward displacement is located at D and the max magnitude of displacement is located at H, as designated in Table 3.

Table 3: Location of Extreme Displacement and Stress Values for each Copper Thickness.

Copper Thickness (μm)	Max and Min X-Stress Location	Max upward Y-Displacement Location	Max magnitude Y-Displacement Location (downward)
9	H, bottom CU strip	A	I
3	E, bottom CU strip	C	I
0.5	G, top CU strip	D	H
0.3	G, top CU strip	D	H

The maximum magnitude of displacement occurs near the pressure center in all cases. The maximum upward displacement value is close to zero; it is where the sensor moves in the opposite direction of the pressure force. This occurs because the pressure bends the sensor down at the center node and the copper strips tilt so the sections far from the pressure are lifted. The location of this maximum upward displacement is farther from the pressure sensor for thicker copper because the thicker copper bends less easily. There is less bend in the thicker copper strips, so they reach their maximum lift farther away from the pressure center.

The extreme values of stress are located where the copper is deformed and under pressure, often near bends in the strips. The extreme values of stress are always in the copper, not the polymer. The maximum and minimum values are located close to each other, on the top and bottom of the same copper strip. For the thinner copper strips they are located in the top copper strip, and for the thicker copper strips they are located in the bottom copper strip.

3.3 Electrostatic Analysis

The finite element model was designed to be used for either mechanical or electrostatic analysis, or a coupled electro-mechanical analysis. Although electrostatic behavior was not the subject of this project, the model was solved for an electrostatic solution to make sure it was capable of producing such a solution. The same geometry and mesh density were used for the electrostatic and mechanical models, but the element type was changed to an electrostatic element compatible with the mechanical element type. The mechanical loads were removed for the electrostatic analysis, and electric loads were applied to the model. Electric loads included a voltage of 20V applied to the top layer of copper strips and a voltage of 0V (ground) applied to the bottom layer of copper. The copper was idealized as a perfect conductor, with uniform voltage throughout.

The undeformed model was analyzed to determine the electric potential and electric field throughout the undeformed sensor. The electric potential in the sensor is shown in Figures 21 and 22.

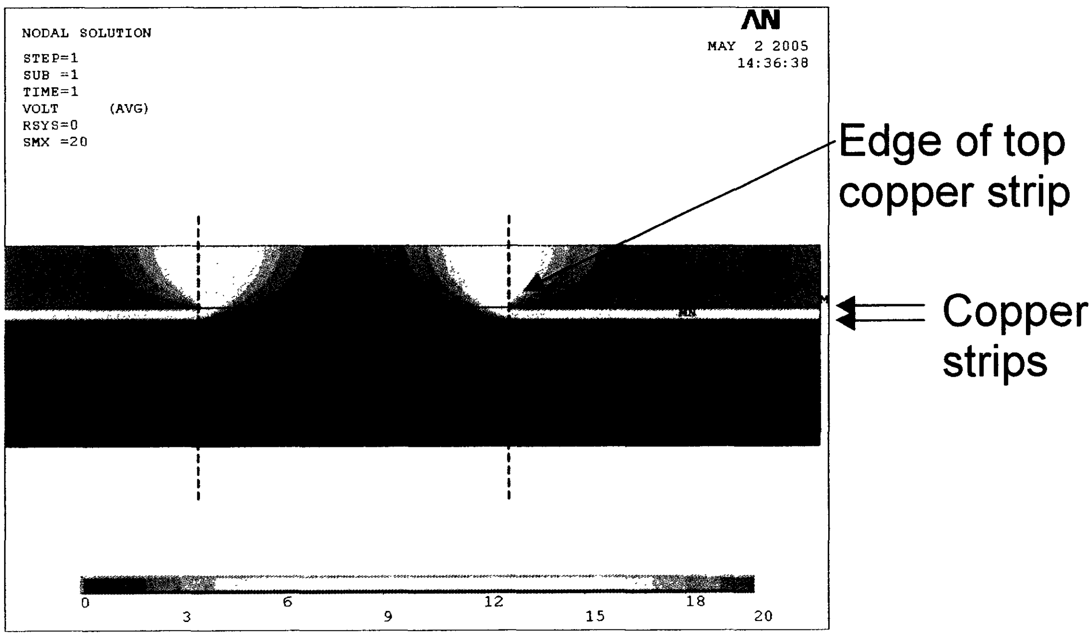

Figure 21: Electric potential (V) in the undeformed sensor (with 0.5μm thick copper strips). Loads of 20V were applied to each copper strip in the top layer and a load of 0V was applied to the bottom copper strip.

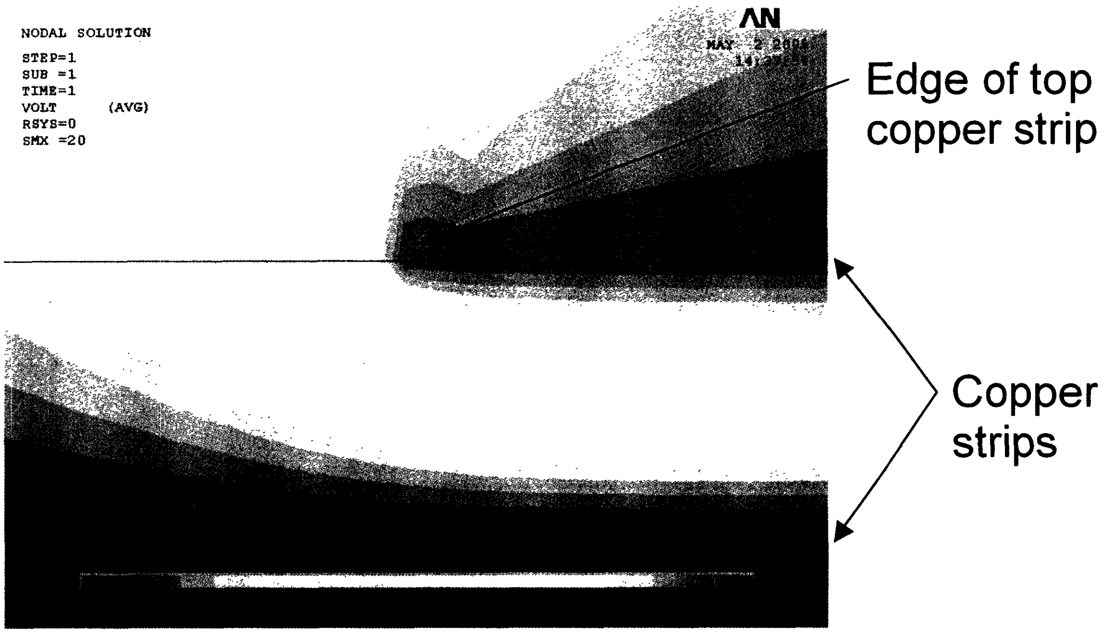

Figure 22: Electric potential (V) in the undeformed sensor (with 0.5μm thick copper strips) at the edge of a top copper strip.

The electric field resulting from the same load condition is shown in Figure 23. The electric field is concentrated where the copper strips are in close proximity and reaches a maximum on the top copper strip at the edges of each copper node. The area at the edge of one node is shown in Figure 24.

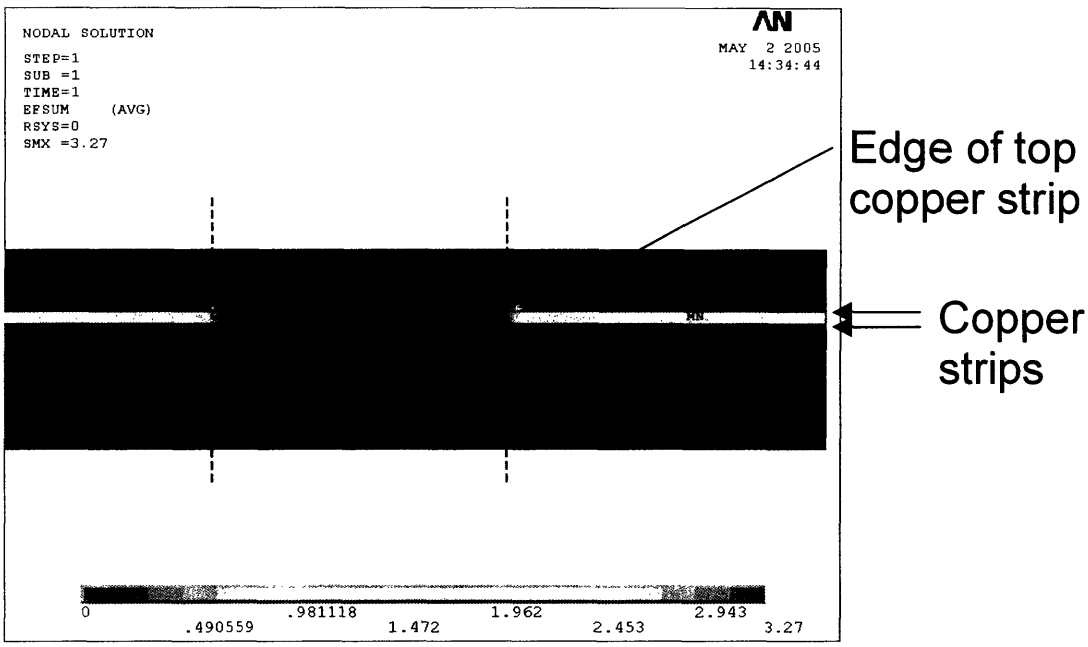

Figure 23: Electric field (V/μm) in the undeformed sensor (with 0.5μm thick copper strips) with a load of 20V on each top copper strip and a load of 0V on the bottom copper strip.

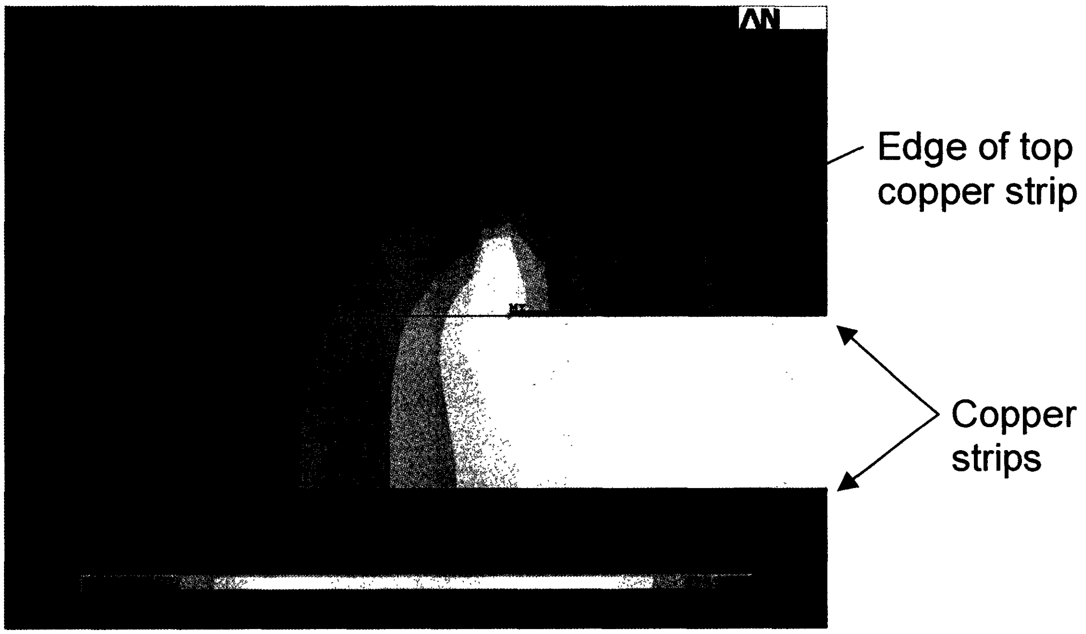

Figure 24: Electric field (V/μm) in the undeformed sensor (with 0.5μm thick copper strips) with a load of 20V on each top copper strip and a load of 0V on the bottom copper strip, magnified at the edge of one copper node.

The electrical and mechanical models can be coupled to conduct an analysis that accounts for both electrical and mechanical behavior. The loads can either be applied and analyzed simultaneously, or they can be sequentially coupled, with a solution produced for an applied pressure and then for a voltage applied to the deformed structure. This coupled analysis can be used in the future to determine how mechanical and electric behavior influence each other and to achieve solutions that more accurately reflect experimental behavior.

4.0 Discussion and Conclusions

Each solution was tested to determine whether the finite element model's behavior corresponded to the theories used to run the analysis, and comparisons of the specific solutions were used to develop recommendations for sensor optimization.

4.1 Finite Element Capabilities

The finite element model produced results that corresponded to theoretical predictions of actual behavior under the model's simplified assumptions. Magnitude of stress and displacements had approximately a linear relationship to load magnitude, for one geometric condition, as is expected with linear material stiffness. For loads between 0.02MPa and 0.06MPa, the elements deformed according to conservation of mass, so they were thinned and elongated under pressure, with deformations applied from each element to its neighbors. The same was true for loads of 0.08MPa on all copper thicknesses except 9μm. An accurate mesh deformation is shown at the center of pressure in Figure 25 and at the edge of the center node in Figure 26. In these figures, the number of elements in the deformed and undeformed mesh are equal, but the elements in the deformed mesh have changed shape under pressure.

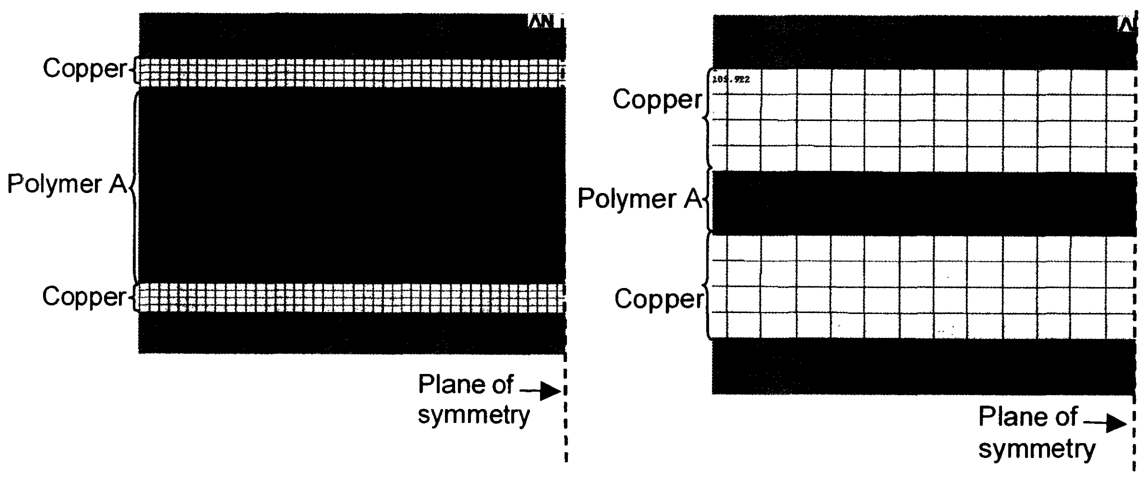

Figure 25: Mesh near the center of pressure for 3μm thick copper strips under 0.08MPa pressure. The figure on the left is the undeformed mesh (no load) and the figure on the right is the deformed mesh. There are an equal number of elements showing in the deformed mesh and the undeformed mesh, but the elements in the deformed mesh are thinned and elongated under pressure.

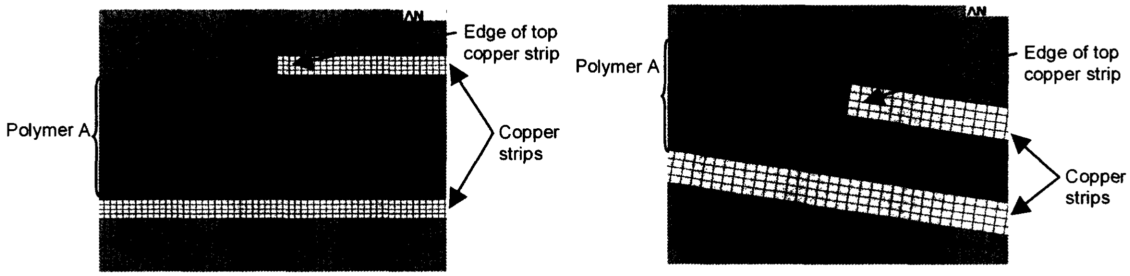

Figure 26: Mesh near the edge of the center node for 3μm thick copper strips under 0.08MPa pressure. There are an equal number of elements showing in the undeformed mesh (left) and the undeformed mesh (right), but the elements in the deformed mesh are bent under pressure.

However, for loads of 0.1MPa on all geometries and 0.08MPa on the 9μm copper thickness, the size of the elements was not sufficient to capture the large deformations. A small gradient in deformations from one element to the next was not possible, so elements put under greatest pressure were displaced past the neighboring elements, and the elements overlapped each other. The overlapped deformation occurred at the center of pressure or at the edge of the center node, at the location of minimum strip separation.

A deformed mesh for 3μm and 0.5μm thick copper strips under loads of 0.1MPa are shown in Figure 27 for the section near the center of pressure. In the 3μm model, the elements of the copper strips have not only overlapped each other, but have completely overlapped the polymer between them. In the 0.5μm model, the middle layer of polymer is visible between the copper strips, but thin sections of overlapped polymer elements are visible near the top and bottom edge of the polymer layer.

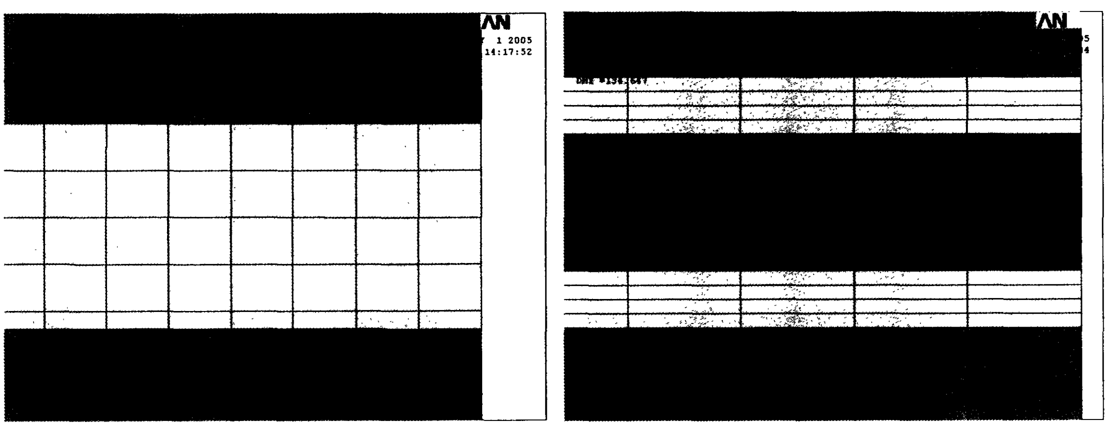

Figure 27: Deformed mesh of the copper strips and middle layer of polymer near the center of pressure in the 3μm model (left) and 0.5μm model (right) under a load of 0.1MPa. In the 3μm model, the copper strips completely overlap the middle polymer layer. In the 0.5μm model small sections of the overlapped polymer elements are visible at the top and bottom of the layer.

In some models, the overlapped mesh occurred at the edge of the center copper node instead of at the center of pressure. One such case, for 9μm thick copper strips under a load of 0.1MPa, is shown in Figure 28. Here, the copper strips overlap each other and the layer of polymer between them.

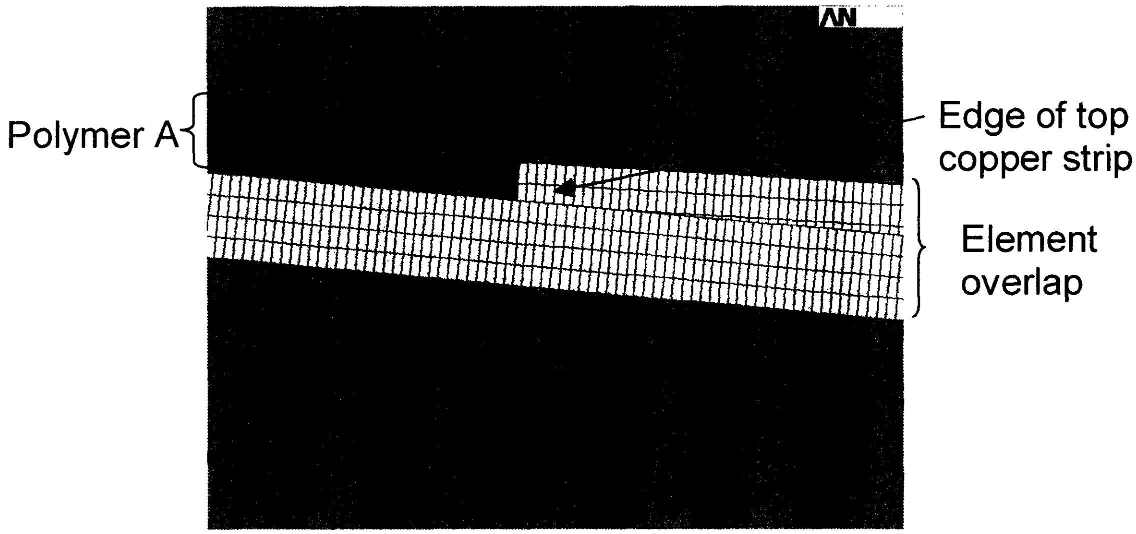

Figure 28: Deformed mesh in the 9μm model near the edge of the center node under 0.1MPa. The elements of the top copper layer are displaced past the middle polymer layer and part of the bottom copper layer, and the elements are too large to capture this displacement.

The mesh was too large to accurately solve the model for a load of 0.1MPa on all copper thicknesses and for a load of 0.8MPa on the 0.9μm thick copper. To improve the mesh, the number of horizontal rows of elements in the copper and middle polymer layers must be increased. The height of each element decreases as the number of rows within each material layer increases. In order to form rows of elements, the horizontal width of each element cannot be much larger than the height of each element. As a result, the number of elements in the copper and middle polymer layers must be increased drastically for small increases in the number of element rows. The mesh density in these material layers could not be increased high enough to produce accurate results without exceeding the element limit imposed by the available version of ANSYS.

The second limitation was the thickness of the copper strips that could be tested. In order to accurately capture the stress variation across the copper strips, there had to be several layers of elements across the thickness of each strip. For thinner copper strips, the maximum thickness of each element layer was reduced. To mesh the model with rows of elements in these layers, the spacing of the elements along the length of each strip had to be increased for thinner copper strips, and so the number of elements in the model increased. For copper strips of 0.1μm, the total number of elements required to mesh the model exceeded the number of elements in the available version of ANSYS. The thinnest strips accurately analyzed were 0.3μm thick.

4.2 Interpretation of Results and Recommendations

The finite element models analyzed for this project were used to determine how copper strip thickness affects the sensitivity and durability of the sensor. This information can be used to choose a limited range of copper strip thickness to be experimentally tested for verification of the model's predictions and further optimization.

The analysis shows that the separation between copper strips 9μm thick at the center of pressure is significantly (24%) more than thinner strips. Sensors produced with this thick copper are easier to manufacture but produce different electrical signals than sensors with thinner strips. These 9μm sensors may be less sensitive to changes in pressure load, especially at small load magnitudes. Also, the 9μm copper strips are brought so close together at the edge of the copper node that the sensor may resist deformation or may be damaged at high loads. It is necessary to test the variations in capacitance carefully through experimentation to determine whether the 9μm strips produce an accurate sensor.

The stress in the copper is 28.8% higher in the 0.5μm thick strips than in the 0.3μm thick strips, 22.8% higher in the 9μm thick strips than in the 0.3μm thick strips, and 10.2% higher in the 3μm thick strips than in the 9μm thick strips. The higher the stress in the copper strips, the more likely the copper is to decrease in performance quality through use. The highest stress is in the 3μm thick copper strips, so the sensor made with these strips has the least durability. The copper should be tested to determine whether the stress level in the 3μm thick copper strips would significantly affect their behavior under the expected loads and through the expected lifetime of the sensor. If the stress level significantly decreases the durability of the copper, a different strip thickness should be used to ensure accuracy of the sensor throughout its lifetime.

Loads of 0.1MPa are expected in the applications of the sensor, but these loads could not be tested using this finite element model. Sensors should be studied carefully to make sure loads above 0.08MPa are accurately converted to electrical signals, because it is possible that the deformation at these load levels is so great that the material does not deform according to the same patterns observed for smaller loads.

According to the analysis, the 0.5μm and 0.3μm thick copper strips combine high durability and high sensitivity to produce sensors that are expected to have the highest quality performance of the geometries tested. The thinner copper strips are more challenging to manufacture, so the 0.5μm copper strips are optimal for use in the sensor.

