Super Elastic Alloy Eyeglass Frame Design Using the ANSYS Workbench Environment

Peter R. Barrett, P.E.
Computer Aided Engineering Associates Inc.
Patrick Cunningham
Computer Aided Engineering Associates Inc.

Abstract

Up front analysis at the concept stages of developing a device most often result in reduced time to market and a better more efficient product development cycle. Design Modeler (1) / DesignXplorer (2) and ANSYS Structural's super-elastic material model (3) are used in this demonstration of a design process for an eyeglass frame. This example provides an illustration of using the parametric functionality of the ANSYS Workbench environment for a highly nonlinear application. The Workbench environment can be used to perform design iterations in conjunction with material laws only available in the standard ANSYS environment. A shape memory superelastic material law was formally included in ANSYS 8.0. The material law captures the super-elastic behavior of Nitinol (4) along with its unique hysteretic response under cyclic loading. In the demonstration, the eyeglass frame is subjected to crushing and cyclic loading events. Geometric design variables are varied in an attempt to increase the frame stiffness yet keep the peak strain levels in the frame below 10% strain, which is a typical elastic limit for Nitinol. The cyclic behavior of the material is also monitored under the design load to evaluate the potential of fatigue failure. The procedure demonstrates the ease in interfacing the suite of Workbench products with geometric, material and contact nonlinear analysis problems.

Introduction

Finite element analysis (FEA) is commonly used in the verification and validation of product designs through the use of structural analysis simulation models. The analysis models predict behaviors ranging from ultimate strength, buckling resistance, dynamic response to fatigue resistance. FEA is a requirement for many regulatory agencies such as the Food & Drug Administration's (FDA), Nuclear Regulatory Agency (NRC) etc. However, the design value of finite element analysis is much more than a validation tool if it is used early in the design process.

Developing a parametric model of the device is key to allowing for design iterations to be performed quickly and efficiently, since the entire analysis process can be automated. Two options are routinely used in ANSYS to perform these automated analyses:

1. Use APDL scripts to generate keypoint, line, area, and volume geometry in ANSYS. Multiple dimensions can be adjusted using scripted routines. Each analysis is performed using a script that includes model building, meshing, analysis and postprocessing. Side-by-side comparisons can be obtained between design iterations. Furthermore, design optimization routines can be invoked that will find a design that, based on user-defined criterion, provides an analytical optimum. All ANSYS solution functionality is available including material, geometric and contact nonlinearities. The downside to this method is the development of the APDL geometry script can be very time consuming and in most cases is in addition to CAD model development. Also the lack of robustness of re-generation using ANSYS Boolean operations can sometimes lead to geometry failures.

2. Use a feature based CAD package or the ANSYS DesignModeler tool to develop a parametric model. Import the parametric geometry model into the ANSYS Workbench Simulation Tool to perform the analysis. Using the Workbench DesignXplorer tools, determine the sensitivity of the design to changes in the geometry, materials, or loading conditions.

   The advantages of this method are that the geometry re-generation is very robust in the CAD environment and the final design is available for immediate prototyping, etc. The downside of this method is that the analysis functionality is currently limited to linear material properties within the Workbench graphical user interface.

This paper presents the merging of the two techniques where the parametric geometry capabilities of the Workbench environment are used in conjunction with APDL scripting to include geometric, material and contact nonlinearities. The eyeglass analysis also illustrates the use of the new Super-Elastic Material law available in ANSYS to simulate the structural response of Nitinol. Furthermore the DesignXplorer module is used to perform design of experiment simulations to develop a design field for the designer to use without having to perform additional analyses.

Material Model

Eyeglass frames are typically made from plastics or standard metals such as aluminum or titanium. Frames constructed of these materials will exhibit permanent set and or failure if large strains are induced. Building eyeglass frames from a shape memory alloy such as Nitinol (or Flexon – a brand name) have an advantage in that the material can withstand roughly 10% strain and fully recover to its original shape. Frames made of Flexon snap back into shape even after severe twisting, bending and crushing. Flexon frames are also lightweight, hypoallergenic and corrosion-resistant.

Nitinol is an acronym for NIckel TItanium Naval Ordinance Laboratory since the alloy was originally developed at the U.S. Naval Laboratory (4). It is used to describe a family of materials, which contain a nearly equal mixture of nickel and titanium. Nitinol alloys are attractive because they are biocompatible and are at their optimum superelastic behavior at room temperature when processed properly.

