LASER INTERFEROMETER GRAVITATIONAL WAVE OBSERVATORY

LIGO Laboratory / LIGO Scientific Collaboration

LIGO-T0900480-v1 *LIGO* 5 Oct 2009

Notes from FEA Studies of HAM ISI Spring

Andy Stein

Distribution of this document:
LIGO Scientific Collaboration

This is an internal working note
of the LIGO Laboratory.

California Institute of Technology
LIGO Project – MS 18-34
1200 E. California Blvd.
Pasadena, CA 91125
Phone (626) 395-2129
Fax (626) 304-9834
E-mail: info@ligo.caltech.edu

Massachusetts Institute of Technology
LIGO Project – NW22-295
185 Albany St
Cambridge, MA 02139
Phone (617) 253-4824
Fax (617) 253-7014
E-mail: info@ligo.mit.edu

LIGO Hanford Observatory
P.O. Box 1970
Richland WA 99352
Phone 509-372-8106
Fax 509-372-8137

LIGO Livingston Observatory
P.O. Box 940
Livingston, LA 70754
Phone 225-686-3100
Fax 225-686-7189

http://www.ligo.caltech.edu/

1 Introduction

This document provides a quick summary of finite element analysis (FEA) studies of the HAM ISI Spring, D071100. These studies were run in an effort to duplicate results previously obtained by High Precision Devices, Inc. (HPD), the design firm contracted by LIGO to help design the Enhanced LIGO version of the HAM ISI Isolation Table, D071400. The HPD results are presented in their Final Design Review document, G070156.

The primary goal of the current studies is to demonstrate that the SolidWorks Simulation Professional FEA package used at LIGO's MIT office is sufficient to analyze the 'geometrically nonlinear' case of a D071100 Spring loaded to its equilibrium (i.e. 'flat') shape.

2 FEA Studies

This section is divided into three parts: 1) FEA of the D071100 Spring, using a linear solver; 2) FEA of the D071100 Spring, using the 'large displacement' solver; and 3) analysis results from HPD's G070156 document.

2.1 Linear Solution

The analysis is run on the 'FEA' configuration of D071100-v1, which is available on the LIGO PDMWorks vault. This is the same model used by HPD for the FEA results shown in G070156. The model is one-half of the Spring and takes advantage of the Spring's mirror symmetry.

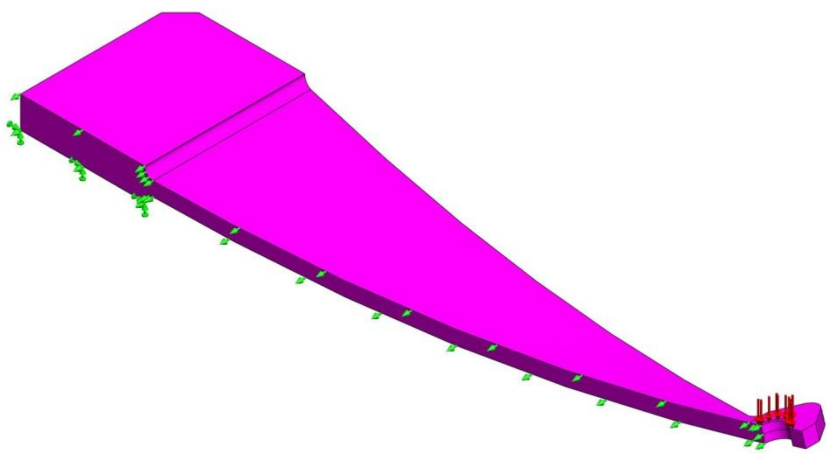
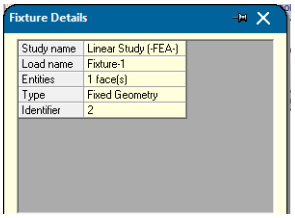


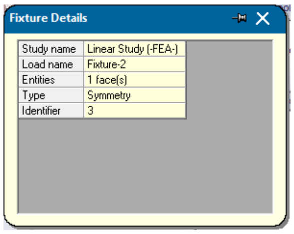
Figure 1. The model used for static FEA of the D071100 HAM ISI Spring.