4.3 Future Use of the Model

Finite element analysis is a powerful tool for optimizing the design of the sensor, but it should not be used without experimental verification and iterations with improved modeling accuracy. The results of this analysis should be used to choose copper strip thicknesses near 0.5μm to be tested experimentally. The actual material properties and durability of nano-scale strips of this thickness should be determined and input to the model to improve the model's accuracy. The model should also be used to vary the stiffnesses and thicknesses of polymers A and B to determine which stiffnesses and thicknesses are optimal for each layer and to vary the width of the copper strips and spacing between them to determine the optimal geometry for the sensor. Once these and any additional parameters of interest are optimized through FEA, materials with the optimized characteristics should be produced and tested experimentally to verify the model and to further optimize the design. Nonlinear material properties from experimental data should be input to the model and analyzed. Finally, sensors with the optimized parameters should be produced and tested experimentally to verify the whole model and to fine-tune the sensor design. Finite element modeling significantly reduces the number of combinations of parameters that must be tested to optimize the sensor design, but it cannot replace experimentation. This model forms a foundation for extensive future analysis and experimental testing.

Appendix A

System of Units and Material Property Data

Table A1: Conversion from MKS units to μMKSv (microMeter-Kilogram-Second-volt) units[1]

Mechanical Parameter	**MKS Unit**	**Dimension**	**Multiply by this Number**	**To Obtain μMKSv Unit**	**Dimension**
Length	m	m		μm	μm
Force	N	$(kg)(m)/(s)^2$	01E+06	μN	$(kg)(\mu m)/(s)^2$
Time	s	s	1	s	s
Mass	kg	kg	1	kg	kg
Pressure	Pa	$kg/(m)(s)^2$	01E-06	MPa	$kg/(\mu m)(s)^2$
Stress	Pa	$kg/(m)(s)^3$	01E-06	MPa	$kg/(\mu m)(s)^2$
Young's Modulus	Pa	$kg/(m)(s)^4$	01E-06	MPa	$kg/(\mu m)(s)^2$
Thermal Parameter					
Conductivity	W/mK	$(kg)(m)/(K)(s)^{\wedge 3}$	1.00E+06	pW/(μm)K	$(kg)(\mu m)/(K)(s)^3$
Specific Heat	J/(kg)K	$(m)^2/(K)(s)^2$	1.00E+12	pJ/(kg)K	$(\mu m)^2/(K)(s)^2$
Electrical Parameter					
Voltage	V	$(kg)(m)^2/(A)(s)^3$	1	V	$(kg)(\mu m)^2/(pA)(s)^3$
Conductivity	S/m	$(A)^2(s)^3/(kg)(m)^3$	1	pS/μm	$(pA)^2(s)^3/(kg)(\mu m)^3$
Resistivity	Ωm	$(kg)(m)^3/(A)^2 (s)^3$	1.00E+06	Toμm	$(kg)(\mu m)^3/(pA)^2(s)^3$
Permittivity	F/m	$(A)^2(s)^4/(kg)(m^3)$	1.00E+06	pF/μm	$(pA)^2(s)^2/(kg)(\mu m)^3$
Capacitance	F	$(A)^2(s)^4/(kg)(m^4)$	1.00E+12	pF	$(pA)^2(s)^4/(kg)(\mu m)^2$
Electric Field	V/m	V/m	1.00E-06	V/μm	$(kg)(\mu m)/(s)^3(pA)$

[1] System of Units, ANSYS Coupled-Field Analysis Guide, ANSYS Release 8.1 Documentation Preview

Table A2: Material properties conversion from MKS to μMKSv units

Mechanical Parameter	**MKS Unit**	**Dimension**	**Multiply by this Number**	**To Obtain μMKSv Unit**	**Dimension**
Young's Modulus	Pa	$kg/(m)(s)^4$	01E-06	MPa	$kg/(\mu m)(s)^2$
Copper	1.10E+11	1.10E+11	01E-06	1.10E+05	**1.1E+05**
PDMS, middle	46614	4.66E+04	01E-06	0.0682	**0.0682**
PDMS, outer	202854	2.03E+05	01E-06	0.20285	**0.20285**
Poisson's ratio	constant	constant		in ANSYS	in ANSYS
Copper	0.343	3.43E-01	1.00E+00	0.343	**0.343**
PDMS	0.5	5.00E-01	must be below 0.5	0.499	**0.499**
Thermal Parameter					
Conductivity	W/mK	$(kg)(m)/(K)(s)^3$	1.00E+06	pW/(μm)K	$(kg)(\mu m)/(K)(s)^3$
Copper	385	385	1.00E+06	3.85E+08	**3.85E+08**
PDMS	0.15	0.15	1.00E+06	1.50E+05	**1.5E+05**
Specific Heat	J/(kg)K	$(m)^2/(K)(s)^2$	1.00E+12	pJ/(kg)K	$(\mu m)^2/(K)(s)^2$
Copper	385	385	1.00E+12	3.85E+14	**3.85E+14**
PDMS	1.46E+03	1460	1.00E+12	1.46E+15	**1.46E+15**
Electrical Parameter					
Conductivity	S/m	$(A)^2(s)^3/(kg)(m)^3$	1	pS/μm	$(pA)^2(s)^3/(kg)(\mu m)^3$
Copper	5.88E+07	58823529.4	1	5.88E+07	**58823529.412**
PDMS	2.50E-14	2.5E-14	1	2.50E-14	**03E-14**
Resistivity	Ωm	$(kg)(m)^3/(A)^2(s)^3$	1.00E+06	TΩμm	$(kg)(\mu m)^3/(pA)^2(s)^3$
Copper	1.70E-08	1.7E-08	1.00E+06	0.017	**0.017**
PDMS	4.00E+13	4E+13	1.00E+06	4.00E+19	**4.00E+19**
Relative Permittivity	constant	constant		in ANSYS	in ANSYS
Copper	infinite	infinite	must be below 1e60	9.99E+59	**9.99E+59**

Appendix B

Model Development Procedure

These are instructions according to my preferred modeling and analysis methods. There are alternate approaches, but these instructions can be used to replicate the model that produced the results included in this paper. These instructions used the GUI because I am more experienced with this interface; however all actions have equivalent commands for the command-line interface.

Designate a structural discipline for any mechanical analysis and an electromagnetic discipline for an analysis that includes electricity. This is done in Preferences.

Enter the preprocessor.
Set the element type. Use Plane82 for structural analysis and Plane121 for electric field analysis. These element type were chosen because they are planar eight-node elements and are compatible for coupled electro-mechanical coupled analysis.
Menu Path: Preprocessor>Element Type>Add

Input material parameters. There are three materials used in this model, copper and two polymers. The material properties for all these materials are listed in Table 1. Input the material properties into the material models. All properties were inputted in μMKSv (microMeter, Kilogram, Second, Volt) units, which are listed in Appendix A. These properties can be written to a material data file to be used in future models.
Menu Path: Preprocessor>Material Props>Material Models
Menu Path: Preprocessor>Material Props>Write to/Read from File

Build the geometry.
The geometry consists of rectangular areas, which were built according to the dimensions shown in Figures # and #. The geometry was constructed by creating one rectangle and dividing it into the components of the sensor. After divisions, a merge command must be used to ensure there are no duplicate entities. The model is symmetric, so, to produce an efficient analysis, only half the geometry was modeled and symmetry boundary conditions were used along the vertical middle line of the center node (see Loads).
Menu Path: Preprocessor>Modeling>Create>Areas>Rectangle>By 2 Corners
Menu Path: Preprocessor>Modeling>Operate>Booleans>Divide
Menu Path: Preprocessor>Numbering Controls>Merge Items
Menu Path: Preprocessor>Numbering Controls>Compress Numbers

Mesh the model
The model must be meshed very carefully to ensure that the mesh is fine enough in all areas, especially those with concentrated loads or extreme dimensions, but large enough to be processed with the available computing power. First, element and material types must be assigned to each area.
Menu Path: Preprocessor>Meshing>Mesh Attributes>Picked Areas

Size Controls

The size of the elements was set manually. The number of divisions required for each line varied with copper thickness and increased with pressure load. These size controls were chosen to make sure the variations in displacement and stress across each material layer were accurate, the mesh was small in concentrated areas but large along the edges of the model (to reduce the number of required elements), and the aspect ratios of individual elements were not extreme.

In order to use create a mesh that is concentrated along one side of each area and expands across the areas, the number of elements on the sides of each area must be equal and the difference between the number of elements on the top and bottom of each model must be an even number. Each area must be meshed with free meshing, so the elements can conform to the changing size controls across each area.
Menu Path: Preprocessor>Meshing>Size Contrls>ManualSize>Lines>Picked Lines
Menu Path: Preprocessor>Meshing>Mesh>Areas>Free>Pick All

Element Connectivity

Merge and renumber all items to ensure that there are no extra nodes or elements, so all entities at the same location are considered one entity.
Menu Path: Preprocessor>Numbering Controls>Merge Items
Menu Path: Preprocessor>Numbering Controls>Compress Numbers

Loads

For mechanical models, set the displacement of all bottom lines of the model to be zero, for all degrees of freedom. This edge of the model is fixed. Add a pressure load to the line on top of the center node of copper, indicated by the arrow in Figure #. The pressure loads used in this analysis ranged from 0.02-0.1MPa, corresponding to intervals in the 0g-10g range intended for experimentation. A symmetry boundary condition must be placed on the plane of symmetry in the middle of the center copper node.
Menu Path: Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Lines
Menu Path: Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Lines
Menu Path: Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Lines

For electrical models, all applied loads were voltages on boundaries. A voltage of 20V was applied to the areas corresponding to copper segments in the top row, and a voltage of 0V was applied to the areas corresponding to the copper in the bottom row.
Menu Path: Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Areas

Coupled Analysis

For coupled analysis, it is necessary to save the electric physics environment and the structural physics environment. The physics environment includes the geometry, mesh, and loads in a model. Physics environments can be written when the model is ready to be solved and then read whenever the environment should be applied in the coupled analysis.
Menu Path: Preprocessor>Physics>Environment>Write

Solution

All analysis conducted were static analysis, which is the default in ANSYS. Once the loads and mesh are created, the model can be solved.

Menu Path: Solution>Analysis Type>New Analysis

Menu Path: Solution>Solve>Current LS

Post processing

Once the model is solved, the last set of results should be read and then plotted. Plots can include the deformed mesh of the model and contours of the vertical (y) displacement or horizontal (x) stress. The number of contours can be increased by setting the plot controls, and hardcopies of the images can be written to image files.

Menu Path: General Postprocessor>Read Results>Last Set

Menu Path: General Postprocessor>Plot Results>Deformed Shape

Menu Path: General Postprocessor>Plot Results>Contour Plot>Nodal Solu

Menu Path: General Postprocessor>Plot Results>Contour Plot>Element Solu

Utility Bar: PlotCtrls>Style>Contours>Uniform Contours>Number of contours

Utility Bar: PlotCtrls>Device Options>Use extra colors for>Contours

Utility Bar: PlotCtrls>Hard Copy>To File

Appendix C

Contour plots for all load cases for all copper thicknesses.

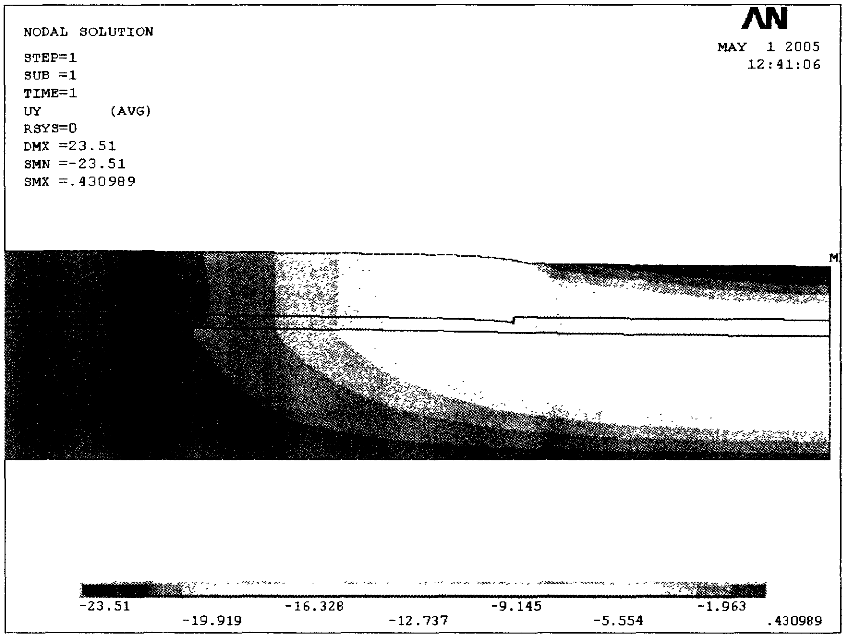

Figure C1: Y-displacement (μm) of copper 9μm thick under a pressure of 0.02MPa.

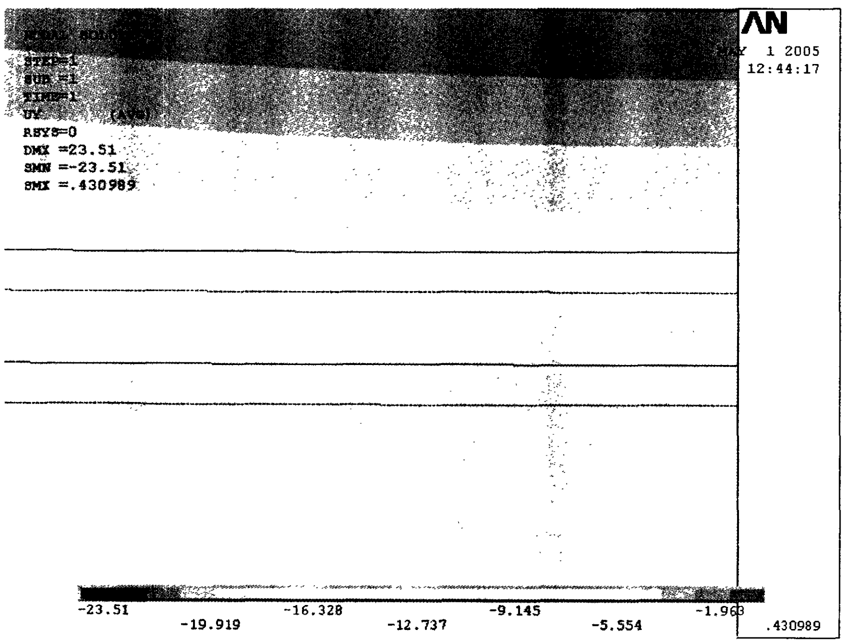

Figure C2: Y-displacement (μm) of copper 9μm thick under a weight of 0.02MPa. The vertical line farther right is the center of pressure.

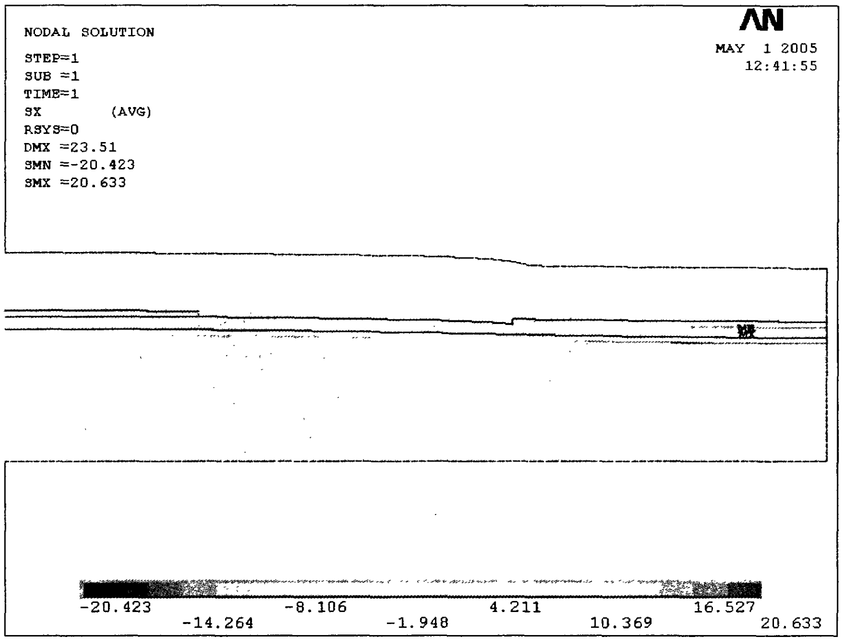

Figure C3: X-component of stress (MPa) of copper 9μm thick under a weight of 0.02MPa.

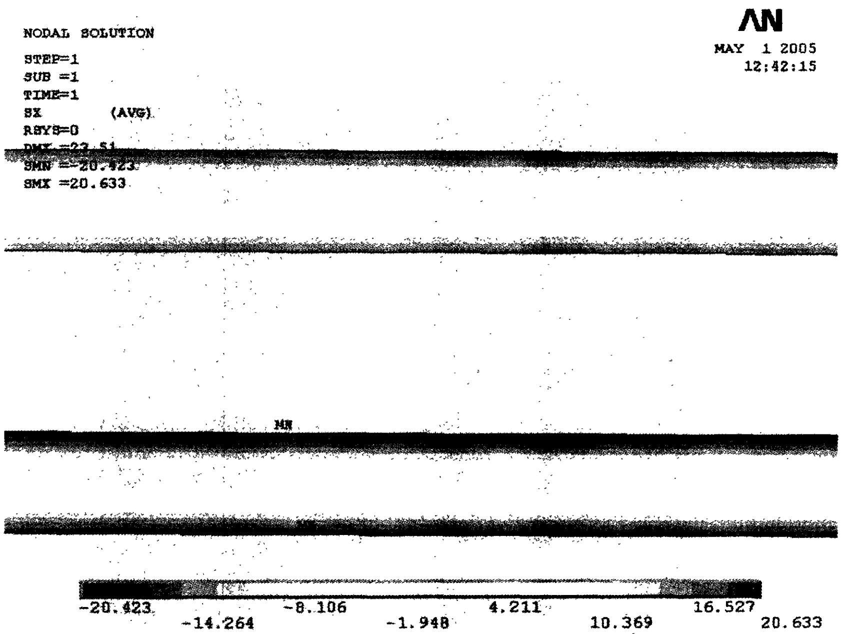

Figure C4: X-component of stress (MPa) of copper 9μm thick under a weight of 0.02MPa at the point of maximum and minimum stress.

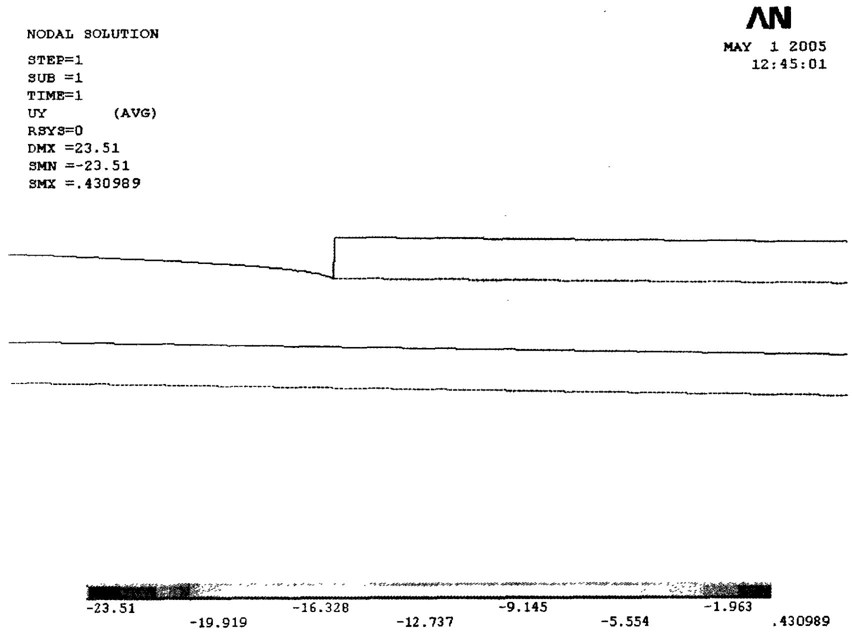

Figure C5: Y-displacement (μm) of copper 9μm thick under a weight of 0.02MPa, at the edge of the center copper node.

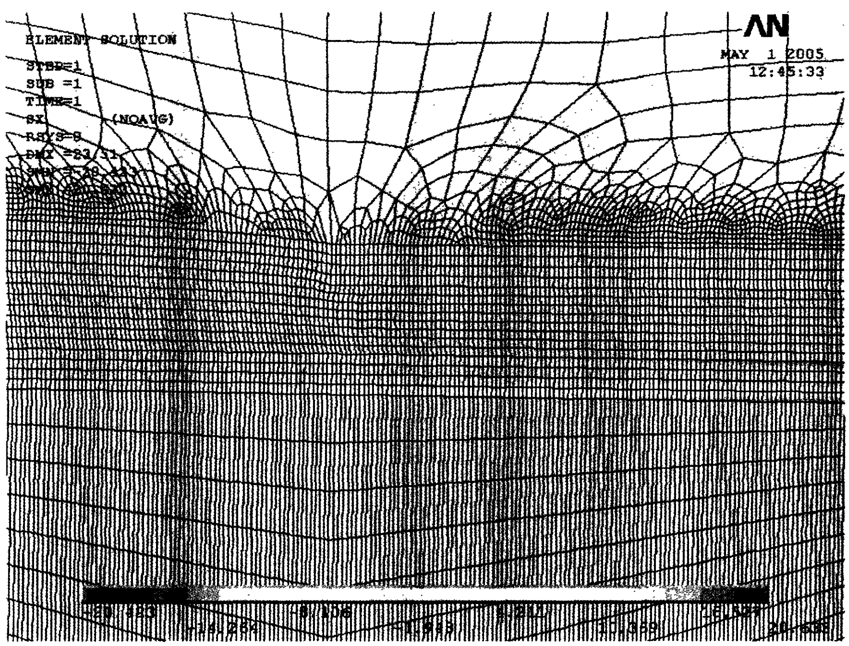

Figure C6: X-component of stress (MPa) of copper 9μm thick under a weight of 0.02MPa, at the edge of the center copper node, mesh showing.

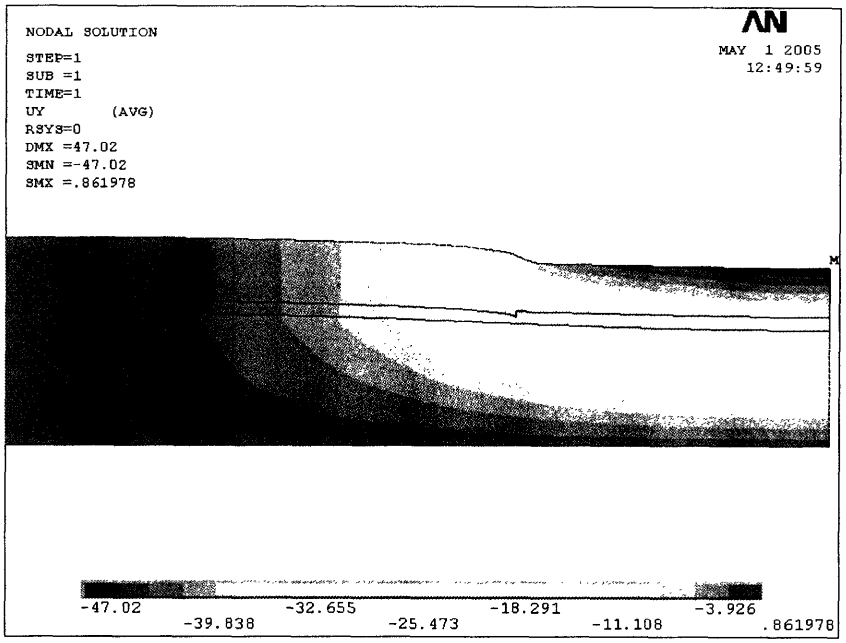

Figure C7: Y-displacement (μm) of copper 9μm thick under a pressure of 0.04MPa.

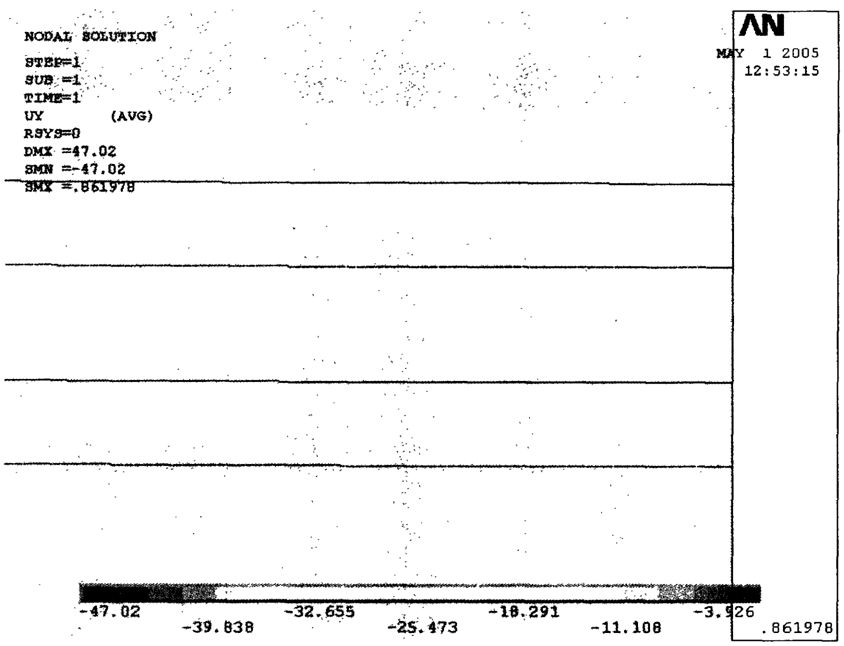

Figure C8: Y-displacement (μm) of copper 9μm thick under a weight of 0.04MPa. The vertical line farther right is the center of pressure.