Nitinol exhibits a different stress-strain relationship for loading and unloading that cannot be modeled with kinematic or isotropic hardening laws. ANSYS 8.0 introduced a shape memory material law that can accurately predict this unique loading and unloading behavior (See Figure 1). The ANSYS Nitinol material model is based on the Auricchio Algorithm (5), which has been implemented in other commercial finite element programs as well.

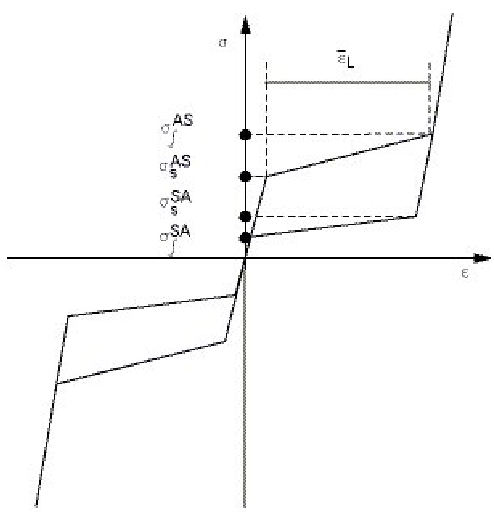

Figure 1 – Shape Memory Material model from the ANSYS 8.0 Structural Analysis Guide section 8.3.1.6

Depending upon the processing and mixture of nickel and titanium, Nitinol can exhibit quite different material response. The material response is also sensitive to temperature, therefore physical testing is required to determine the actual input data that should be used in finite element simulations. The material data listed in this example are representative properties at room temperature and are provided solely to demonstrate the ANSYS input structure:

Constant	Meaning
SIG-SAS (C1)	Starting stress value for the forward phase transformation
SIG-FAS (C2)	Final stress value for the forward phase transformation
SIG-SSA (C3)	Starting stress value for the reverse phase transformation
SIG-FSA (C4)	Final stress value for the reverse phase transformation
EPSILON (C5)	Maximum residual strain
ALPHA (C6)	α material responses ratio between tension and compression
YMRT (C7)	Modulus for Martensite (This is Beta in 8.0)

```
…………………………………………………………..
! Example Nitinol Material Properties (Room Temperature)
E_A = 3E6          ! psi
NU  = 0.3
S_ASS = 52000    ! psi
S_ASF = 60000    ! psi
S_SAS = 30000    ! psi
S_SAF = 20000    ! psi
EPS_L = 0.07       ! in/in
ALP   = 0.00        ! defaults to 1.0
YMRT  = 3E6       ! psi
! User specified Nitinol material properties
mp,ex,1,E_A
mp,nuxy,1,NU
tb,sma,1,1
tbdata,1,S_ASS,S_ASF,S_SAS,S_SAF,EPS_L,ALP
tbdata,7,YMRT
………………………………………………………
```

Analysis

Three different Workbench modules are used in the eyeglass frame design analysis. DesignModeler is used to develop the parametric geometry. DesignSimulator is used in conjunction with APDL for meshing, solution and postprocessing. DesignXplorer is used to evaluate the design sensitivity.

Geometry creation using the DesignModeler module

Design Modeler is a feature based solid modeling tool that is ideal for developing parametric geometry for use with the DesignSimulation tool in the Workbench environment. All dimensions used to create the geometry can be defined as parameters. Relationships between parameters can also be defined to accommodate design requirements. In addition to defining the geometry of the frame, the model also includes additional geometric subdivisions or cuts to facilitate a brick mesh for more efficient solution times. The eyeglass model is shown in the DesignModeler environment in Figure 2.