The boundary conditions are the same as used by HPD for their FEA studies. The bottom of the Spring base is Fixed, while the mid-plane surface is given a Symmetric constraint.


Fixture Details

Study name	Linear Study (-FEA-)
Load name	Fixture-1
Entities	1 face(s)
Type	Fixed Geometry
Identifier	2

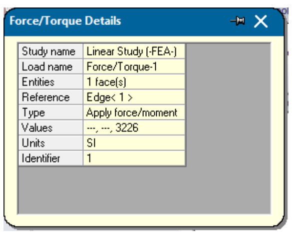
Figure 2. SolidWorks Simulation summary of Fixed constraint.


Fixture Details

Study name	Linear Study (-FEA-)
Load name	Fixture-2
Entities	1 face(s)
Type	Symmetry
Identifier	3

Figure 3. SolidWorks Simulation summary of Symmetry constraint.

A 3226 N force is applied to the split-line annulus at the tip of the Spring, acting in the vertical direction. This force equals half the 6451.71 N 'Equilibrium Load' presented on p. 74 of HPD's G070156 document.


Force/Torque Details

Study name	Linear Study (-FEA-)
Load name	Force/Torque-1
Entities	1 face(s)
Reference	Edge< 1 >
Type	Apply force/moment
Values	---, ---, 3226
Units	SI
Identifier	1

Figure 4. SolidWorks Simulation summary of external loads applied to Spring model.

The mesh for this study has a global element size of 6 mm, and a finer 1 mm element size on both base fillet surfaces. (This deviates from HPD's mesh settings, where they used an h-adaptive iterative mesh to automatically increase the mesh density around areas with high strain.)

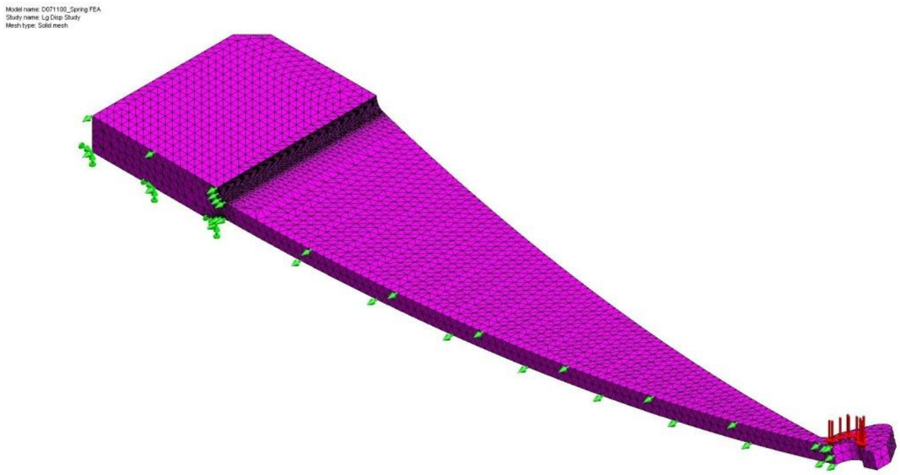


Figure 5. The Spring model after meshing.

For the first study, the 'Large Displacement' flag is disabled. This means the study assumes a constant stiffness matrix, regardless of the magnitude of the part's deformation.