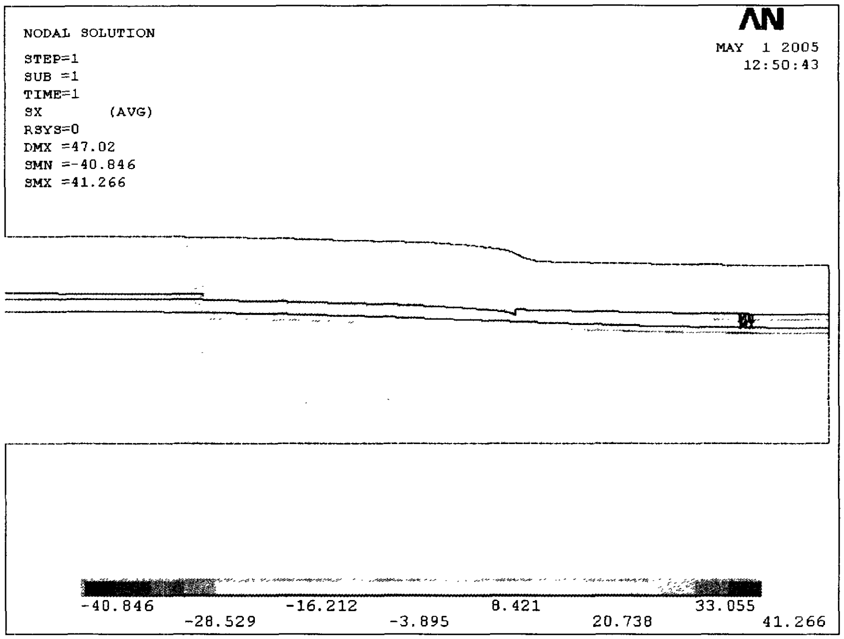

Figure C9: X-component of stress (MPa) of copper 9µm thick under a weight of 0.04MPa.

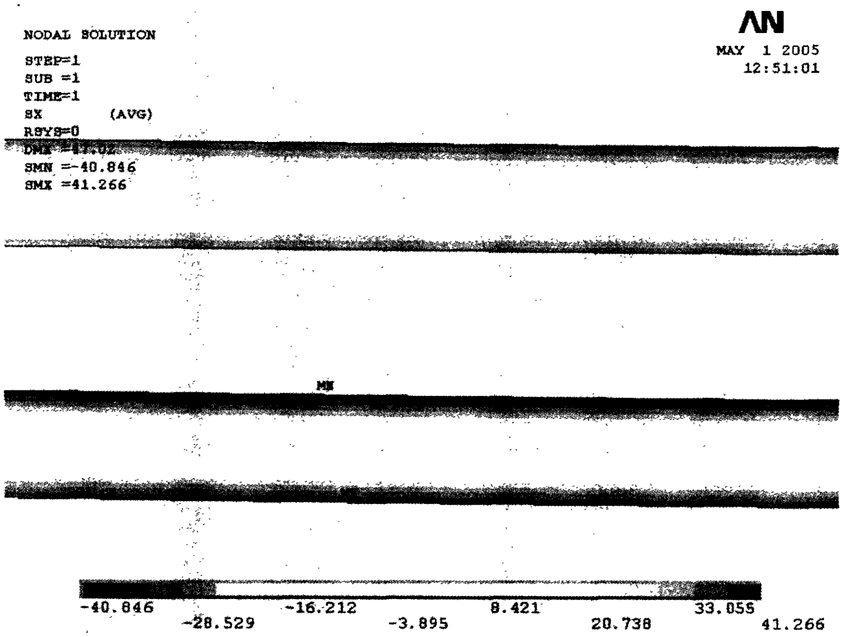

Figure C10: X-component of stress (MPa) of copper 9µm thick under a weight of 0.04MPa at the point of maximum and minimum stress.

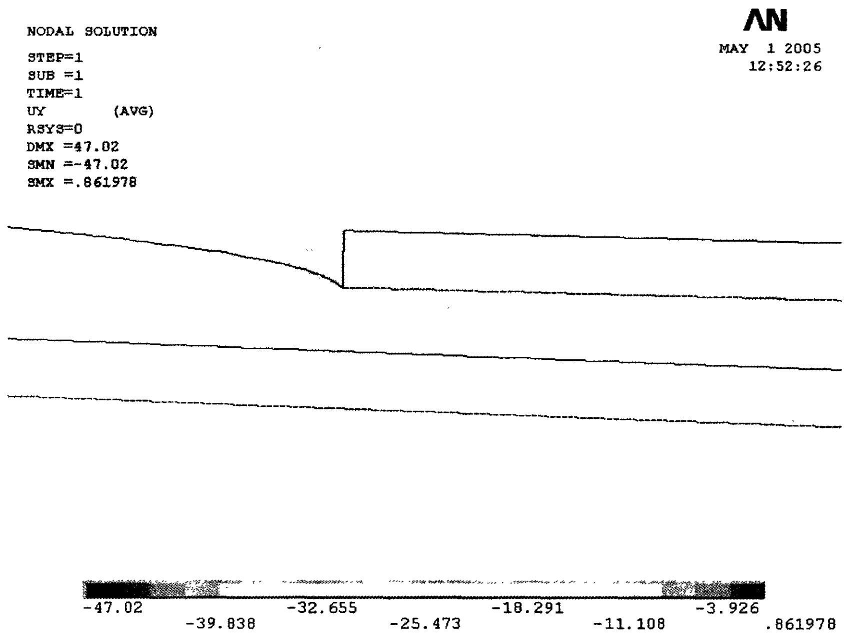

Figure C11: Y-displacement (μm) of copper 9μm thick under a weight of 0.04MPa, at the edge of the center copper node.

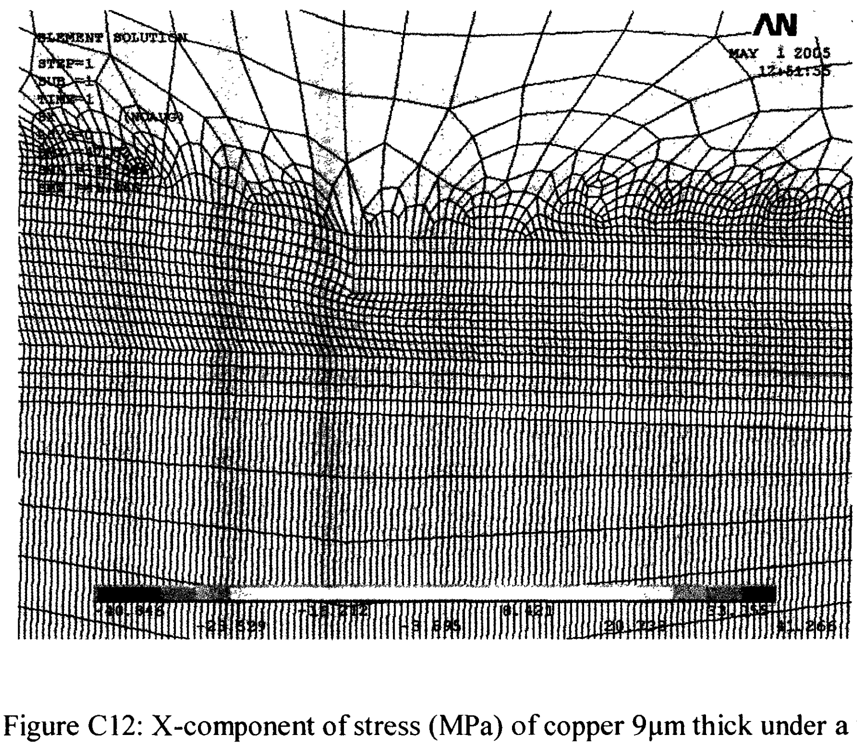

Figure C12: X-component of stress (MPa) of copper 9μm thick under a weight of 0.04MPa, at the edge of the center copper node, mesh showing.

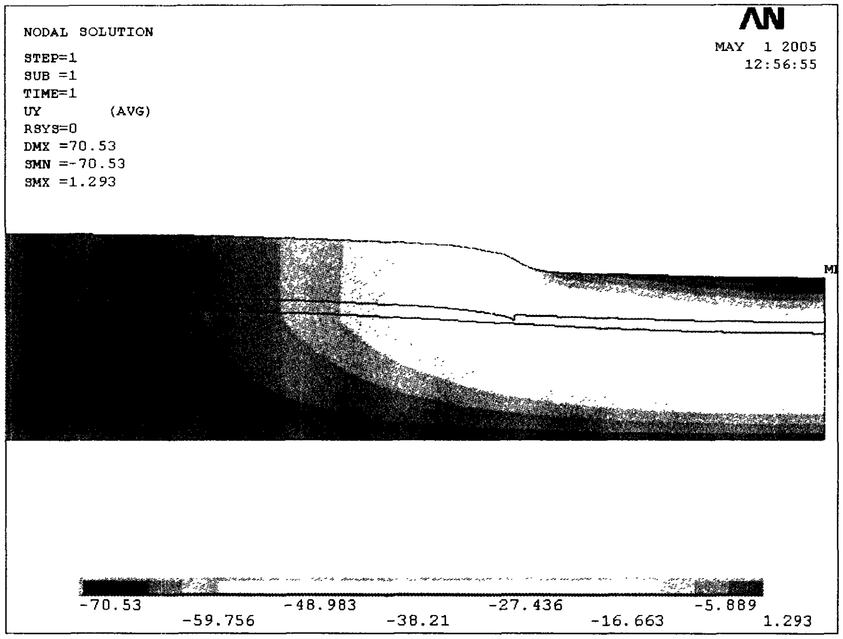

Figure C13: Y-displacement (μm) of copper 9μm thick under a pressure of 0.06a

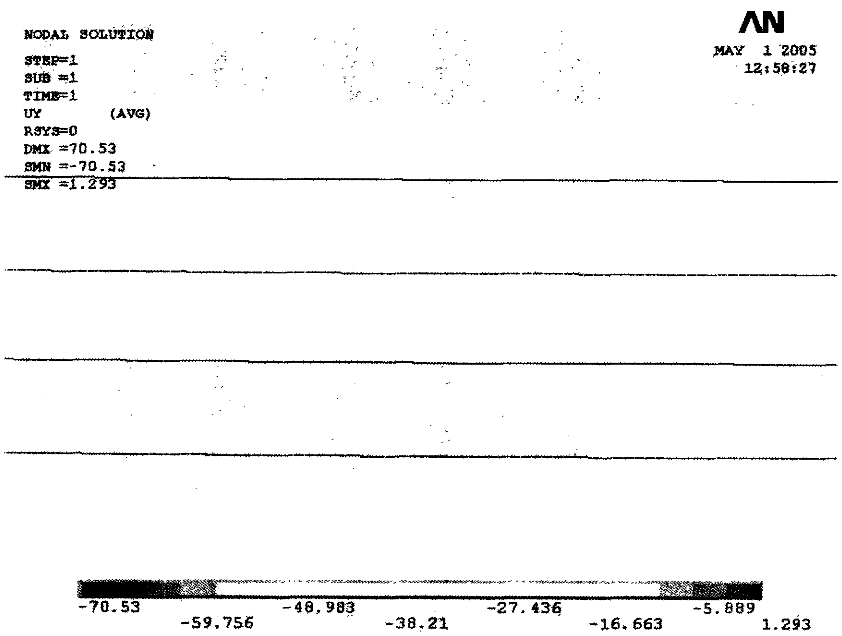

Figure C14: Y-displacement (μm) of copper 9μm thick under a weight of 0.06MPa. The vertical line farther right is the center of pressure.

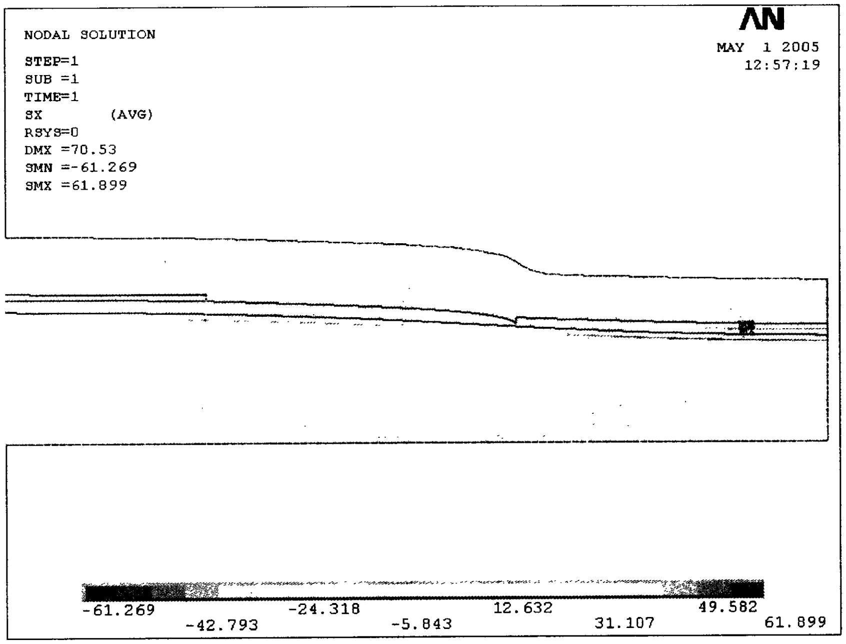

Figure C15: X-component of stress (MPa) of copper 9μm thick under a weight of 0.06MPa.

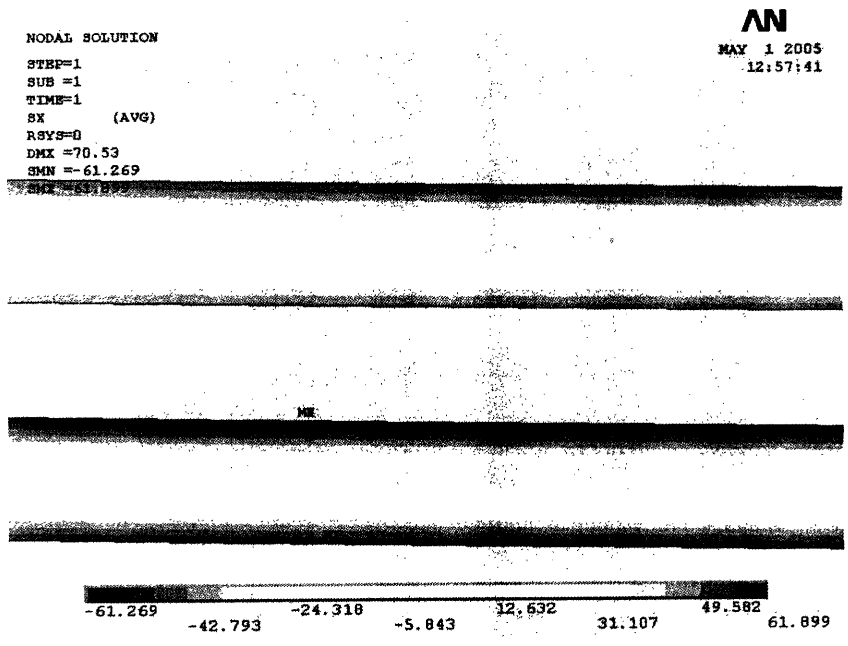

Figure C16: X-component of stress (MPa) of copper 9μm thick under a weight of 0.06MPa at the point of maximum and minimum stress.

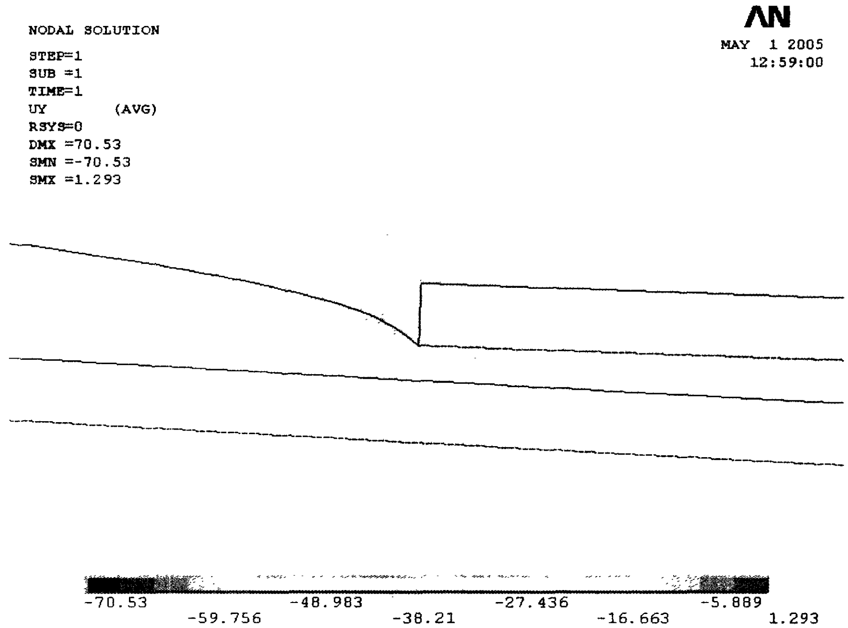

Figure C17: Y-displacement (μm) of copper 9μm thick under a weight of 0.06MPa, at the edge of the center copper node.

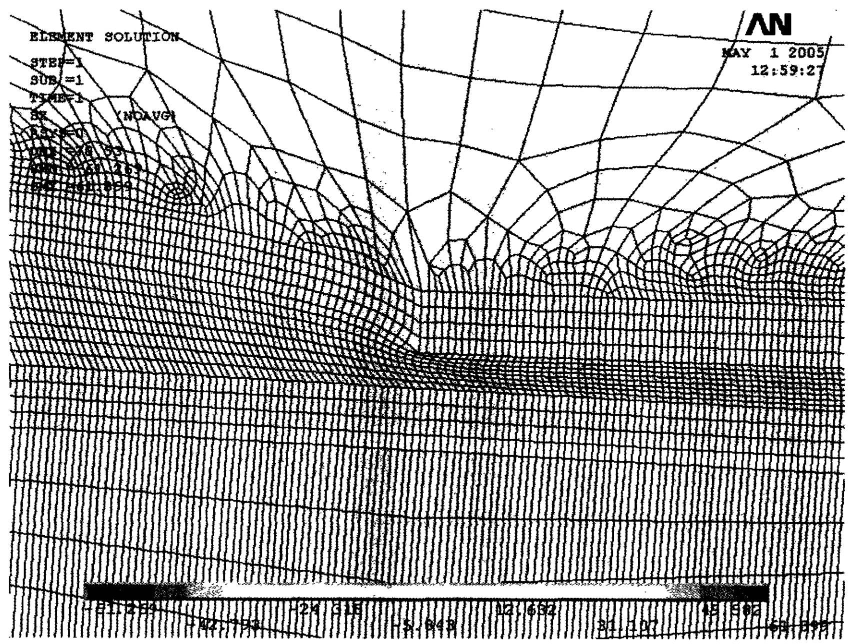

Figure C18: X-component of stress (MPa) of copper 9μm thick under a weight of 0.06MPa, at the edge of the center copper node, mesh showing.

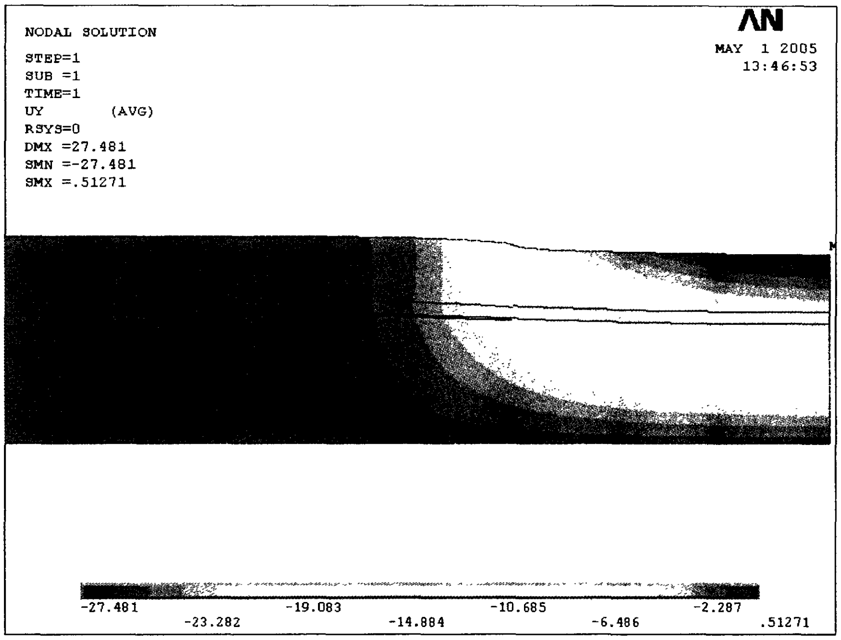

Figure C19: Y-displacement (μm) of copper 3μm thick under a pressure of 0.02MPa.

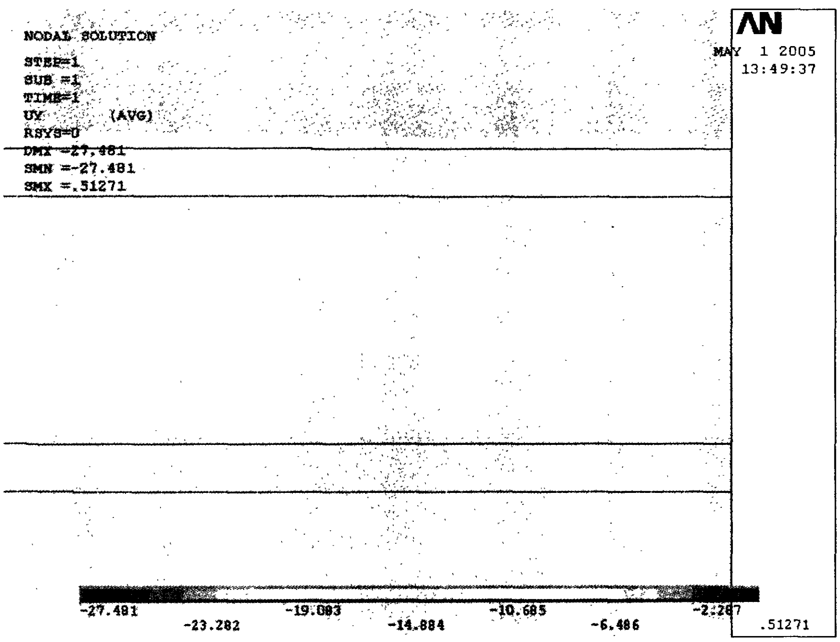

Figure C20: Y-displacement (μm) of copper 3μm thick under a weight of 0.02MPa. The vertical line farther right is the center of pressure.

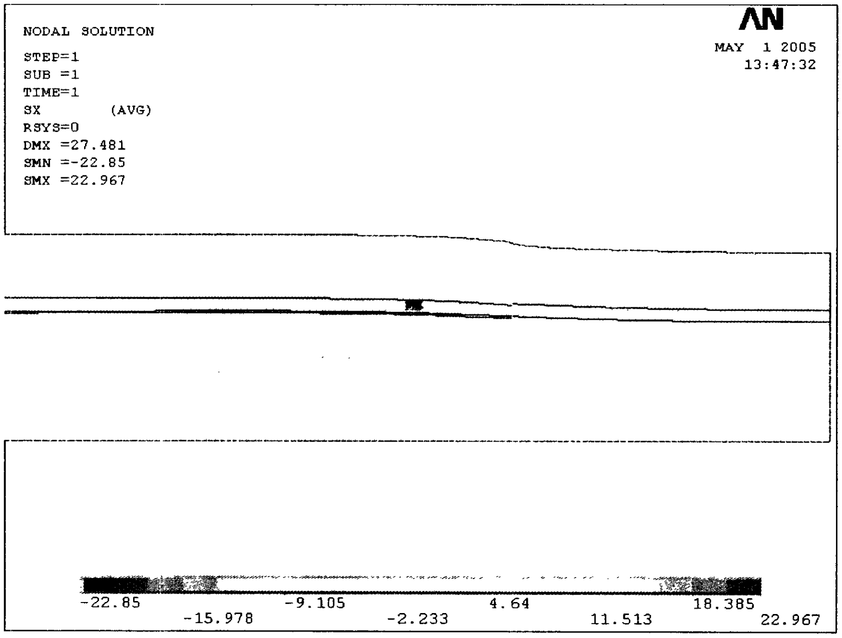

Figure C21: X-component of stress (MPa) of copper 3μm thick under a weight of 0.02MPa.