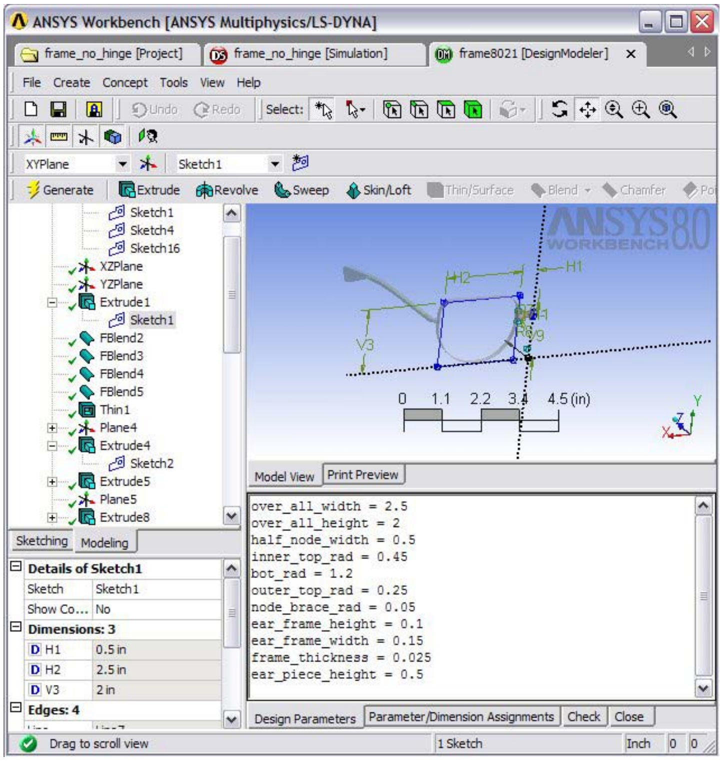

Figure 2 – DesignModeler Geometry

The DesignModeler eyeglass model development is summarized as follows:

- Create 2D planes and sketches as the basis for 3D solid modeling functions such as Extrude. Extruded sketches either add or subtract material from the current geometry.
- Add secondary 3D features that do not require sketch geometry such as fillets and chamfers.
- Specify critical dimensions as named parameters.
- Define relations between critical parameter.
- Use sketching planes to slice the part at specific regions to enable sweep meshing in DesignSimulation.

Analysis using the DesignSimulation module

When the eyeglass frame geometry is ready for the analysis stage, the geometry is loaded into the Workbench DesignSimulation module. The example simulation used on the eyeglass frame consists of a crush load between a pair of rigid plates. The half symmetry model developed in DesignModeler is used as a basis for the frame geometry. The lenses are neglected in this simulation. Allowing the DesignSimulation tool to generate weak springs between the model and grounded free ends prevents rigid body motion. Loading consists of a vertical displacement on the top rigid plate deflecting the frame 1.0 inch.

When the geometry is attached in the DesignSimulation tool all of the specified geometric parameters from DesignModeler are available for adjustment directly in the DesignSimulation module as show in Figure 3.

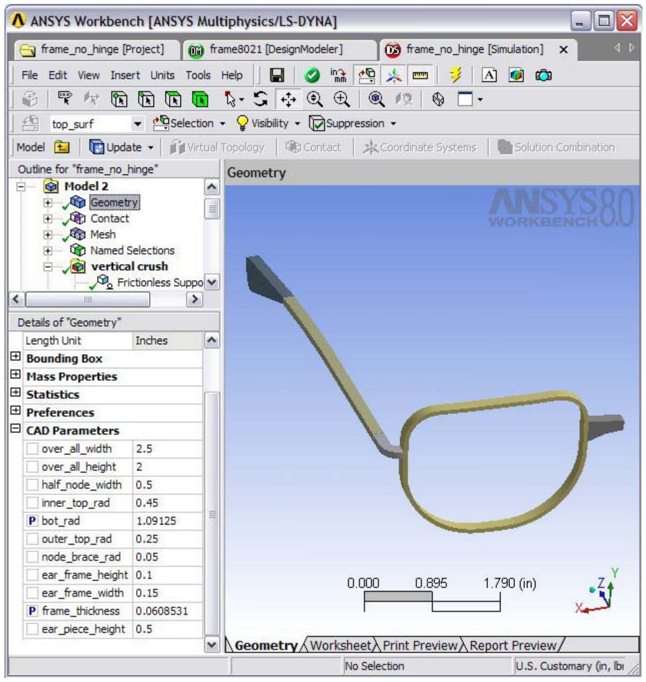

Figure 3 – The DesignSimulation Module

The DesignSimulation tool is the analysis engine of the Workbench environment. The following analysis characteristics for the eyeglass simulation are defined:

1. <u>Default material model</u> - For this case the default material model is used. It is later overwritten with the shape memory material model using added preprocessing commands.
2. <u>Part connections</u> - Since the frame geometry has been sliced to facilitate sweep meshing, the individual parts are connected using surface-to-surface bonded contact pairs. Contact pairs are automatically defined for all adjoining surfaces that fall within a specified geometric tolerance. Contact pairs can also be manually defined in a fashion similar to that of the ANSYS contact wizard.
3. <u>Mesh definition</u> - The default brick mesh was viewed and element size definitions were added to refine the mesh. A coarse mesh is used in this example simulation since this is a demonstration case where relative results are of most interest. The finite element mesh created in Workbench is illustrated in Figure 4.