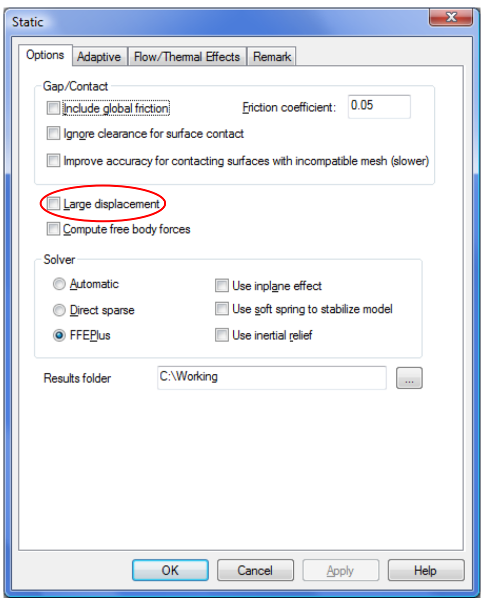


Figure 6. Properties dialog box for the SolidWorks Simulation static study. The 'Large Displacement' flag is disabled for this study.

The solution takes just 13 seconds to compute. The von Mises stress plot is shown in the following figure. The peak stress occurs at the middle of the base fillet, and equals 806 MPa.

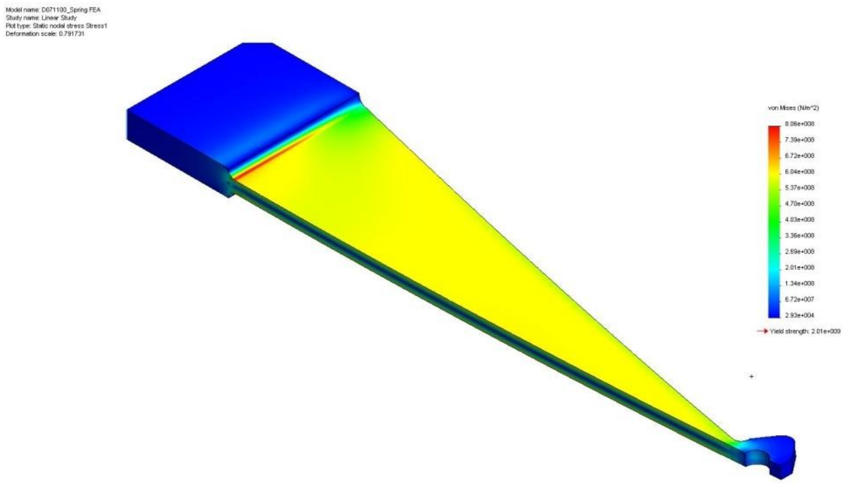


Figure 7. Results plot for stress, computed with linear solver. Note the deformed shape is shown, at true scale.

The vertical displacement of the tip is shown in the following snapshot. The tip moves 73.3 mm, according to this linear solution.