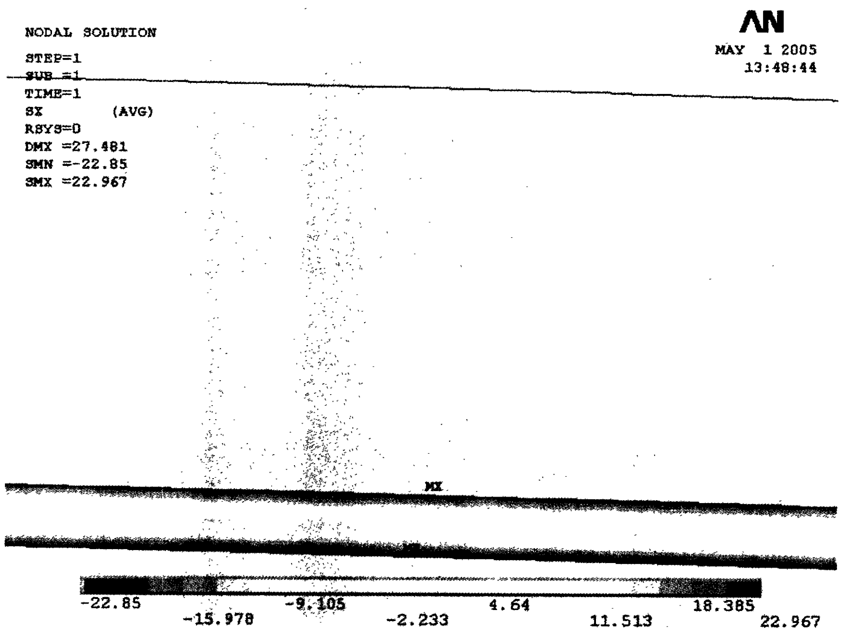

Figure C22: X-component of stress (MPa) of copper 3μm thick under a weight of 0.02MPa at the point of maximum and minimum stress.

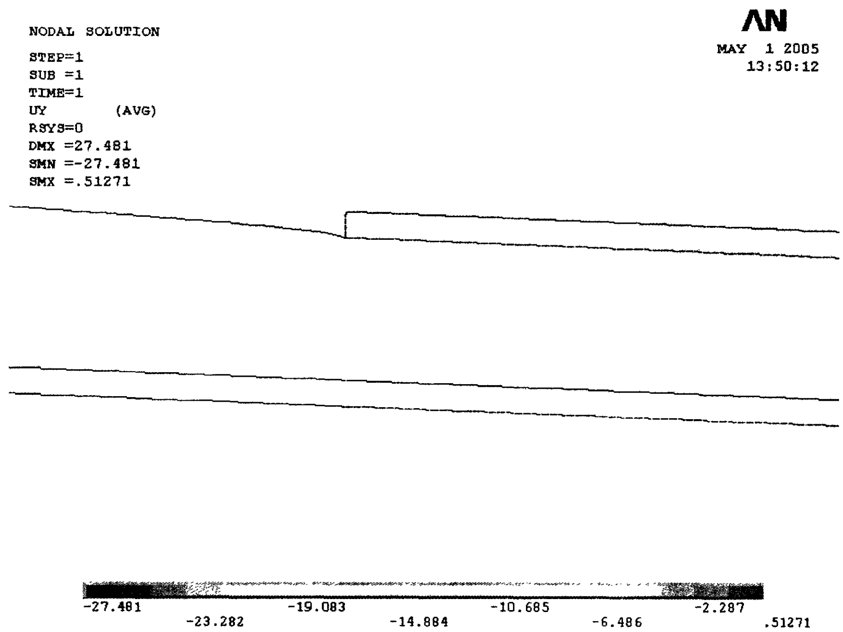

Figure C23: Y-displacement (μm) of copper 3m thick under a weight of 0.02MPa, at the edge of the center copper node.

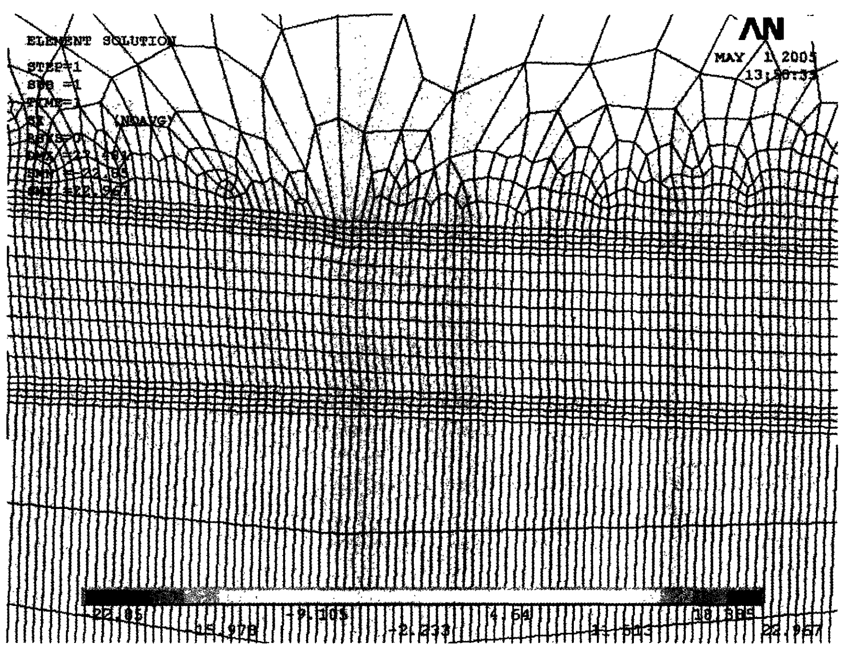

Figure C24: X-component of stress (MPa) of copper 3μm thick under a weight of 0.02MPa, at the edge of the center copper node, mesh showing.

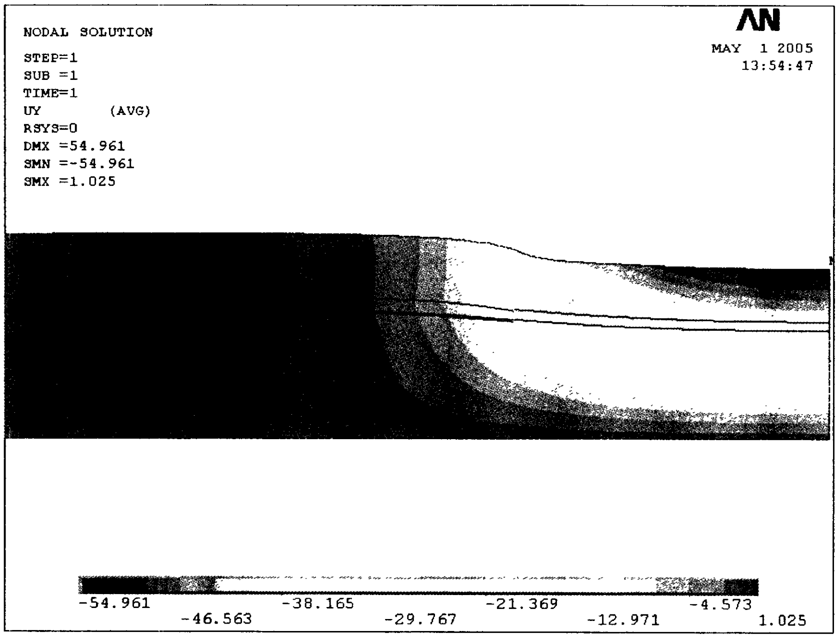

Figure C25: Y-displacement (μm) of copper 3μm thick under a pressure of 0.04MPa.

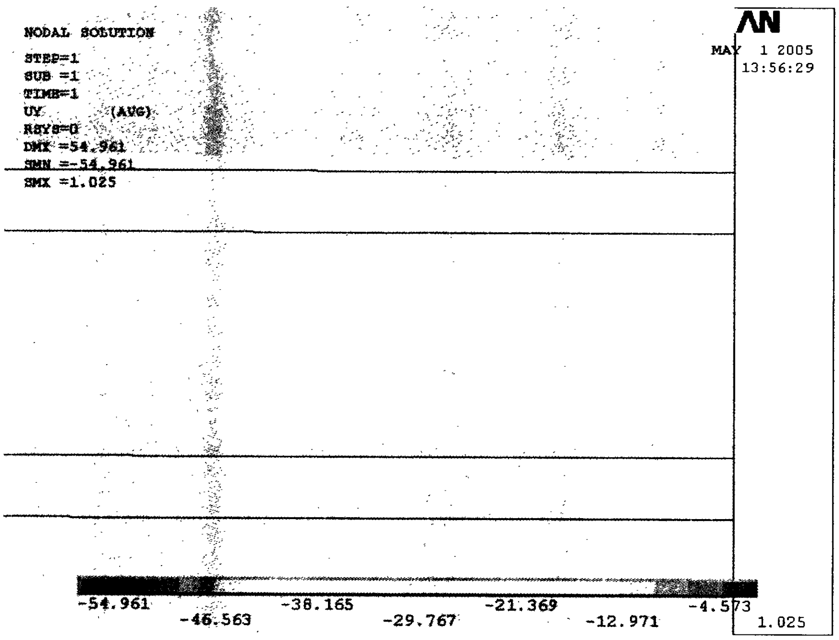

Figure C26: Y-displacement (μm) of copper 3μm thick under a weight of 0.04MPa. The vertical line farther right is the center of pressure.

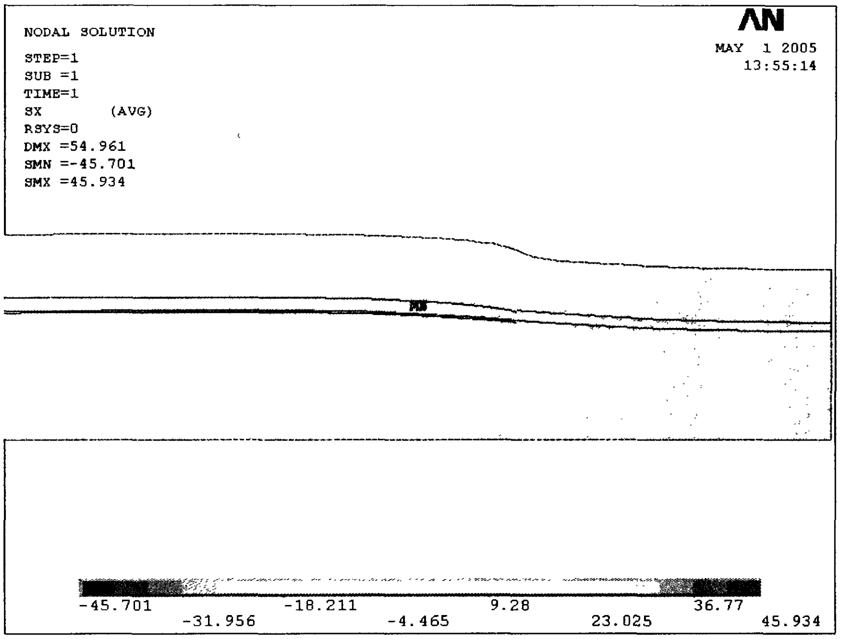

Figure C27: X-component of stress (MPa) of copper 3μm thick under a weight of 0.04MPa.

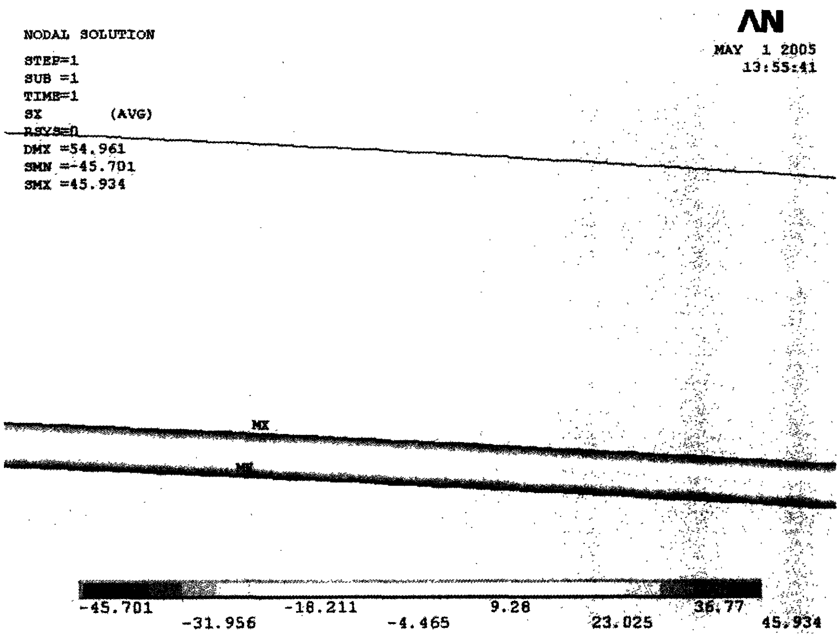

Figure C28: X-component of stress (MPa) of copper 3μm thick under a weight of 0.04MPa at the point of maximum and minimum stress.

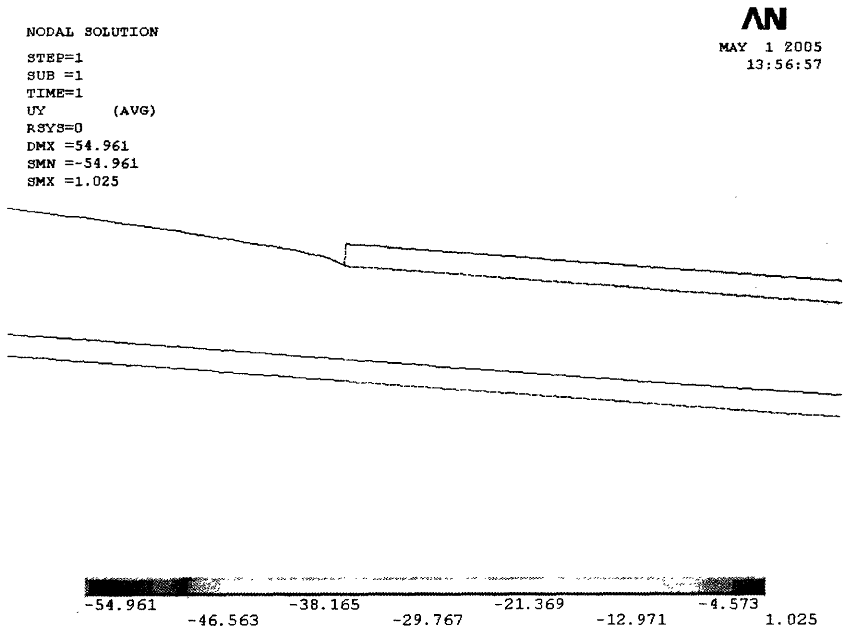

Figure C29: Y-displacement (μm) of copper 3μm thick under a weight of 0.04MPa, at the edge of the center copper node.

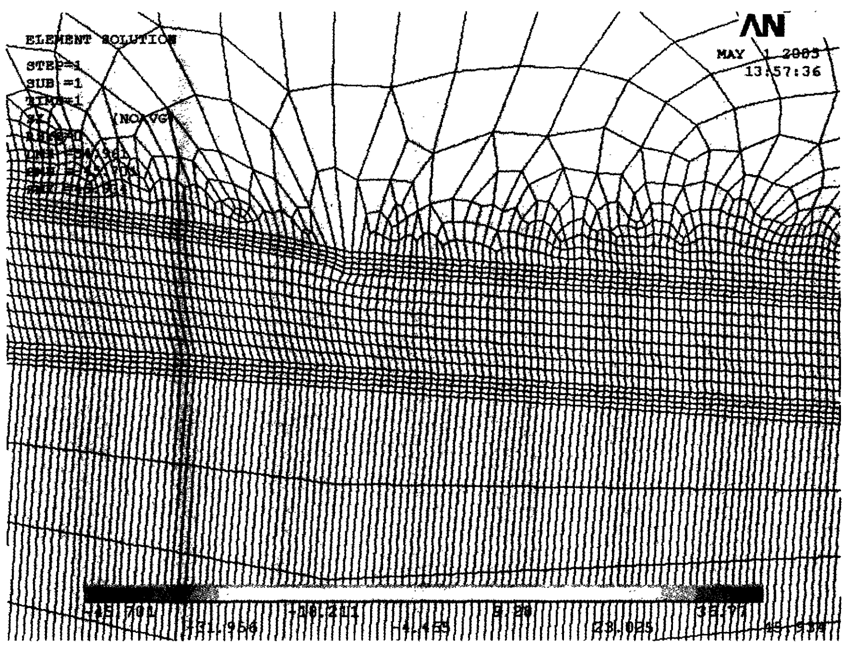

Figure C30: X-component of stress (MPa) of copper 3μm thick under a weight of 0.04MPa, at the edge of the center copper node, mesh showing.

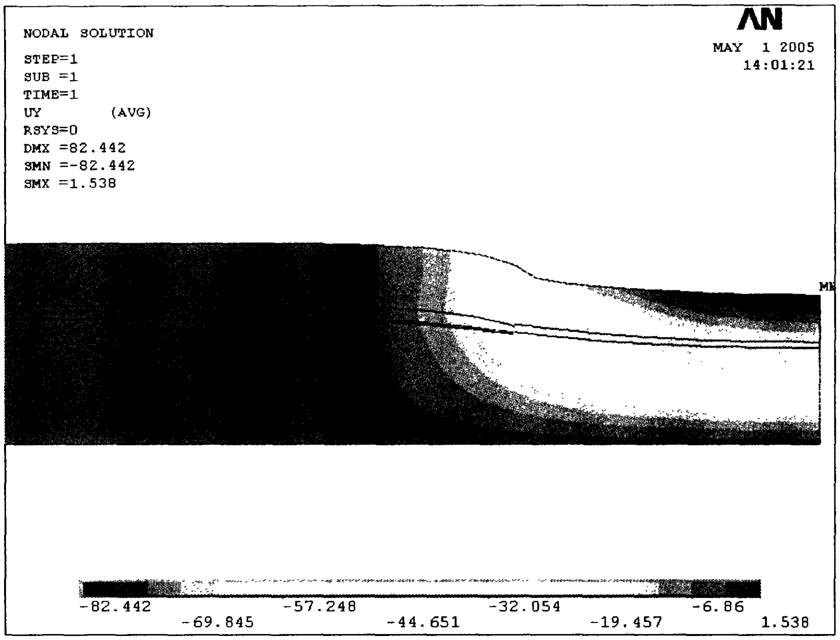

Figure C31: Y-displacement (μm) of copper 3μm thick under a pressure of 0.06MPa.

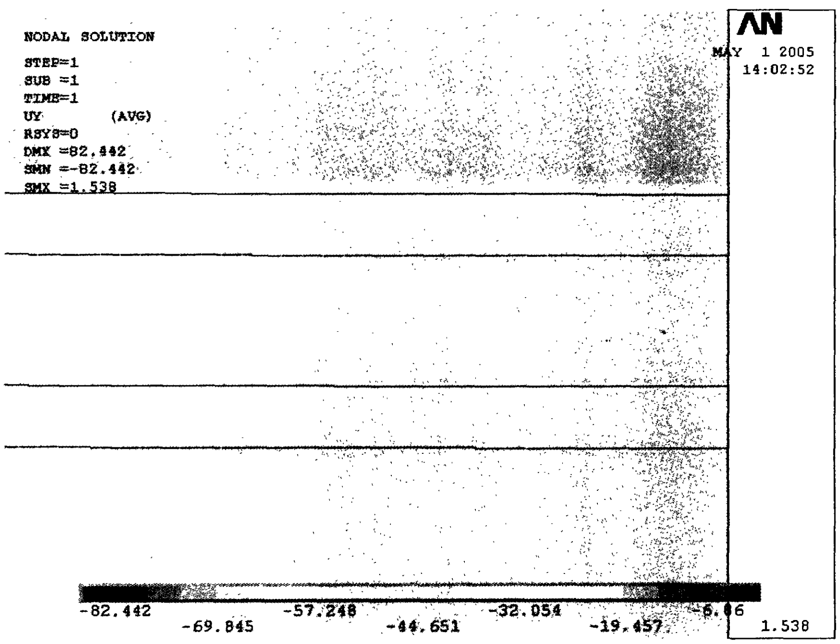

Figure C32: Y-displacement (μm) of copper 3μm thick under a weight of 0.06MPa. The vertical line farther right is the center of pressure.

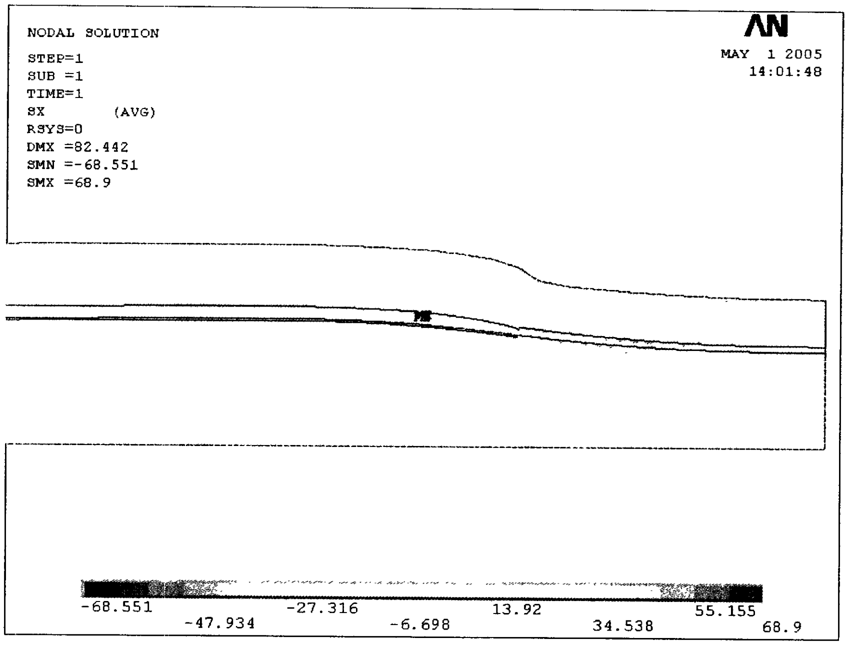

Figure C33: X-component of stress (MPa) of copper 3μm thick under a weight of 0.06MPa.

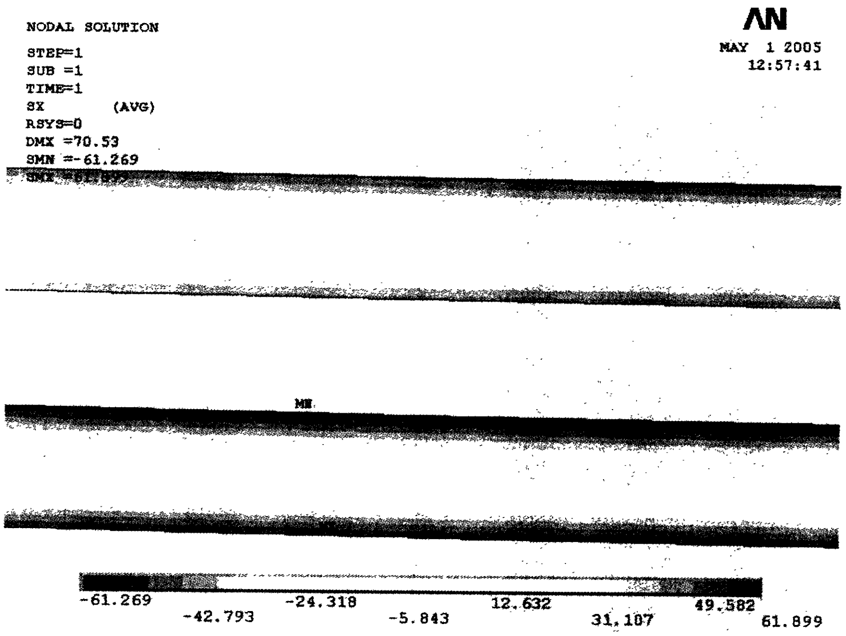

Figure C34: X-component of stress (MPa) of copper 3μm thick under a weight of 0.06MPa at the point of maximum and minimum stress.

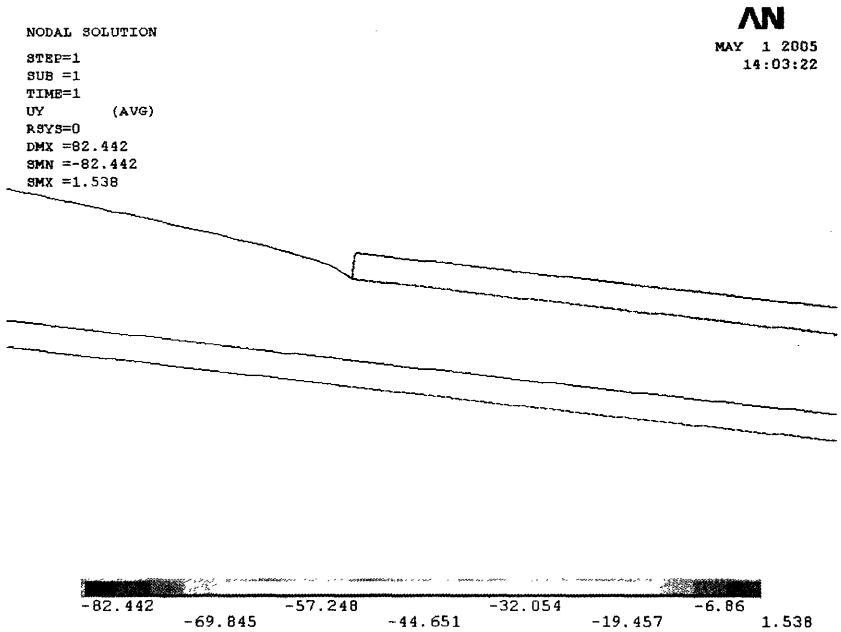

Figure C35: Y-displacement (μm) of copper 3μm thick under a weight of 0.06MPa, at the edge of the center copper node.