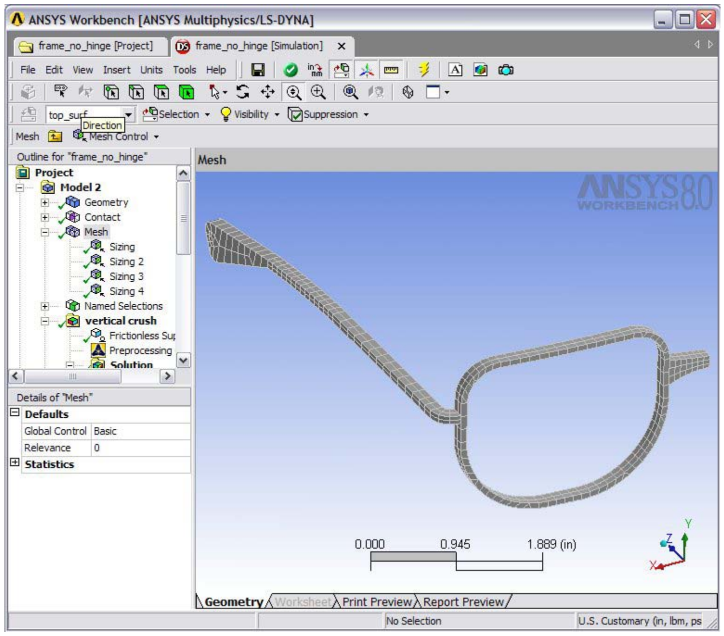

Figure 4 – Finite Element Mesh of the Eyeglass Frame.

4. Named Selections – Groupings of geometric entities can be selected and named. For the eyeglass frame we require named selections for the surfaces where the contact elements will be created on the external faces of the solid elements. A named selection is defined for both the top and bottom surfaces of the eyeglass frame. A named selection for the inner surface of the frame is also defined. A self-contact pair will be defined here to accommodate the frame collapsing onto itself. The named selections are shown in Figure 5.

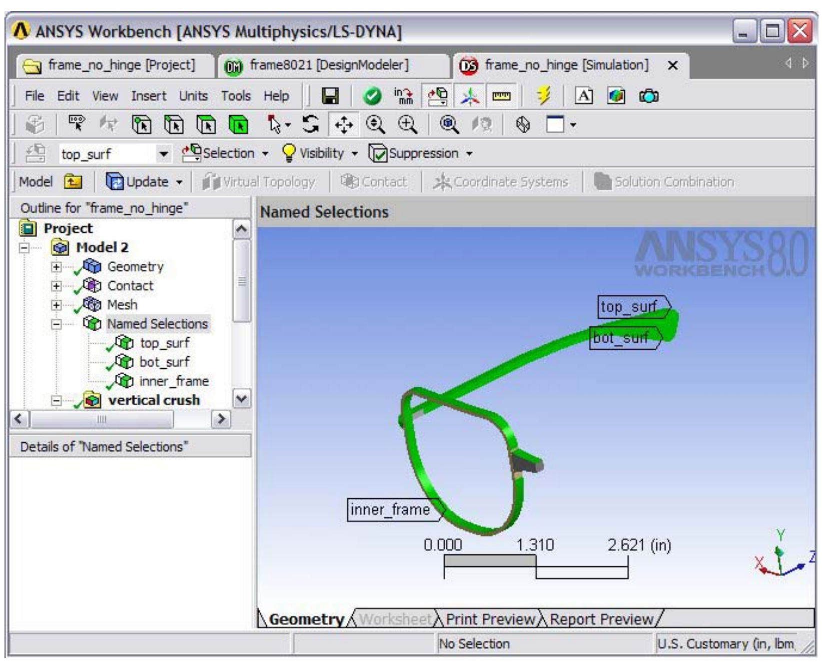

Figure 5 – Named Selections used to define contact pairs for the crushing load.

5. Boundary Conditions - Boundary Conditions are applied to solid model entities in the DesignSimulation tool. For the eyeglass frame a frictionless surface is defined on the symmetry

plane through the bridge of the frame. The remaining boundary conditions are applied using APDL commands in the Preprocessing Commands Worksheet.