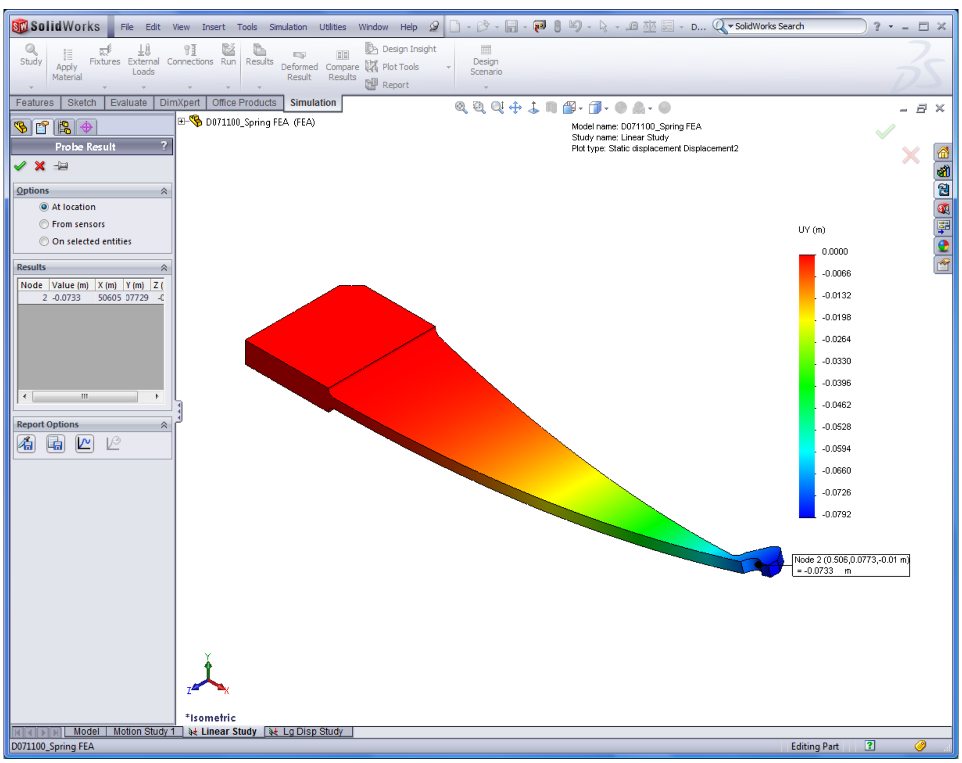


Figure 8. Vertical displacement (UY) results plot. A point on the inside of the tip's hole is probed, to show the displacement at the point of interest (on the tip of the Spring, near the Flexure axis).

2.2 'Large Displacement' Solution

We run the study again, using the same constraints, load, and mesh, but this time with the 'Large Displacement' flag enabled. This allows the solver to iterate the stiffness matrix as the load is applied incrementally. Since the shape of the Spring visibly changes as the load is applied, we expect a significant change in the model's stiffness matrix as it approaches its equilibrium shape. Therefore, this 'Large Displacement' solving method should produce more accurate results than the linear solution shown above.

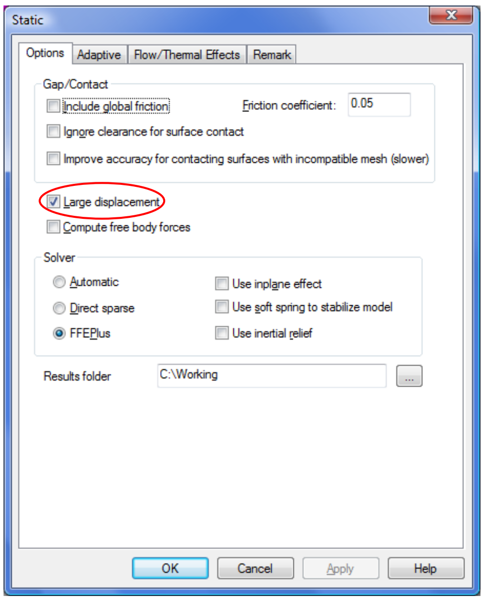


Figure 9. Static study properties dialog box, with the 'Large Displacement' flag enabled.

This study takes slightly more than 4 minutes to solve. The von Mises stress results are shown below. The maximum stress is now computed at 817 MPa.