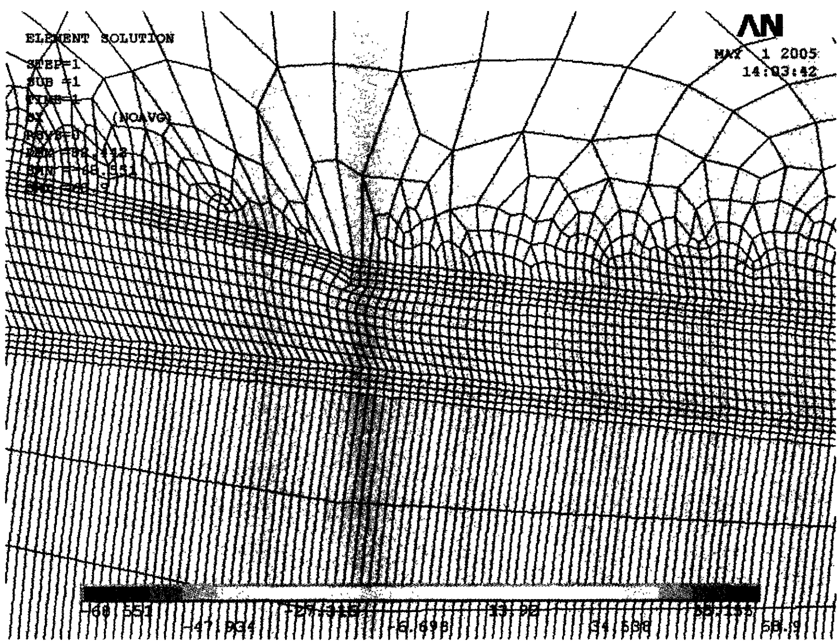

Figure C36: X-component of stress (MPa) of copper 3μm thick under a weight of 0.06MPa, at the edge of the center copper node, mesh showing.

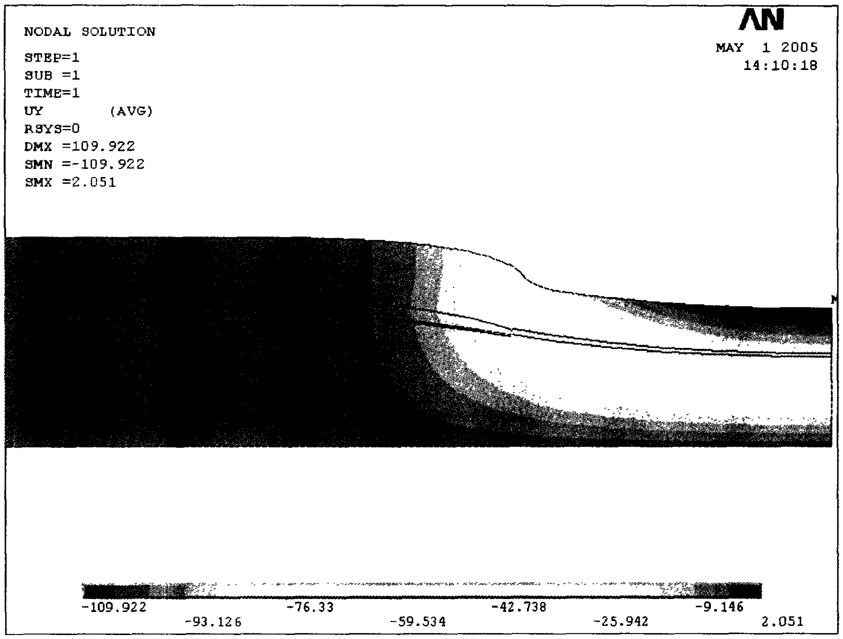

Figure C37: Y-displacement (µm) of copper 3µm thick under a pressure of 0.08MPa.

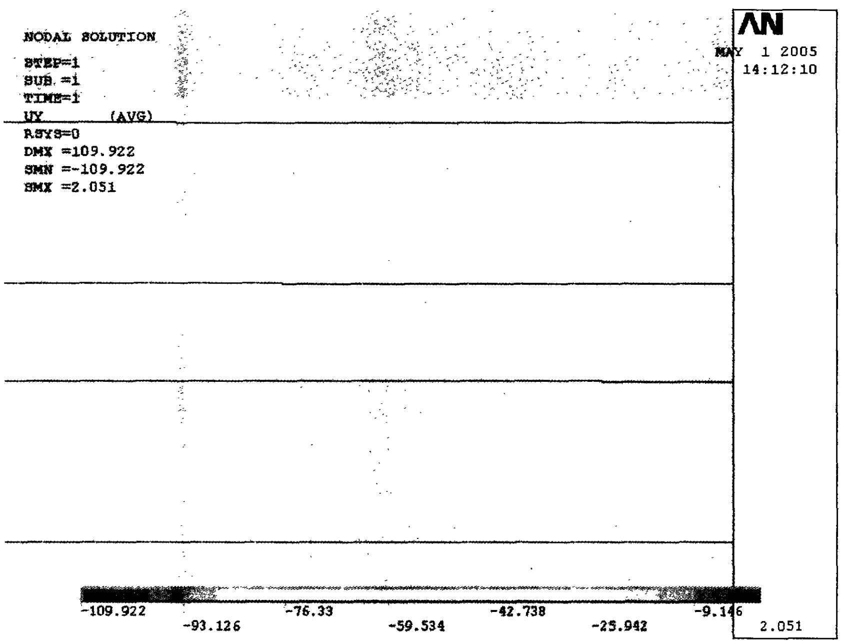

Figure C38: Y-displacement (µm) of copper 3µm thick under a weight of 0.08MPa. The vertical line farther right is the center of pressure.

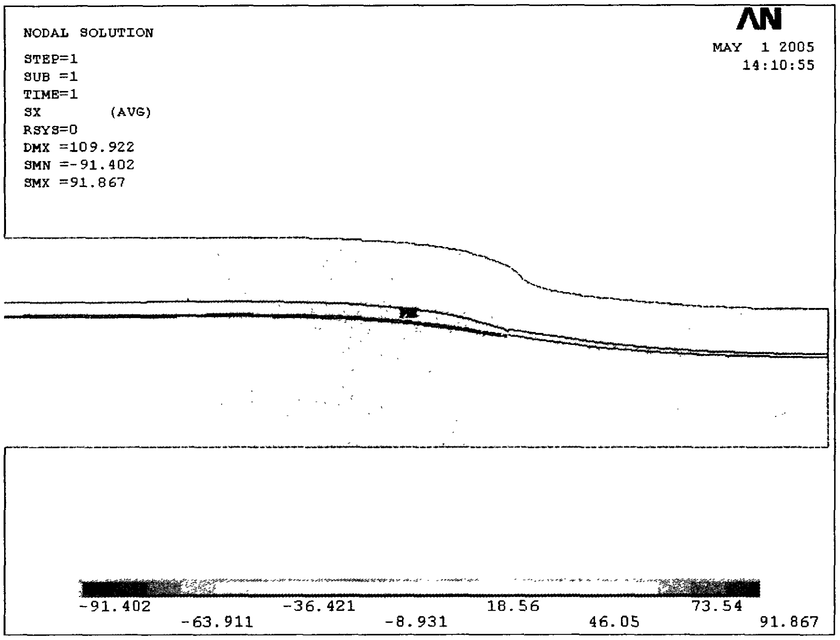

Figure C39: X-component of stress (MPa) of copper 3μm thick under a weight of 0.08MPa.

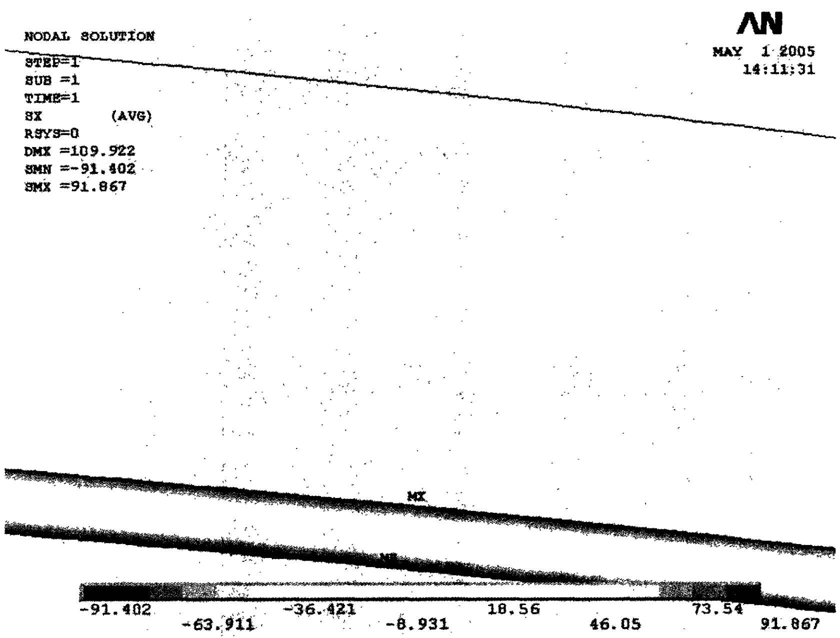

Figure C40: X-component of stress (MPa) of copper 3μm thick under a weight of 0.08MPa at the point of maximum and minimum stress.

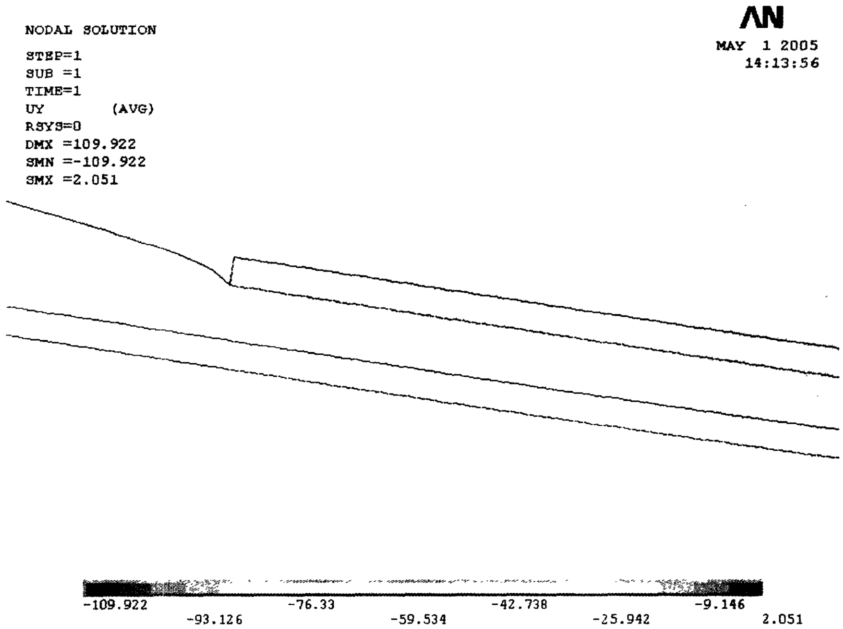

Figure C41: Y-displacement (μm) of copper 3μm thick under a weight of 0.08MPa, at the edge of the center copper node.

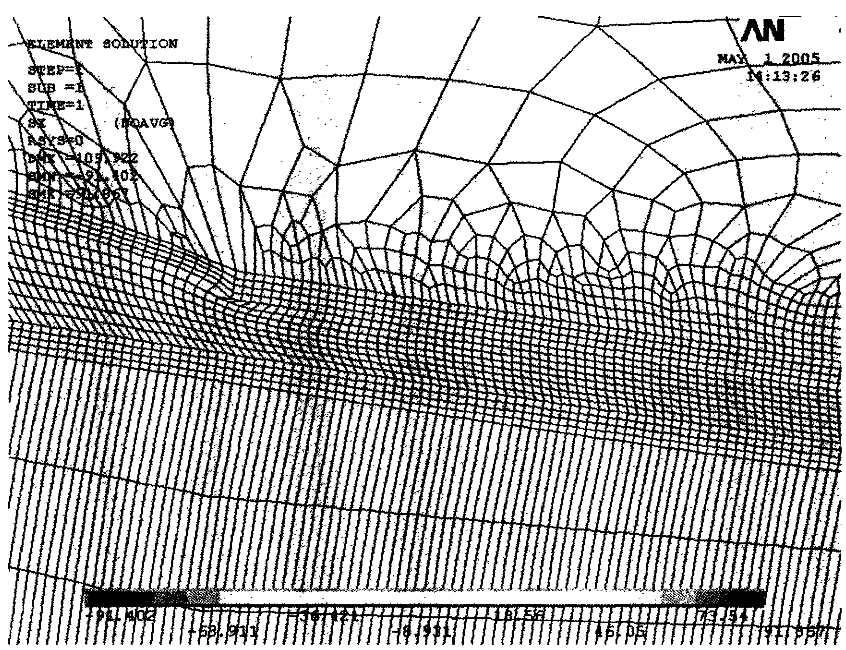

Figure C42: X-component of stress (MPa) of copper 3μm thick under a weight of 0.08MPa, at the edge of the center copper node, mesh showing.

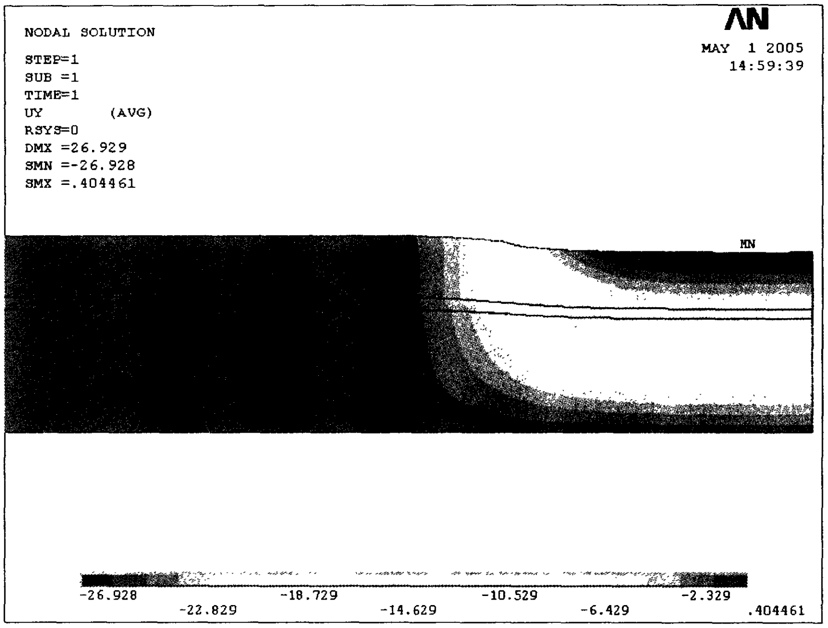

Figure C43: Y-displacement (μm) of copper 0.5μm thick under a pressure of 0.02MPa.

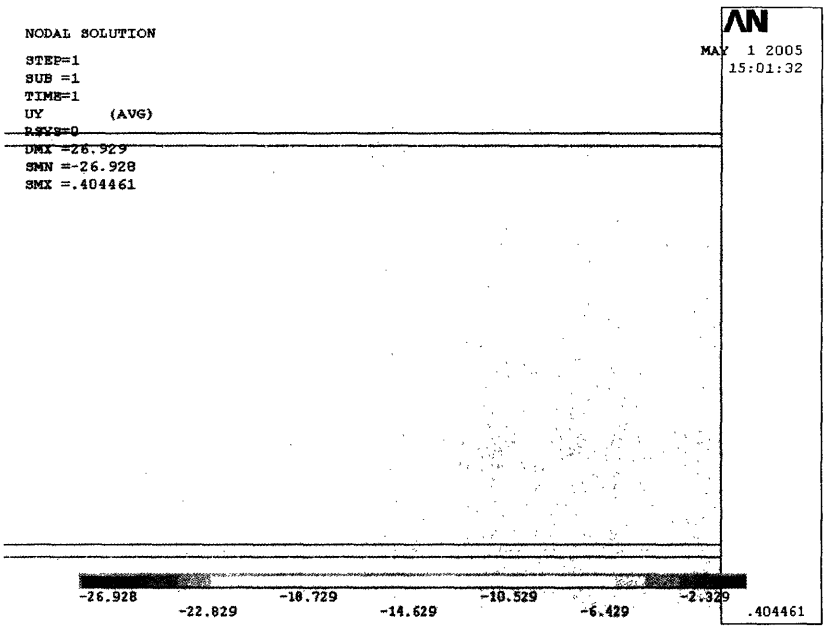

Figure C44: Y-displacement (μm) of copper 0.5μm thick under a weight of 0.02MPa. The vertical line farther right is the center of pressure.

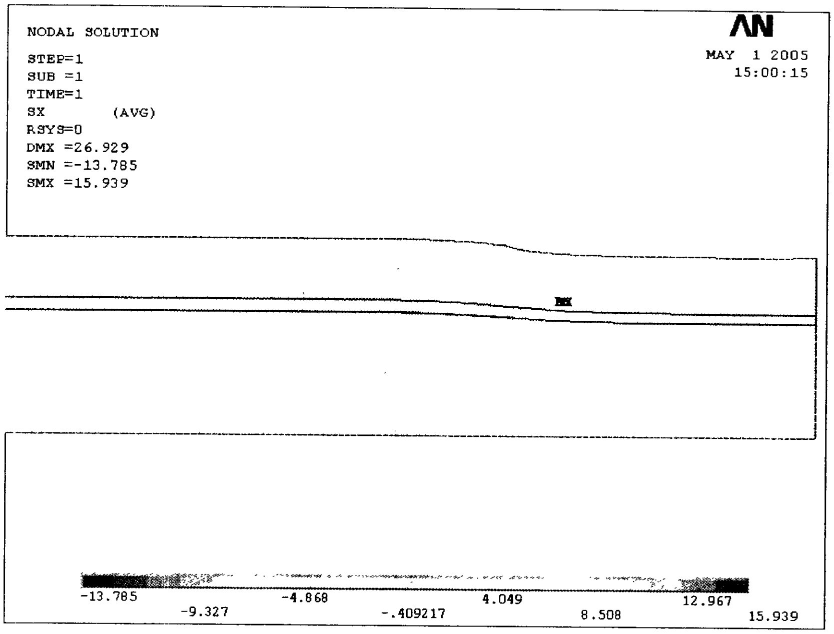

Figure C45: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.02MPa.

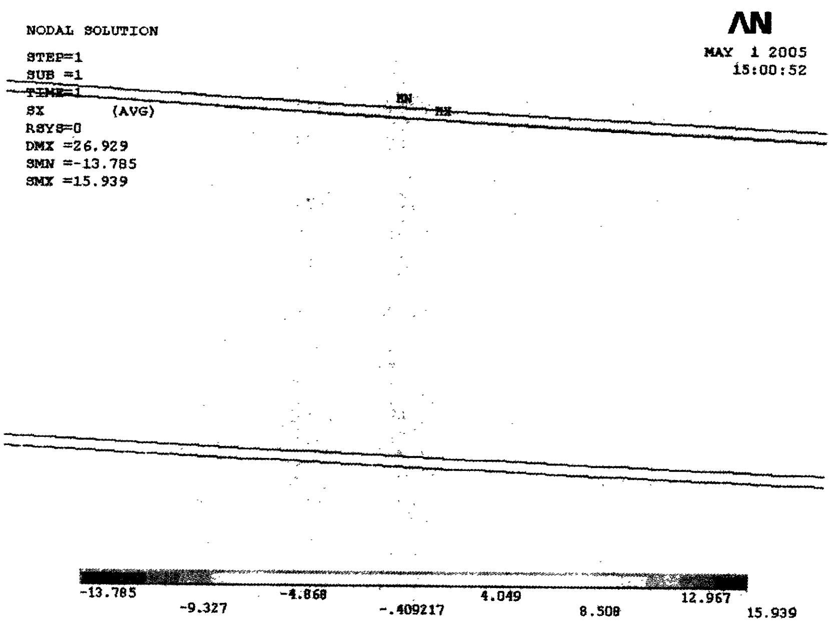

Figure C46: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.02MPa at the point of maximum and minimum stress.

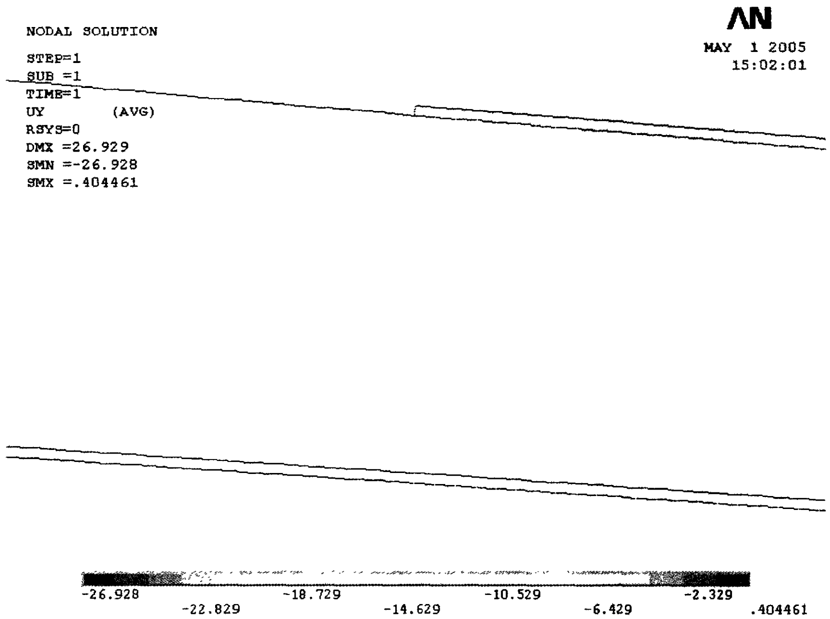

Figure C47: Y-displacement (μm) of copper 0.5μm thick under a weight of 0.02MPa, at the edge of the center copper node.

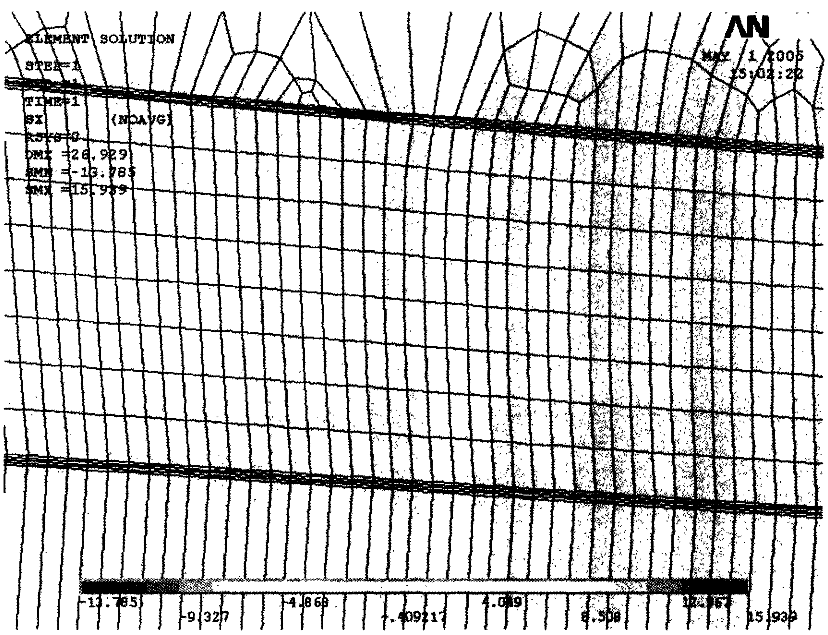

Figure C48: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.02MPa, at the edge of the center copper node, mesh showing.