6. Additional Preprocessing and Solution Specifications – The following model characteristics are defined using the Preprocessing Commands Worksheet:
   a. The shape memory material model.
   b. Additional contact pairs and boundary conditions - In this example the crushing load will be applied by sandwiching the eyeglass frame between two rigid contact surfaces. The boundary conditions for this loading consist of a fixed rigid target below the eyeglass frame and a second rigid target above the frame with a 1" displacement defined downward towards the frame.

      Since we have not created geometry to represent the target surfaces, the contact pair will be generated using APDL commands. The rigid target elements are generated directly using the ANSYS "N" and "E" commands to define nodes and elements respectively. The contact elements associated with each pair are generated using the Named Components defined in the DesignSimulation tool. Although the named components are defined on surfaces in the DesignSimulation tool, they are converted into nodal components that are available for use with APDL commands in the Preprocessing Command tool. The nodal components and the associated element faces are selected and the "ESURF" command is used to generate contact elements on the faces of the solid elements.

      The contact key-options and real constants can also be defined at this stage. In the eyeglass example, the contact stiffness is lowered to account for the frame flexibility.

      Using the named components in this fashion is advantageous in that the resulting contact pairs become parametric because they are dependent on the base geometry. The contact pairs are regenerated for each design iteration automatically with no user intervention required.
   c. Solution controls – Solution parameters commonly used in nonlinear analyses are defined in the Preprocessing Commands worksheet. These include time step controls, large deformation control, line search and solution output control. Note: All of the following controls are available in GUI form using the Preprocessing Command "Builder".

7. Result Entities – Result quantities are defined and specified as output variables in the DesignSimulation tool for the DesignExplorer module.
8. Additional Postprocessing – Since result quantities in the DesignSimulation tool are limited to linear result quantities (with the exception of contact results), the material nonlinear result items for the eyeglass frame can be evaluated using the Postprocessing Command Builder. APDL macros can also be used in the Postprocessing Commands worksheet. Result figures and listings from the Postprocessing Command Builder can be passed back to the DesignSimulation tool for inclusion in the design report.
9. DesignXplorer Input – The identification of input and output parameters for the sensitivity study are required at this step to be used later with the DesignXplorer analysis. For the eyeglass example, the frame thickness and lens corner radii have been selected as input variables. Maximum equivalent stress and total deformation are defined as output variables.

Prior to kicking off the sensitivity study using DesignXplorer a single deterministic analysis using the DesignSimulation tool is used to checkout the model and verify the boundary conditions, the mesh and the solution controls. When all model and solution issues are resolved, the DesignXplorer tool is used to evaluate the sensitivity of the output variables to variations in the input. Postprocessing in DesignXplorer allows for the viewing of a response surface based on a DOE (Design of Experiments) methodology.

Sensitivity Studies using the DesignXplorer Module

After the model and solution issues are resolved for the deterministic run, the DesignXplorer tool is used to evaluate the sensitivity of the output variables to input variations. Upper and lower bounds are defined for the input variables and a design of experiments method is used to determine the system response based on variation of the input. These are the only changes required from the deterministic analyses. Whereas using straight APDL scripting typically requires a significant amount of setup time to define the correct input commands.

For the example eyeglass frame the input variables are defined as the thickness (.05 to .07 inches) and bottom radius (.5 to 1.25 inches) of the eyeglass frame. The output variables are the Von Mises stress and the total displacement (usum). Using the design of experiments methodology nine “sample designs” or deterministic analyses were conducted by the DesignXplorer module in order to generate a response polynomial for each output variable. From the response polynomials the “optimum design” can be interpolated and a “hard design” or deterministic analysis of the optimum characteristics can be run for confirmation.

Analysis Results & Discussion

Results from the analysis from the deterministic and designXplorer simulations are presented in this section. Figure 6 illustrates the crushed response of the eyeglass frame. The crush plates are shown in translucency for graphics clarity.