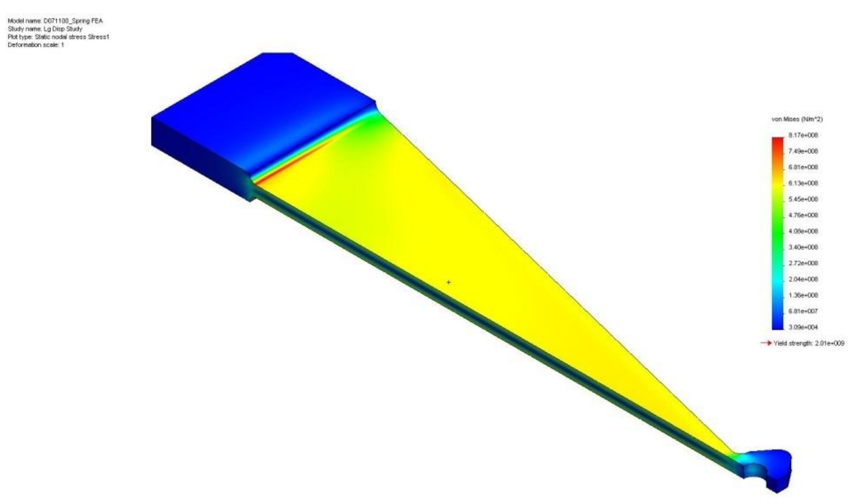


Figure 10. Results plot for stress, computed with 'Large Displacement' solver.

The vertical displacement is shown below. As shown in the legend, the peak displacement is 82.7mm. Using the results probe to pick a point near the Flexure axis, we observe a predicted displacement of 76.7 mm at the Spring tip.

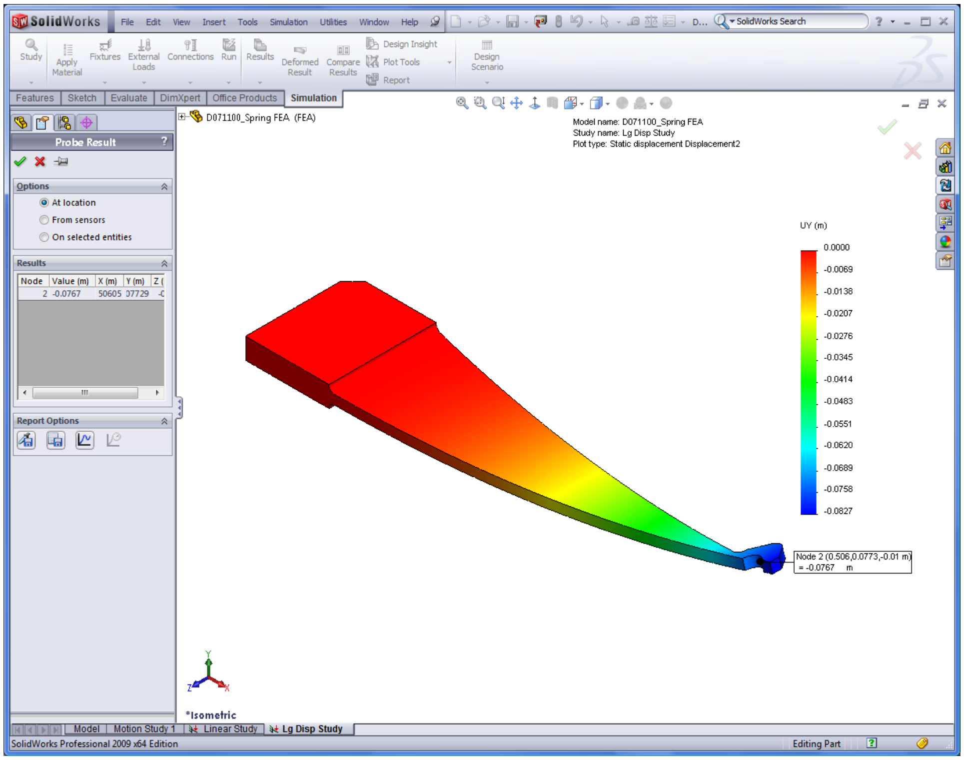


Figure 11. Results plot for vertical displacement, computed with 'Large Displacement' solver. Note the Spring model is shown in its undeformed state.

2.3 HPD's Results

Below are FEA results plots presented in HPD's G070156 document. The maximum von Mises stress calculated for this FEA study is 825 MPa.