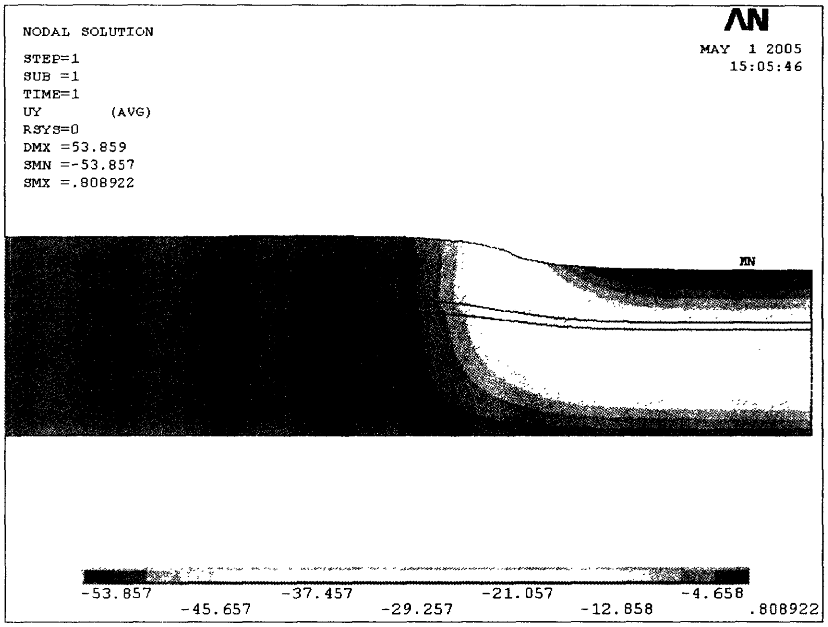

Figure C49: Y-displacement (μm) of copper 0.5μm thick under a pressure of 0.04MPa.

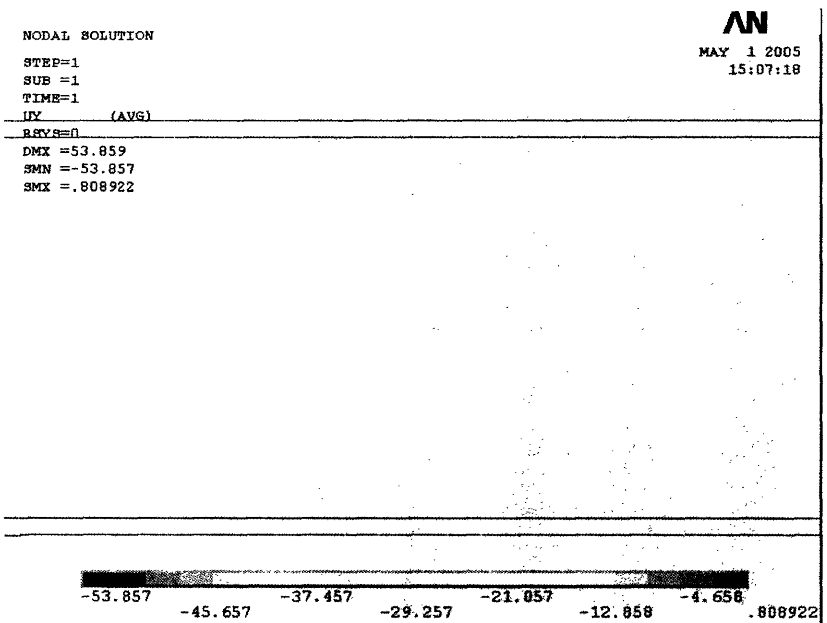

Figure C50: Y-displacement (μm) of copper 0.5μm thick under a weight of 0.04MPa. The vertical line farther right is the center of pressure.

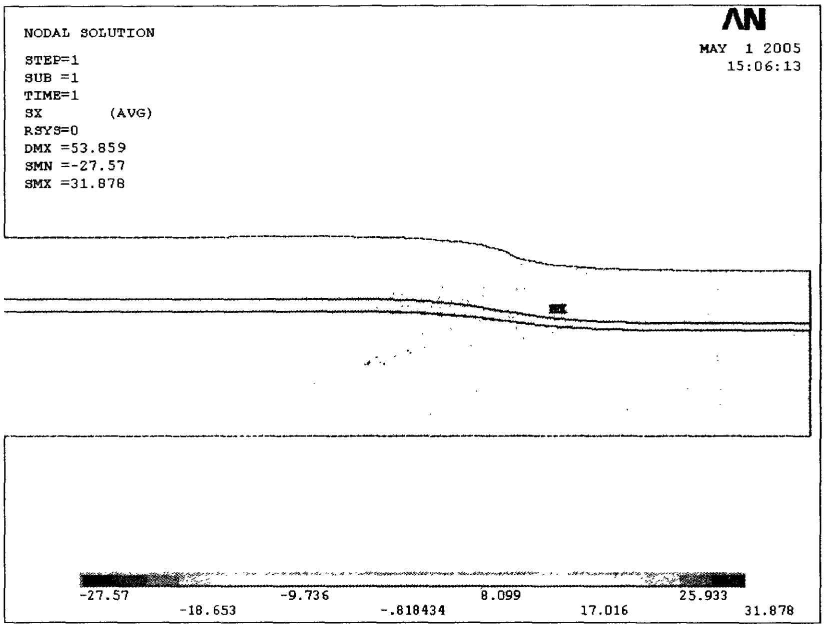

Figure C51: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.04MPa.

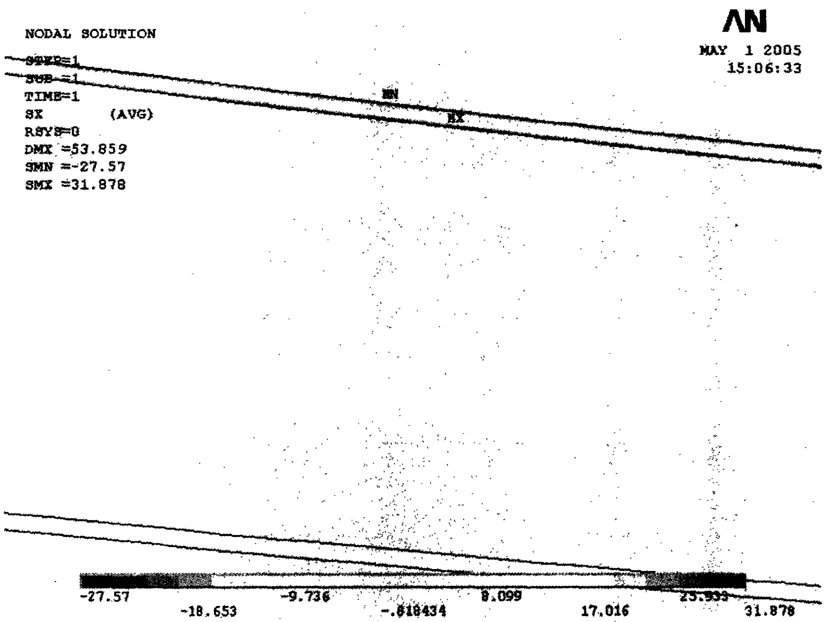

Figure C52: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.04MPa at the point of maximum and minimum stress.

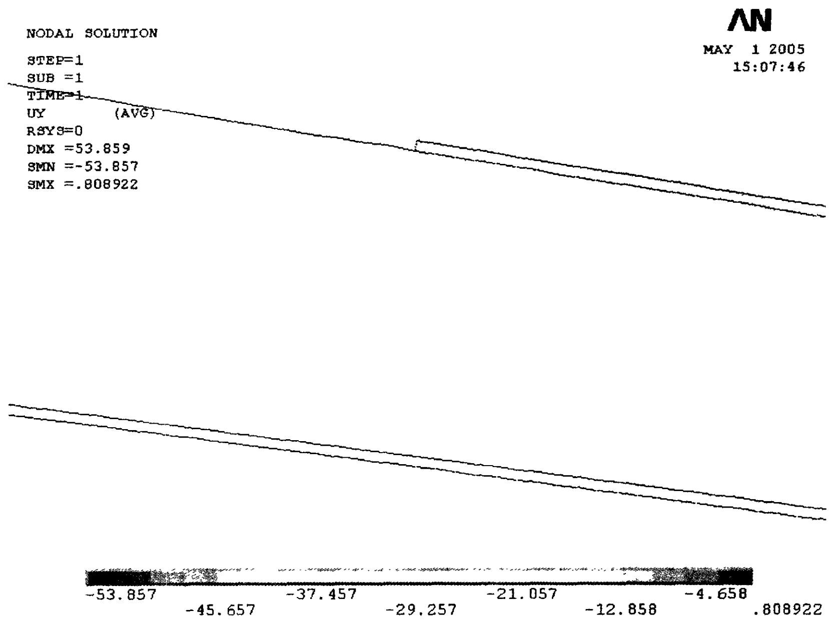

Figure C53: Y-displacement (μm) of copper 0.5μm thick under a weight of 0.04MPa, at the edge of the center copper node.

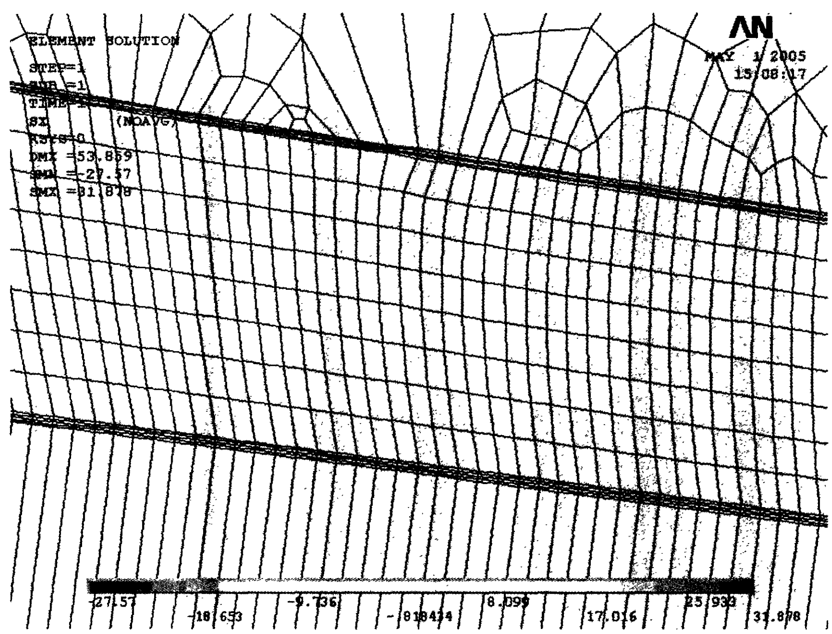

Figure C54: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.04MPa, at the edge of the center copper node, mesh showing.

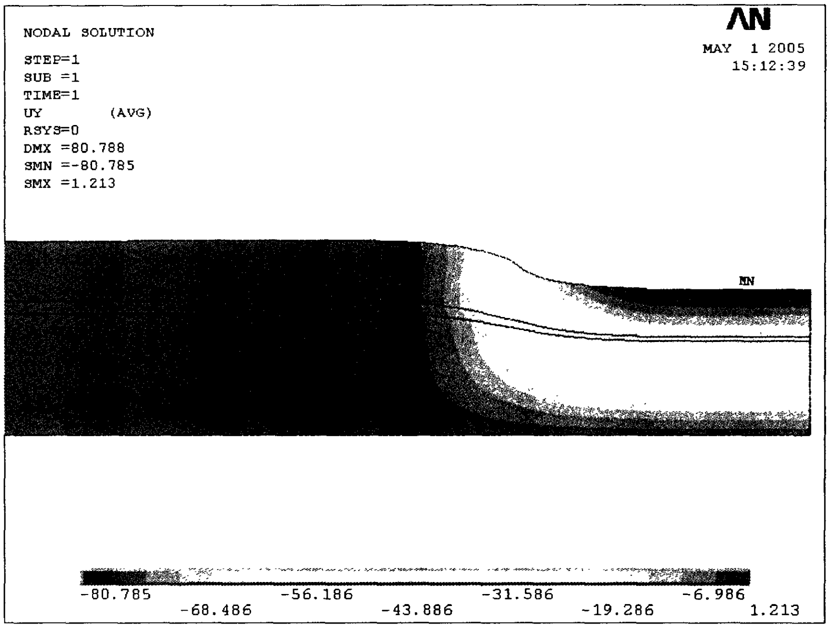

Figure C55: Y-displacement (μm) of copper 0.5μm thick under a pressure of 0.06MPa.

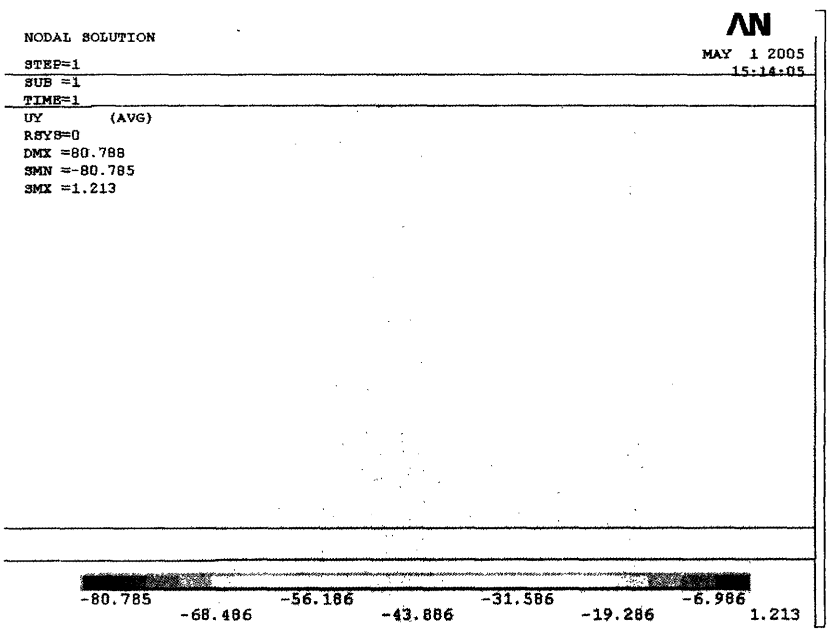

Figure C56: Y-displacement (μm) of copper 0.5μm thick under a weight of 0.06MPa. The vertical line farther right is the center of pressure.

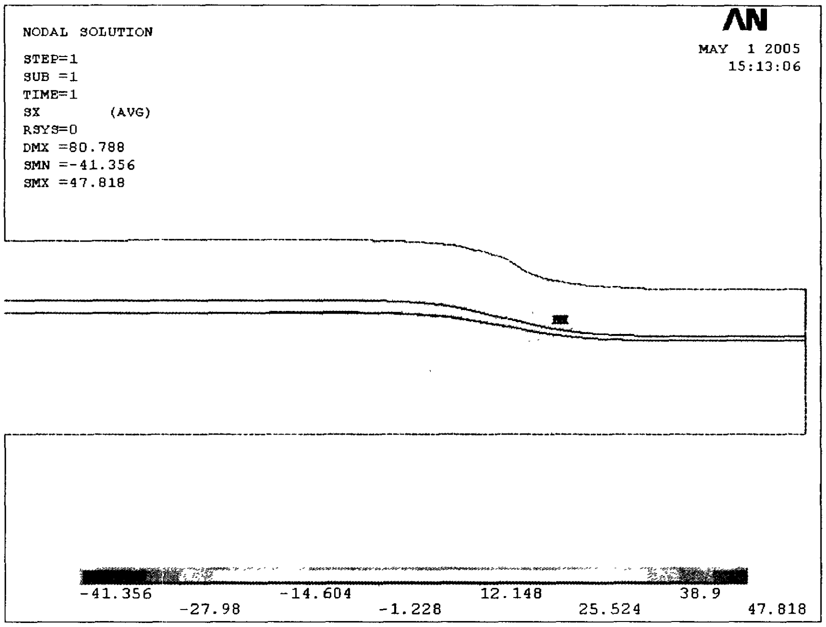

Figure C57: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.06MPa.

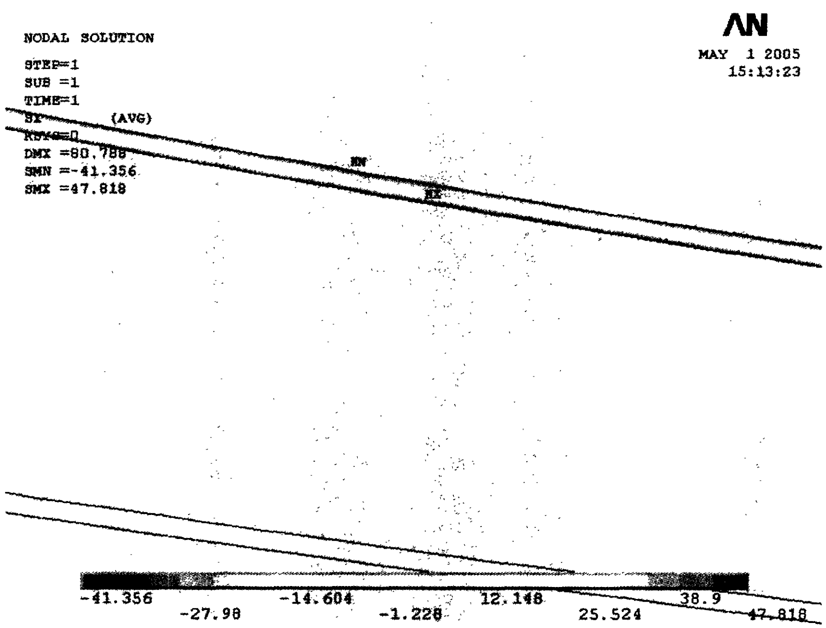

Figure C58: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.06MPa at the point of maximum and minimum stress.

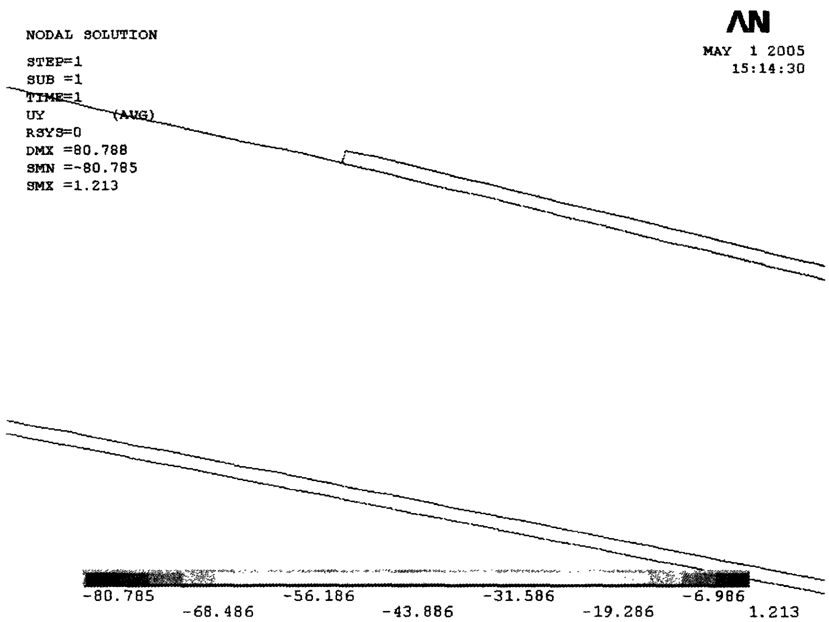

Figure C59: Y-displacement (μm) of copper 0.5μm thick under a weight of 0.06MPa, at the edge of the center copper node.

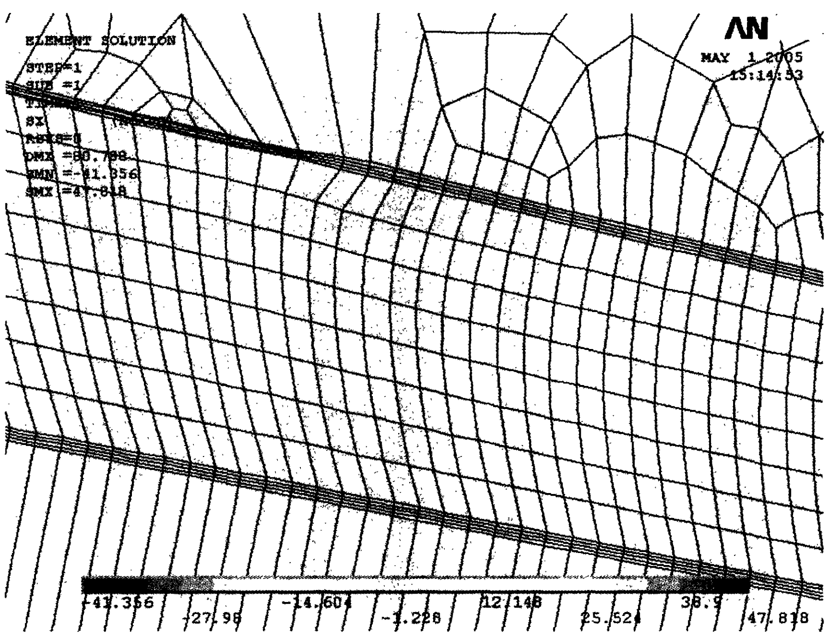

Figure C60: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.06MPa, at the edge of the center copper node, mesh showing.

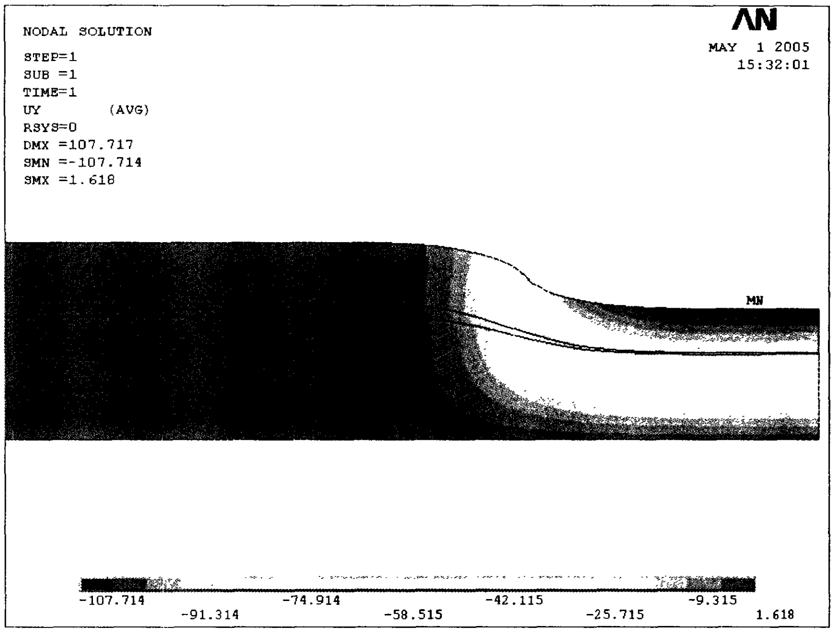

Figure C61: Y-displacement (μm) of copper 0.5μm thick under a pressure of 0.08MPa.

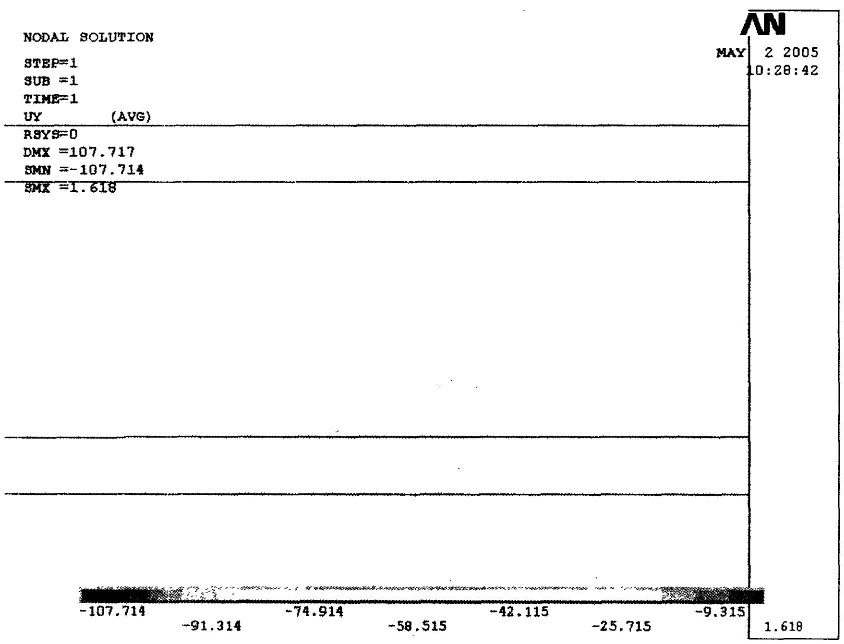

Figure C62: Y-displacement (μm) of copper 0.5μm thick under a weight of 0.08MPa. The vertical line farther right is the center of pressure.