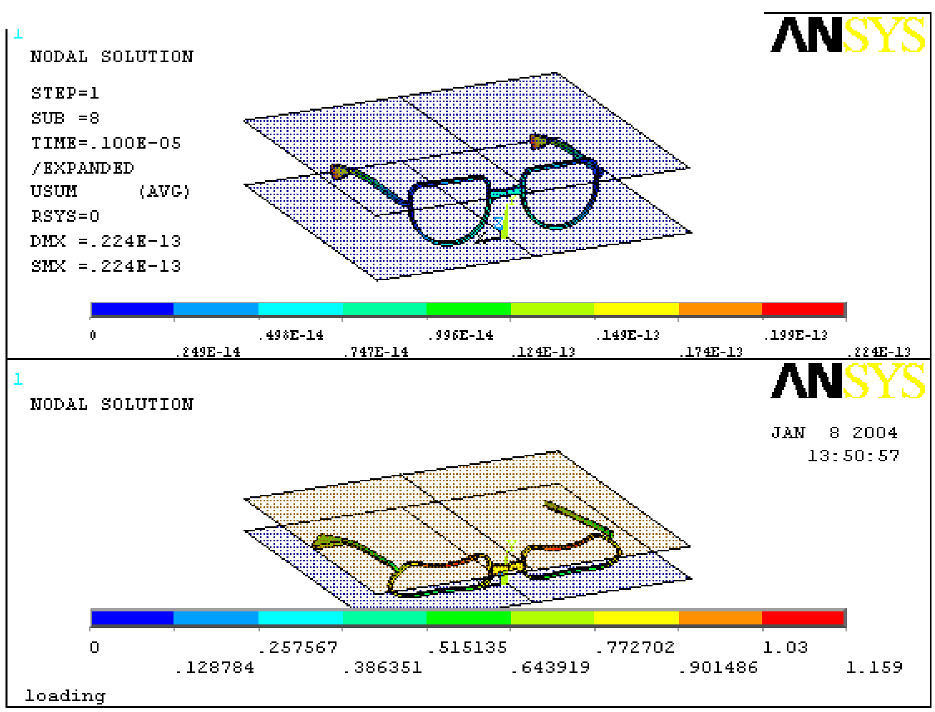

Figure 6 – Total deformation, initial and crushed.

Figures 7 & 8 below illustrate the stress and strain distribution throughout the model in the fully crushed stress state. Maximum values occur in the radii defined as design parameters.