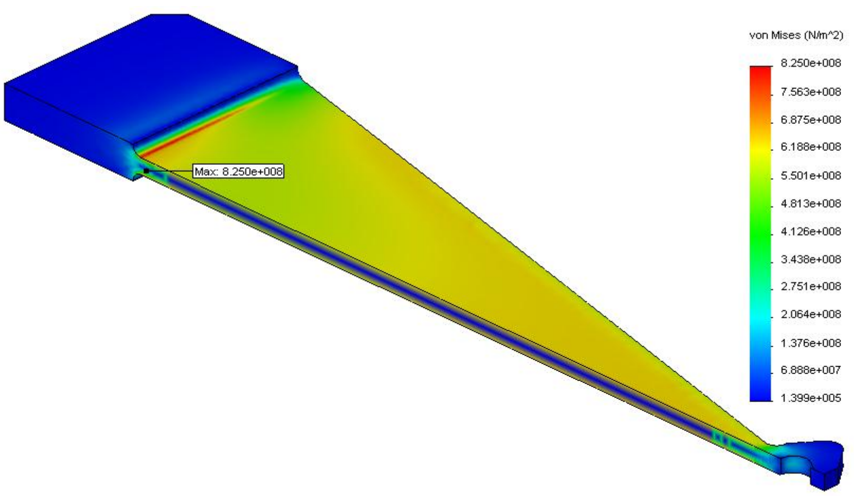


Figure 12. Stress results from HPD's analysis of HAM ISI Spring.

The maximum vertical deflection predicted by this analysis is 82.7 mm. According to the supporting spreadsheet calculations (not posted publicly), the vertical displacement measured at the Spring's tip is 76.7 mm.

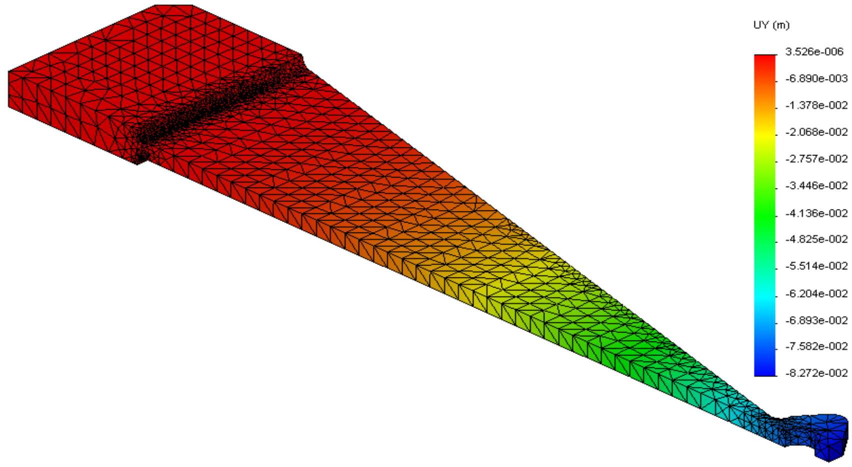


Figure 13. Vertical displacement of HAM ISI Spring resulting from a total vertical load of 6452 N, as predicted by HPD's analysis.

3 Conclusions

Comparing the stress plots for the linear, 'large displacement', and HPD FEA studies, we find that 1) the linear solution predicts a maximum stress that is 2.3% smaller than the HPD results, and 2) the 'large displacement' solution predicts a maximum stress that is 1.0% smaller than the HPD results. However, we should not expect our results to be any closer to the HPD FEA than this, since peak stress values are very strongly dependent on details of the model's meshing, and the meshing used in our studies are known to be different than the meshing used by HPD.

However, when we look at the vertical displacements predicted by us and by HPD, we see that the 'large displacement' solution is identical (to .1 mm) to the results presented by HPD. The linear solution, alternatively, predicts a tip displacement that is 4.4% smaller than the HPD results.

In summary, the results we obtained using the 'large displacement' setting within the SolidWorks Simulation Professional FEA package appear to agree very well with the FEA results presented in HPD's Final Design Review document. We should feel confident using similar methods to analyze the springs in the Advanced LIGO BSC ISI, D0901182.