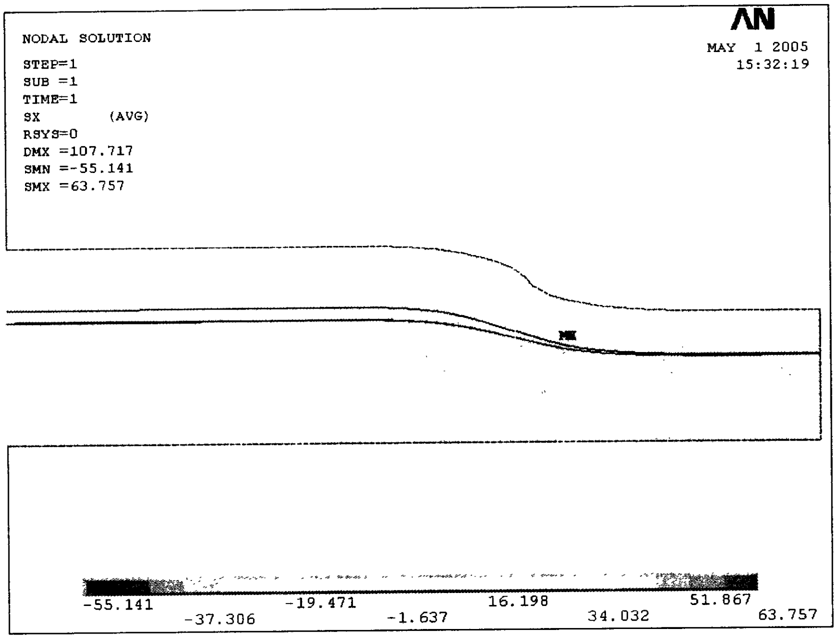

Figure C63: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.08MPa.

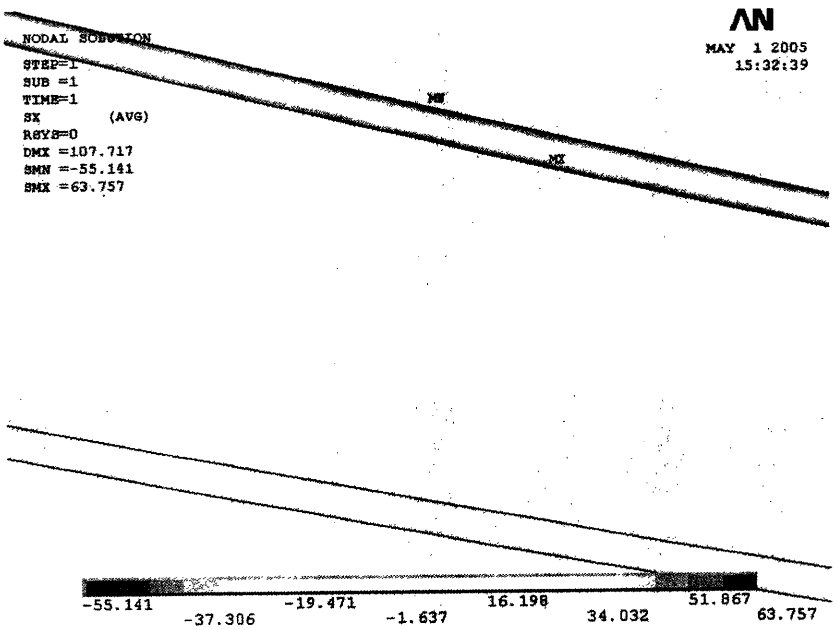

Figure C64: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.08MPa at the point of maximum and minimum stress.

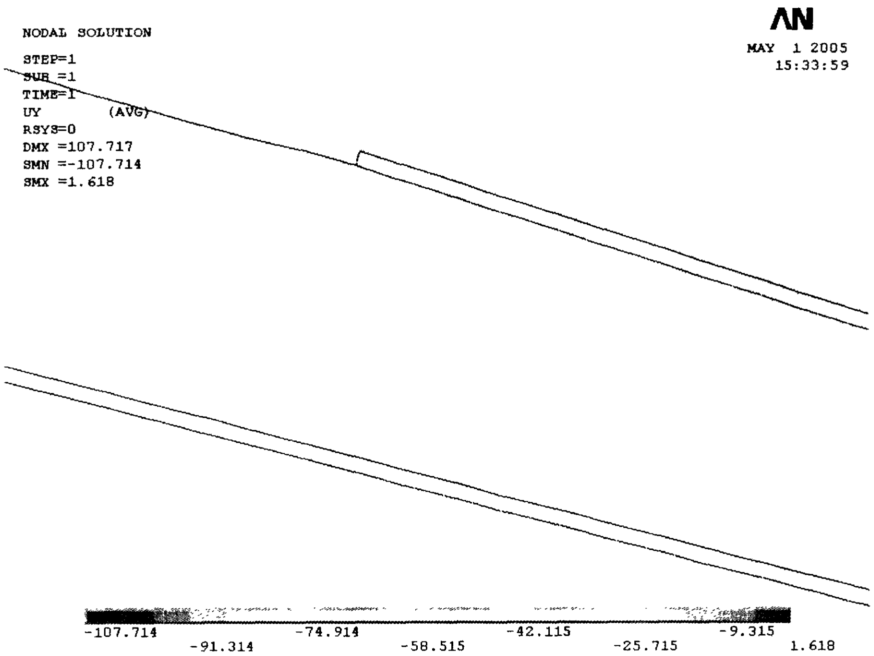

Figure C65: Y-displacement (μm) of copper 0.5μm thick under a weight of 0.08MPa, at the edge of the center copper node.

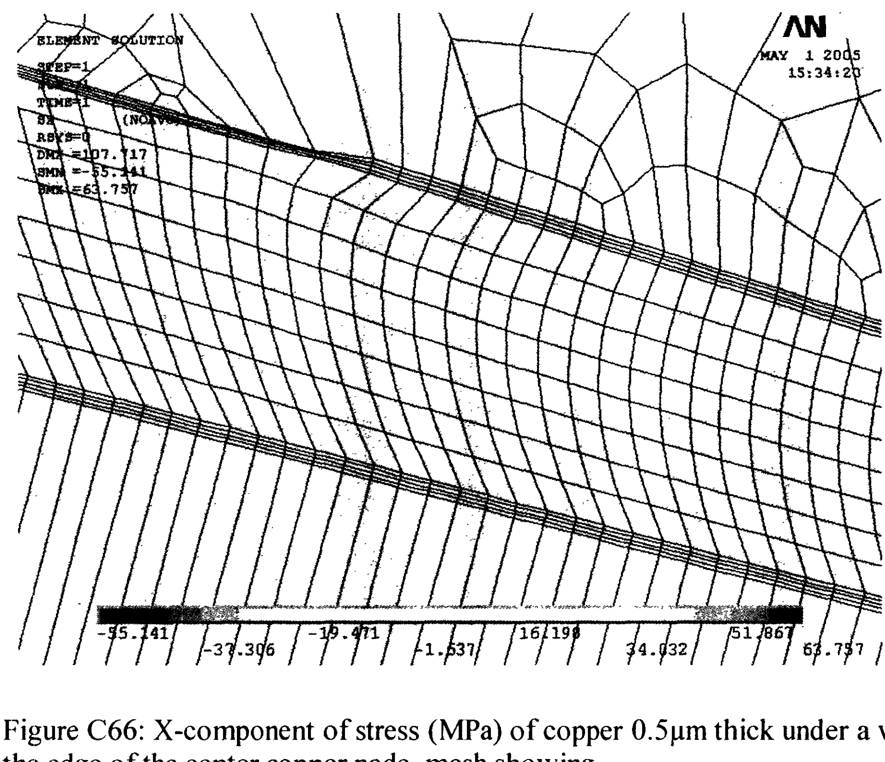

Figure C66: X-component of stress (MPa) of copper 0.5μm thick under a weight of 0.08MPa, at the edge of the center copper node, mesh showing.

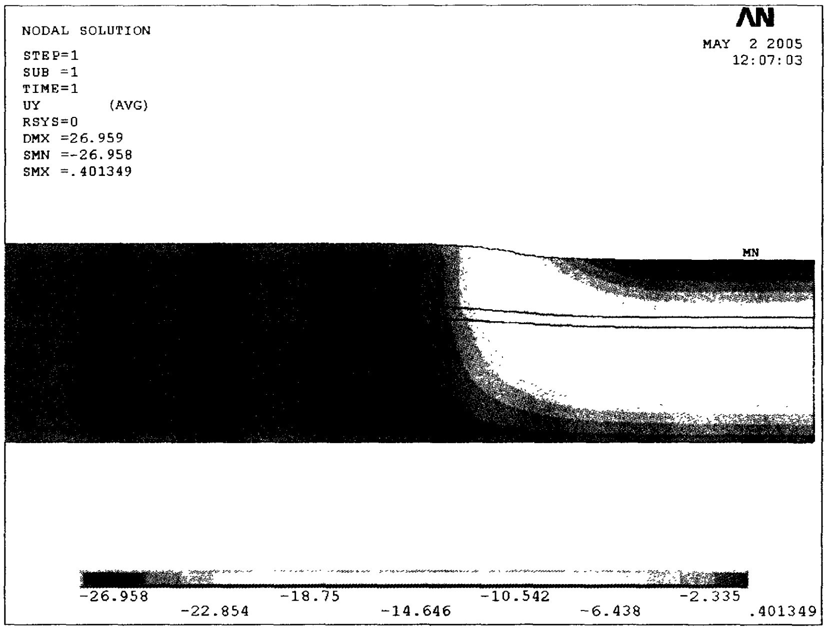

Figure C67: Y-displacement (μm) of copper 0.3μm thick under a pressure of 0.02MPa.

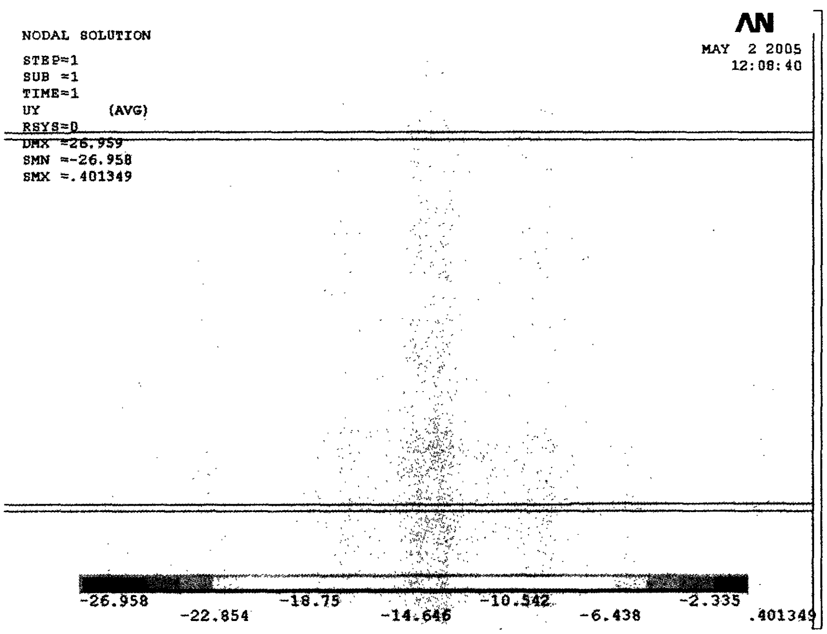

Figure C68: Y-displacement (μm) of copper 0.3μm thick under a weight of 0.02MPa. The vertical line farther right is the center of pressure.

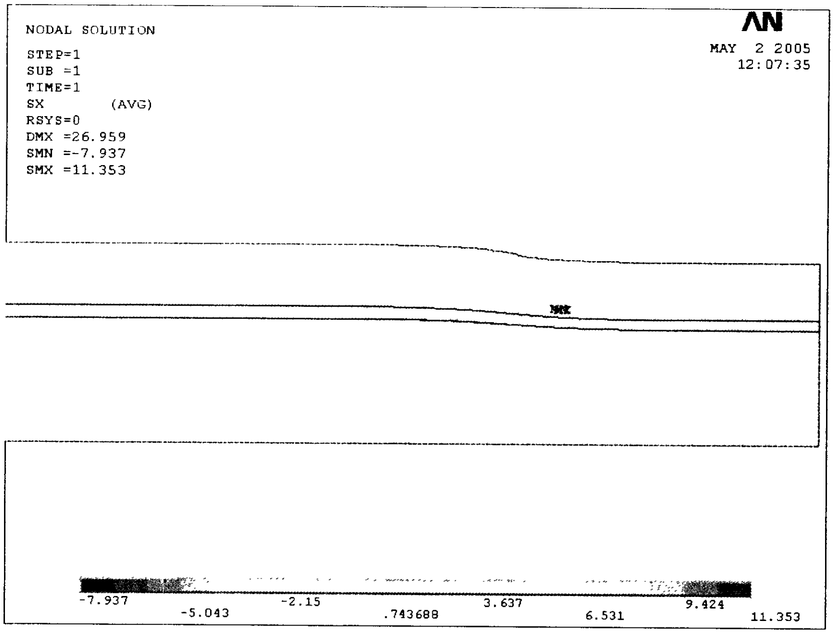

Figure C69: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.02MPa.

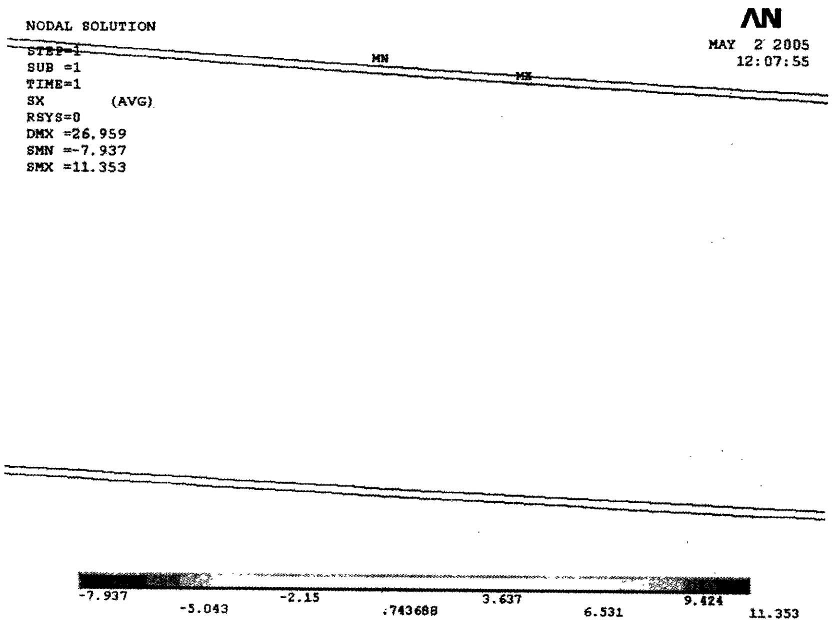

Figure C70: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.02MPa at the point of maximum and minimum stress.

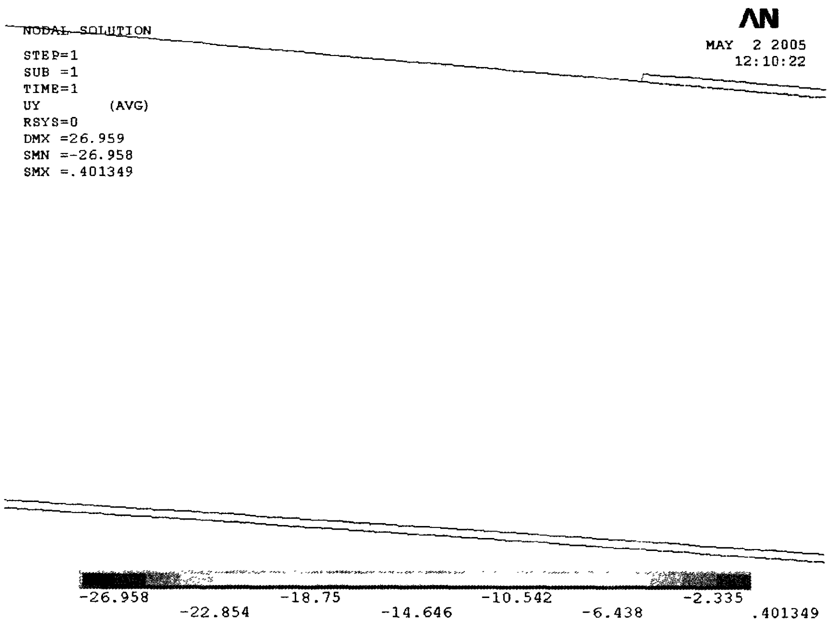

Figure C71: Y-displacement (μm) of copper 0.3μm thick under a weight of 0.02MPa, at the edge of the center copper node.

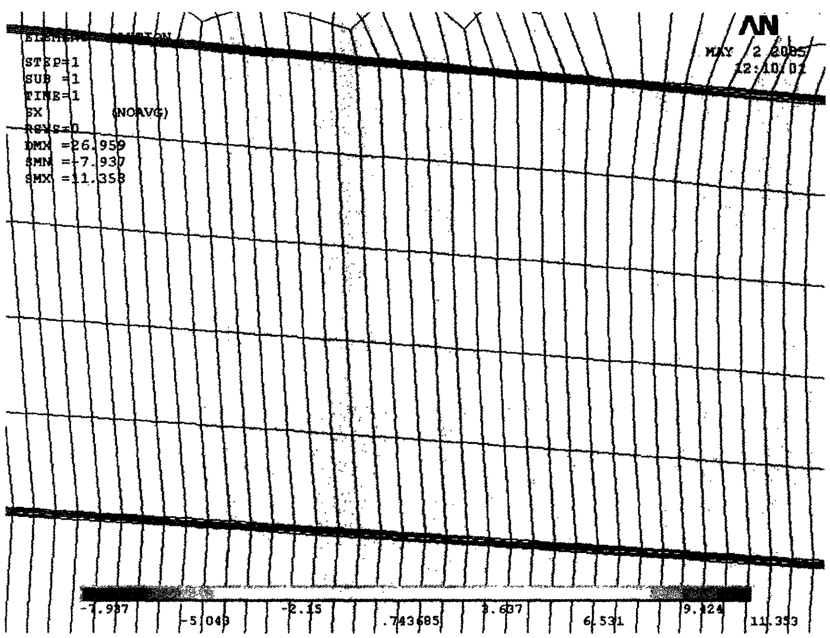

Figure C72: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.02MPa, at the edge of the center copper node, mesh showing.

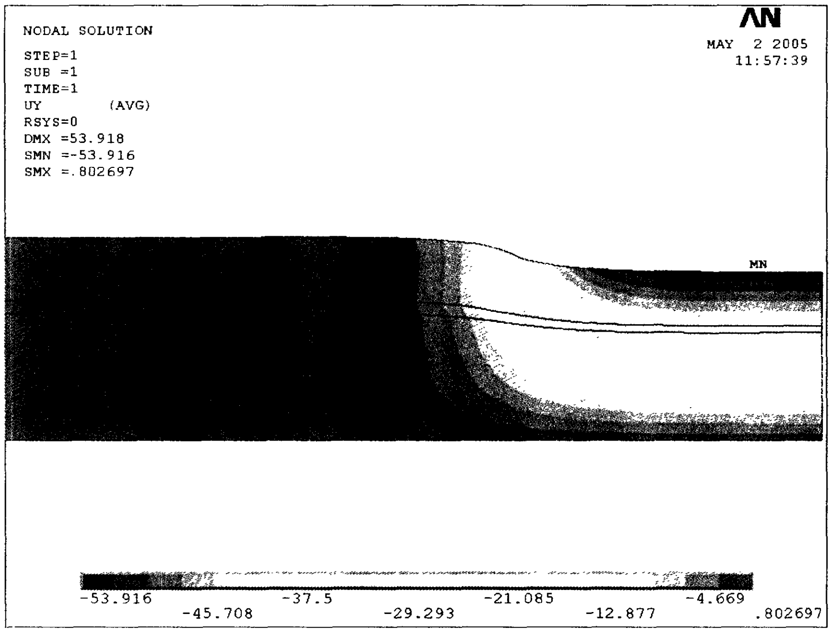

Figure C73: Y-displacement (μm) of copper 0.3μm thick under a pressure of 0.04MPa.

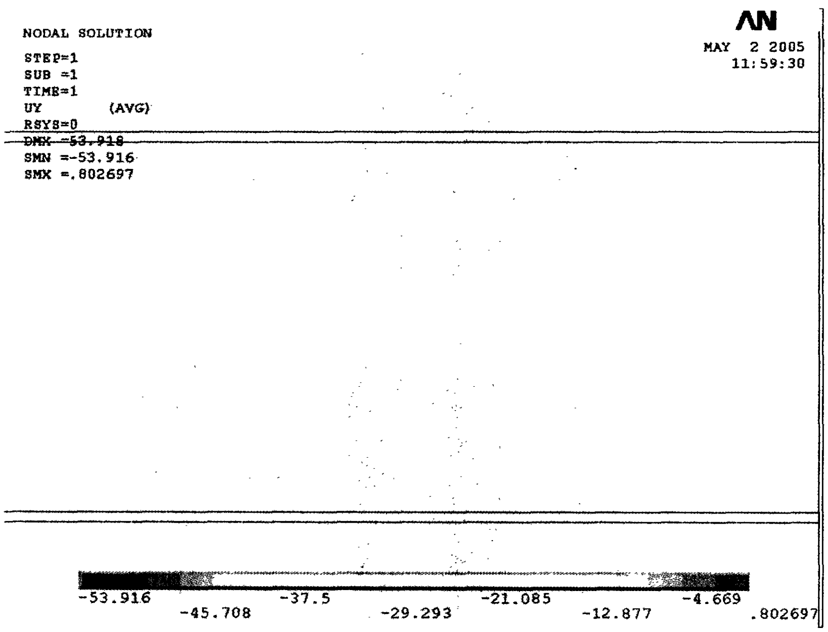

Figure C74: Y-displacement (μm) of copper 0.3μm thick under a weight of 0.04MPa. The vertical line farther right is the center of pressure.

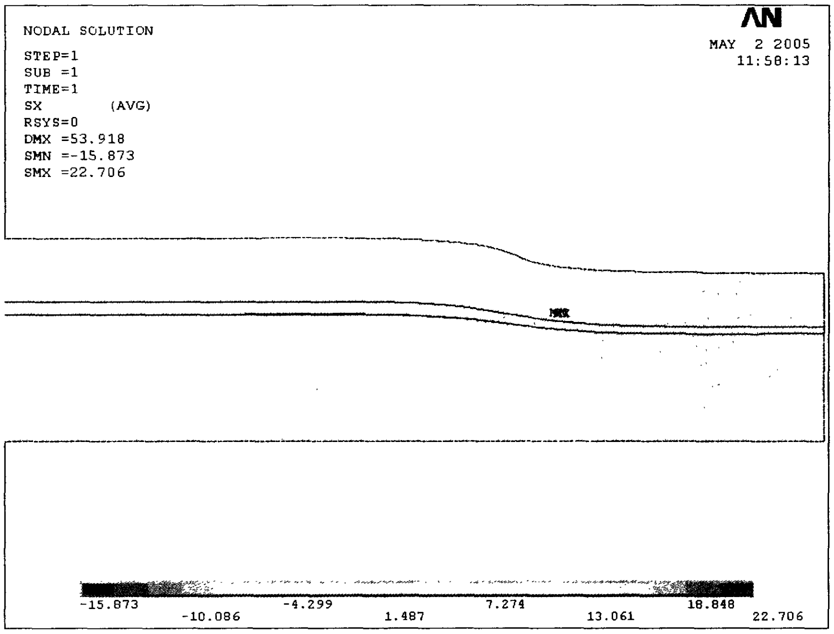

Figure C75: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.04MPa.

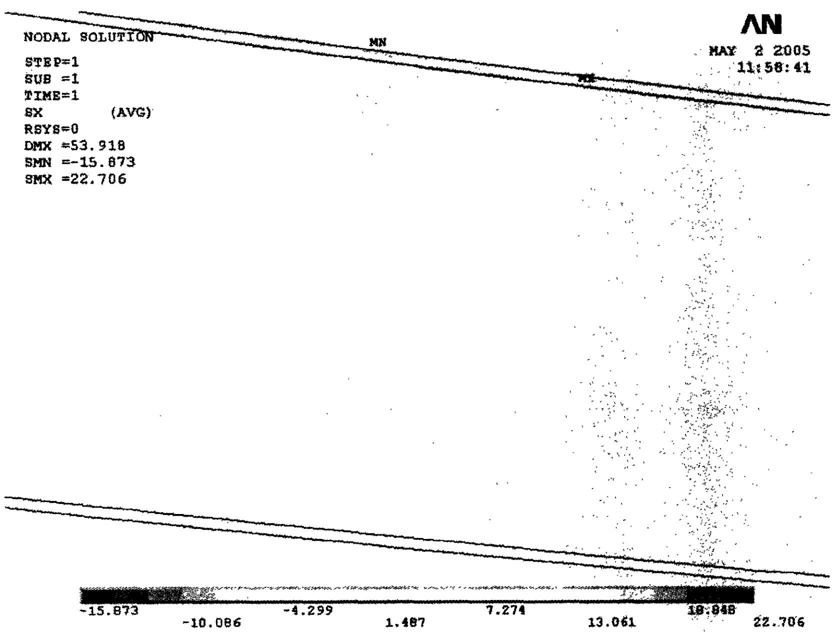

Figure C76: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.04MPa at the point of maximum and minimum stress.