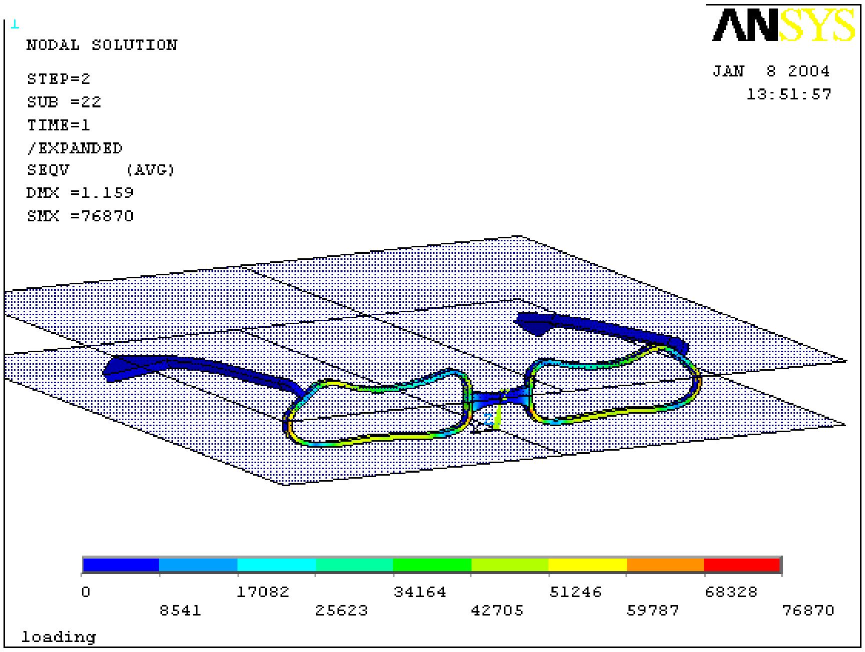

Figure 7 – Von Mises stress

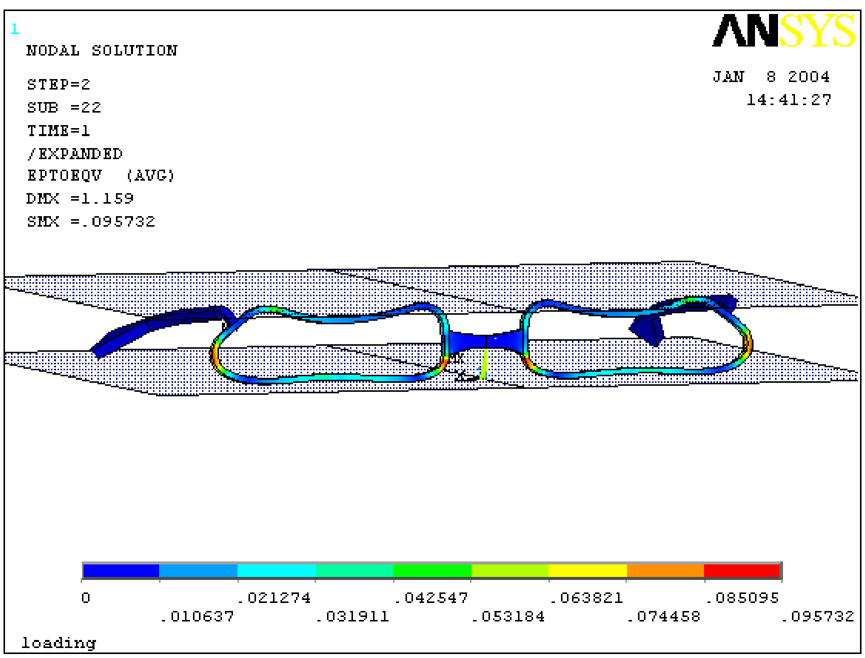

Figure 8 – total strain contour of compressed frame

Figure 9 illustrates the hysteretic response captured by the shape memory material model. The unique differences in loading (the upper curve) vs. unloading (the lower curve) are captured with the super elastic material model. Delta stress or strain states extracted from this type of simulation are typically used in a Goodman fatigue evaluation.

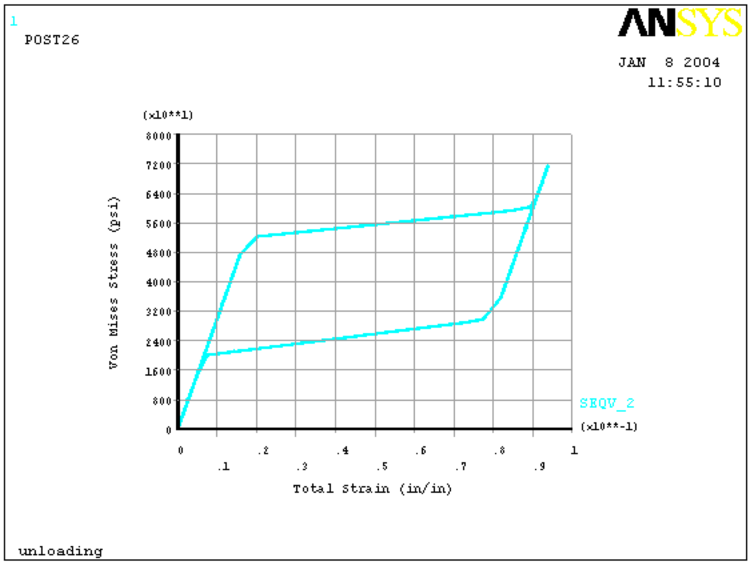

Figure 9 – Shape memory alloy material response

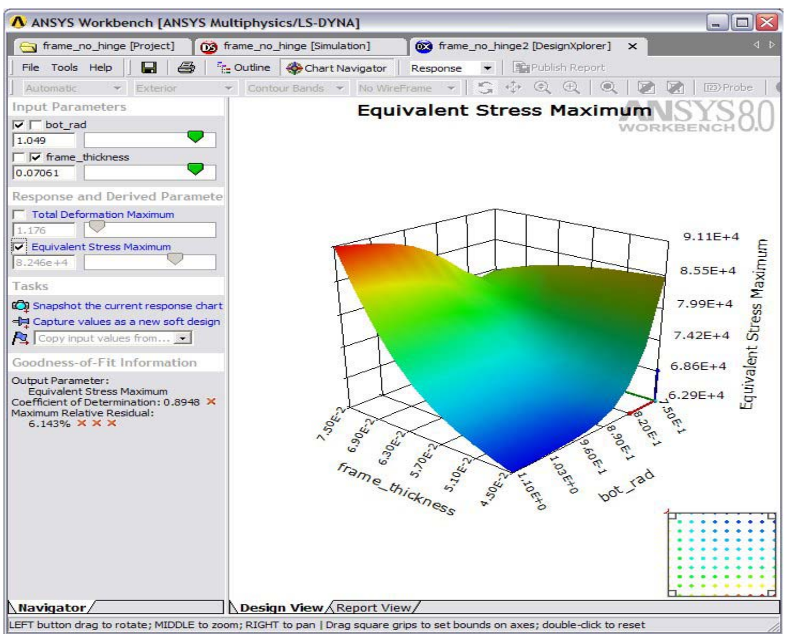

Figure 10 – Response surface for input variables versus Maximum Von Mises stress.

The response surface diagram illustrated above provides graphical representation of the effects changes in the frame thickness and radius has on the peak Von Mises Stress in the model. In addition to the curve, numerical solutions are provided in the left column. For the above example, a maximum stress of 82 ksi is predicted for a frame thickness of 0.0706” and a corresponding inner radius of 1.049 inches. The design engineer can use this data in conjunction with other parameters to evaluate feasible designs with the benefit of not having to run additional simulations. Once a design is selected, a hard analysis can be performed to verify the interpolation determined from the design of experiments calculation. Figure 11 below illustrates the selection process and preview capabilities of different design options.

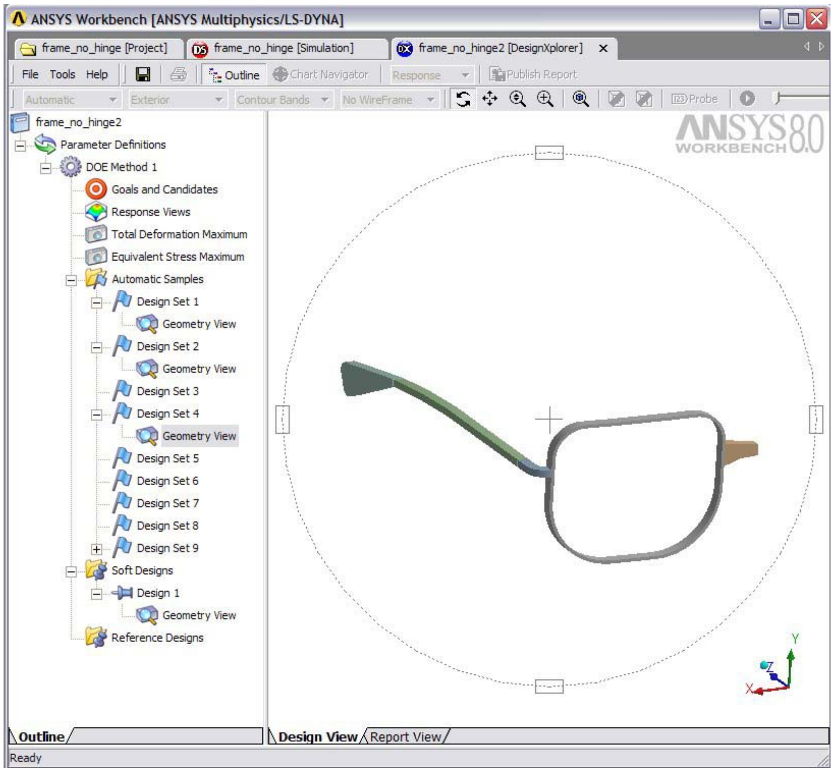

Figure 11 – Geometry of automatic sample Design Set 4

Conclusion

This paper illustrates the use of nonlinear analysis techniques incorporated into the Workbench / designXplorer modules of ANSYS. The tool provides for very complex analyses to be performed fast and efficiently with little user intervention.

Future enhancements of this example simulation in the Workbench GUI or customization of the GUI could result in a robust tool specialized for eyeglass frame design.

References

1) Design Modeler Users Manual, ANSYS Inc. Copyright © 2003 SAS IP, Inc. All rights reserved. Unpublished rights reserved under the Copyright Laws of the United States

2) DesignXplorer Users Manual, ANSYS Inc. Copyright © 2003 SAS IP, Inc. All rights reserved. Unpublished rights reserved under the Copyright Laws of the United States

3) ANSYS Users Manual, ANSYS Inc. Copyright © 2003 SAS IP, Inc. All rights reserved. Unpublished rights reserved under the Copyright Laws of the United States

4) NiTiNOL (from Nickel Titanium Naval Ordinance Laboratory) was developed in 1959 by William J. Buehler

5) Auricchio, Ferdinando, "A robust integration-algorithm for a finite-strain shape-memory-alloy superelastic model," International Journal of Plasticity 17 (2001) 971-990.