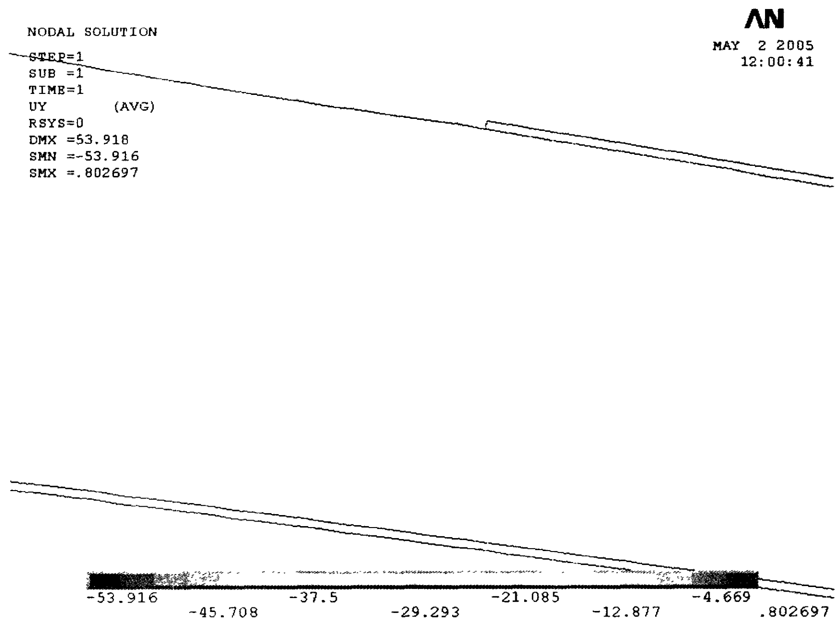

Figure C77: Y-displacement (μm) of copper 0.3μm thick under a weight of 0.04MPa, at the edge of the center copper node.

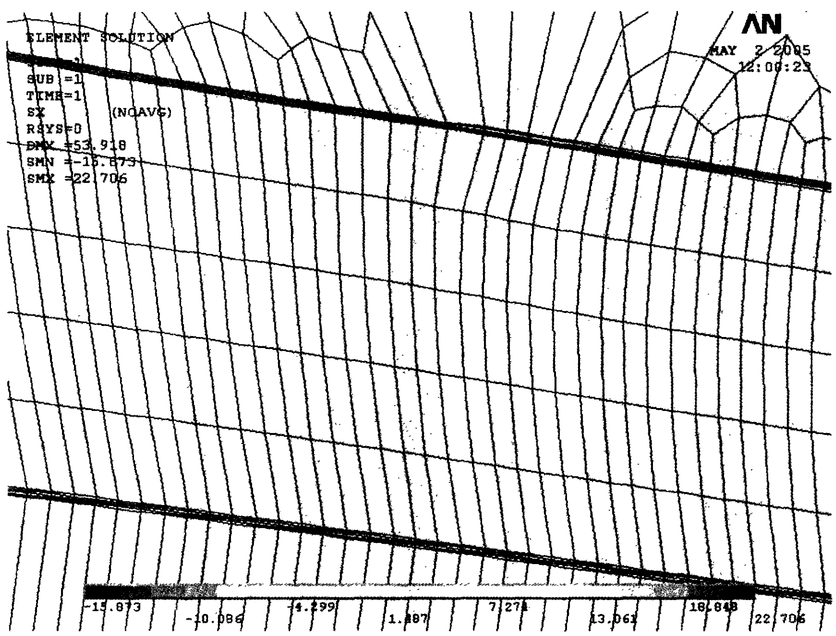

Figure C78: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.04MPa, at the edge of the center copper node, mesh showing.

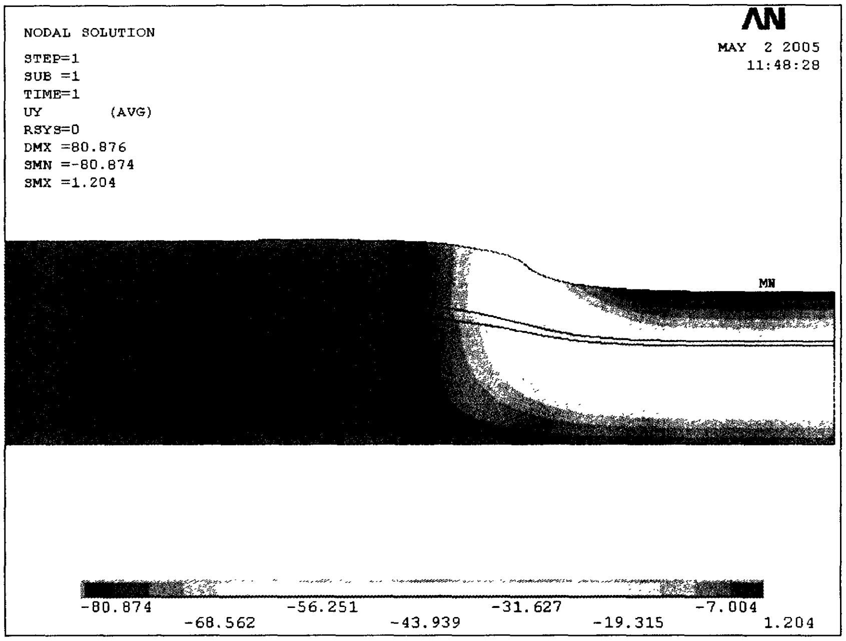

Figure C79: Y-displacement (μm) of copper 0.3μm thick under a pressure of 0.06MPa

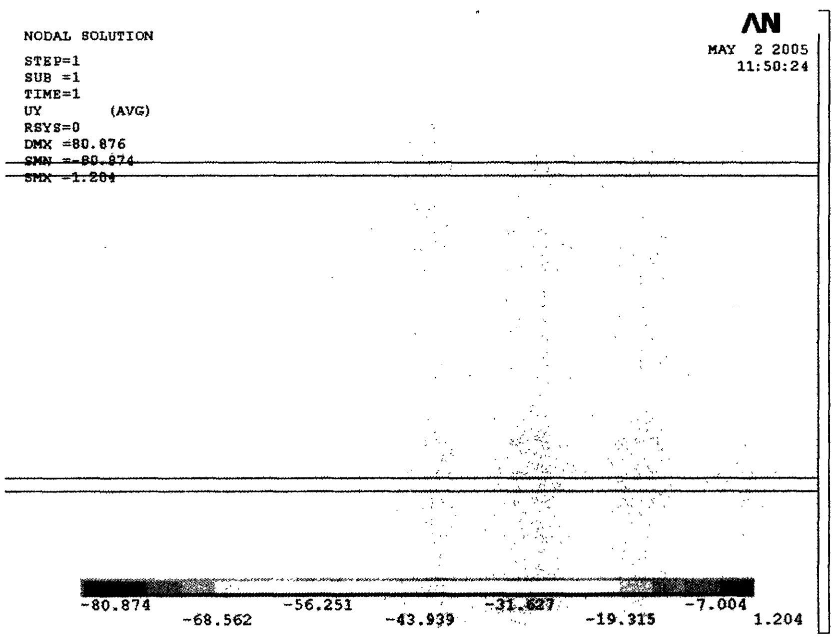

Figure C80: Y-displacement (μm) of copper 0.3μm thick under a weight of 0.06MPa. The vertical line farther right is the center of pressure.

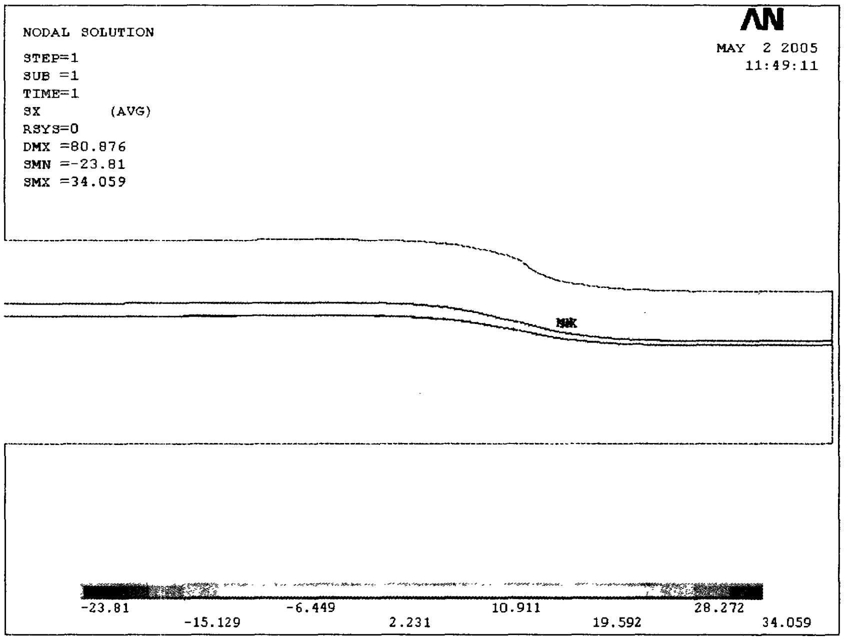

Figure C81: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.06MPa.

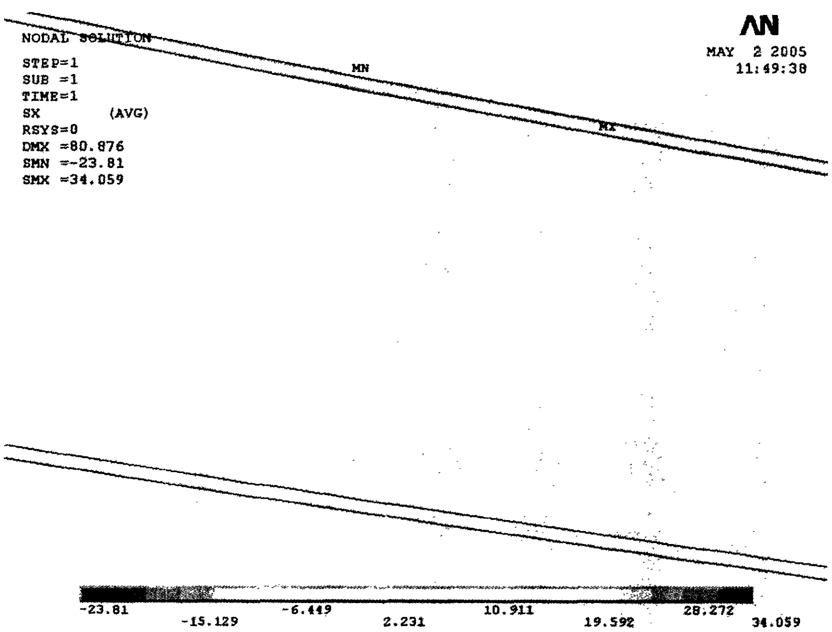

Figure C82: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.06MPa at the point of maximum and minimum stress.

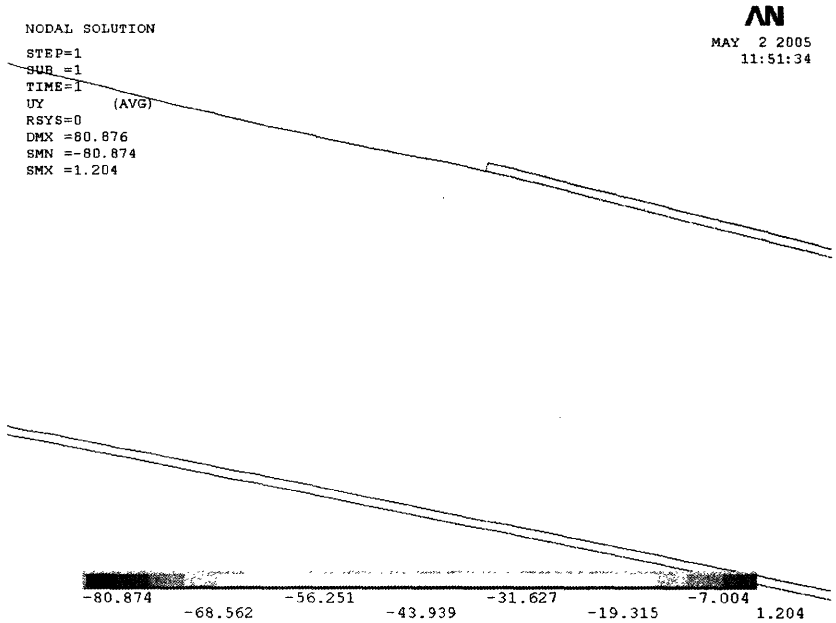

Figure C83: Y-displacement (μm) of copper 0.3μm thick under a weight of 0.06MPa, at the edge of the center copper node.

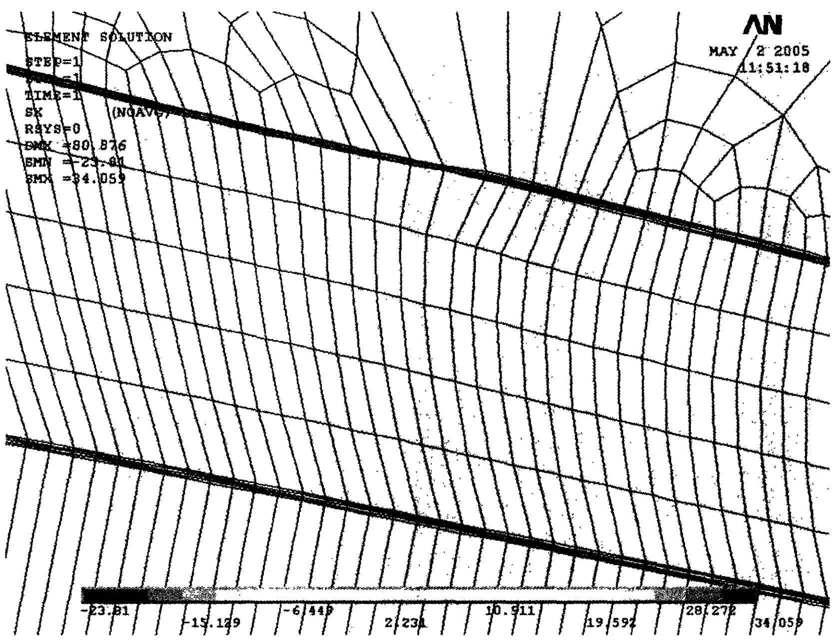

Figure C84: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.06MPa, at the edge of the center copper node, mesh showing.

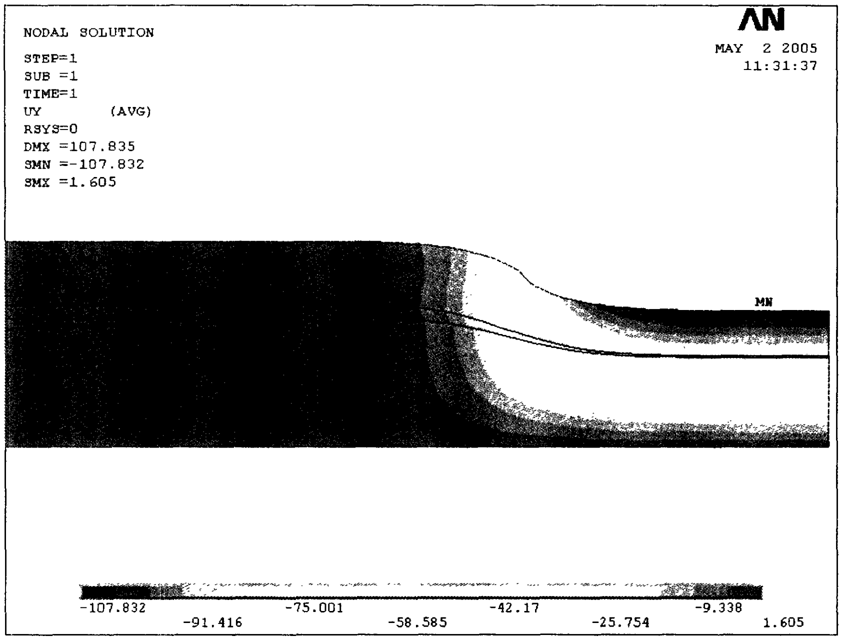

Figure C85: Y-displacement (μm) of copper 0.3μm thick under a pressure of 0.08MPa

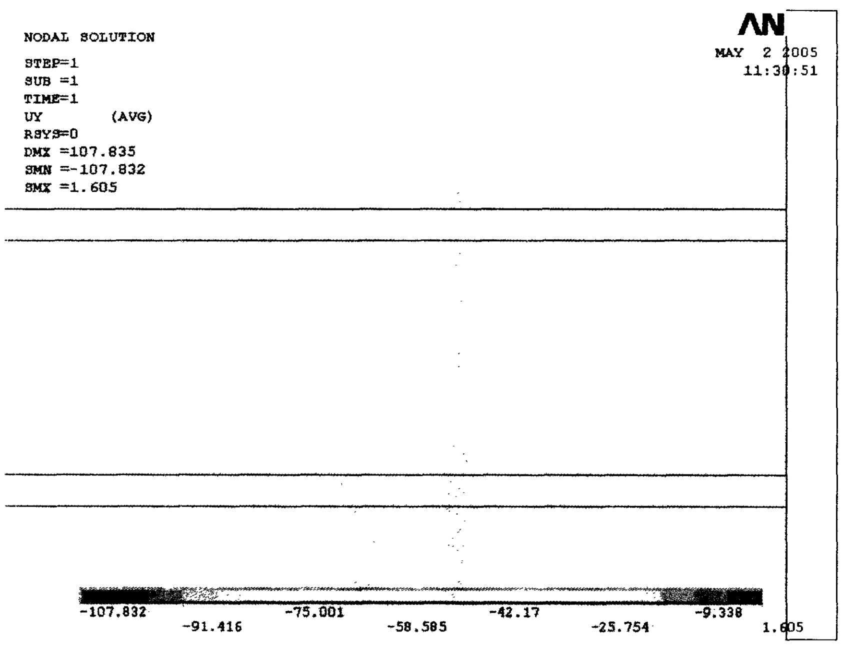

Figure C86: Y-displacement (μm) of copper 0.3μm thick under a weight of 0.08MPa. The vertical line farther right is the center of pressure.

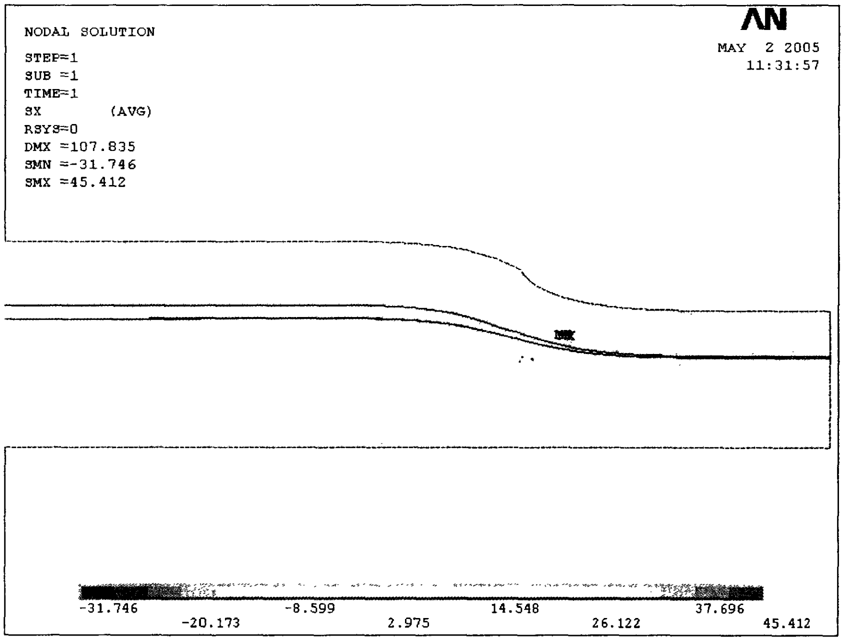

Figure C87: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.08MPa.

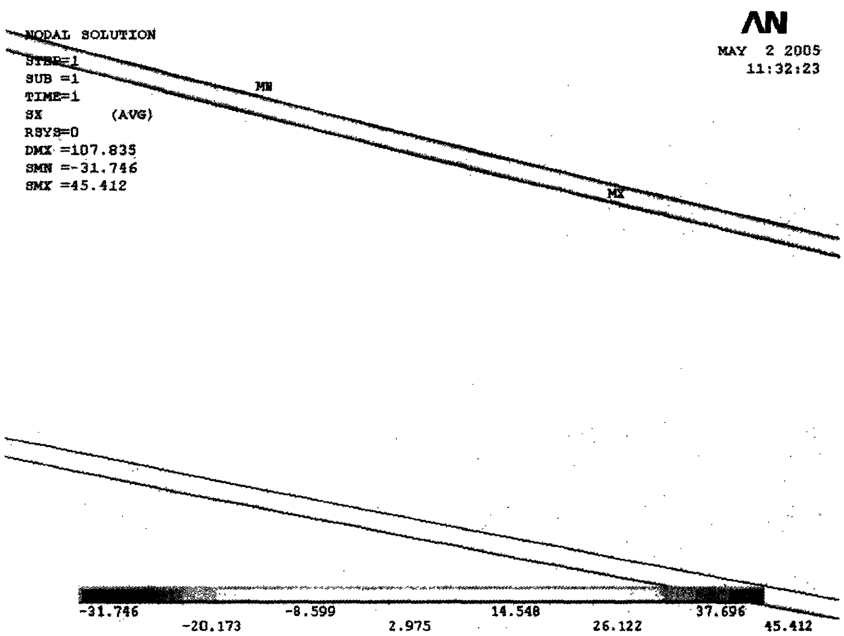

Figure C88: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.08MPa at the point of maximum and minimum stress.

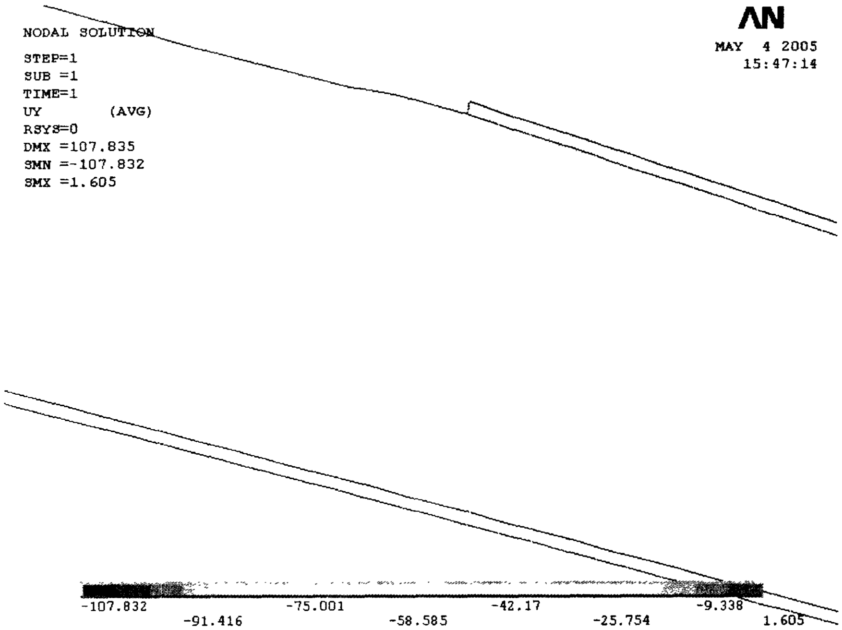

Figure C89: Y-displacement (μm) of copper 0.3μm thick under a weight of 0.08MPa, at the edge of the center copper node.

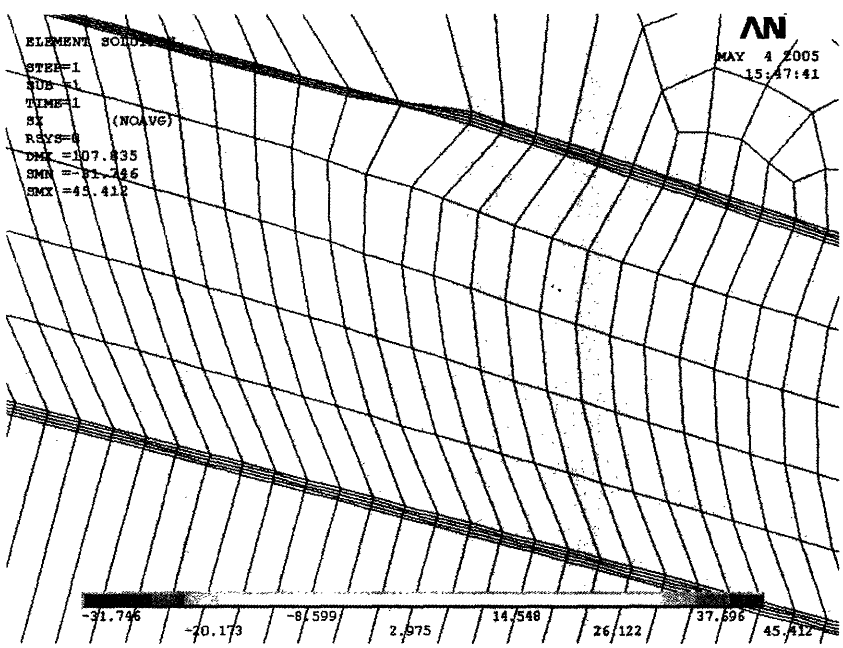

Figure C90: X-component of stress (MPa) of copper 0.3μm thick under a weight of 0.08MPa, at the edge of the center copper node, mesh showing.