LASER INTERFEROMETER GRAVITATIONAL WAVE OBSERVATORY

LIGO Laboratory / LIGO Scientific Collaboration

LIGO-T050184-01-D *Advanced LIGO* 26 Sep 2005

Test Mass Optical Surface Deformation due to Gravity

Dennis Coyne

Distribution of this document:
LIGO Scientific Collaboration

This is an internal working note
of the LIGO Project.

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 – NW17-161
175 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
Mail Stop S9-02
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

The gravitational load (body force) on the Input and End Test Masses (ITM, ETM) are supported in the Advanced LIGO suspensions by "ears" bonded to flats on sides of these optics. The stress field created by the gravitational load and the resulting ear bond reaction forces will cause a deformation of the optic. We are concerned with the deformation of the optical surfaces, which are polished in a horizontal orientation.

2 Model

The fused silica ITM and ETM dimensions[1] are 340 mm diameter by 200 mm thick with 95 mm long flats on each side (nominally 40 kg mass). A three-dimensional finite element model, created with the I-DEAS version-9 software, depicted in Figure 1, represented this geometry. The bevels and the wedge angle of the optics were not included in this model. The ear, which is bonded to the optic and welded to the fused silica fibers (or ribbons), was not modeled either. The bond area was restrained from motion and served to provide a reaction to the gravitational load. The dimensions of the two bond areas were taken[2] to be 18 mm wide by 15 mm high. These bond areas should be placed centered front-to-back, between the two optical faces, but below the horizontal center-plane of the optic, such that the bending flexure point of the fiber/ribbon is slightly above the optic center of mass (by a distance d4 in the suspension design parameter terminology[3]). The vertical placement depends upon the details of the ear design and the desired value of d4. The latest quadruple ITM/ETM suspension design parameter set[4] has a d4 value of 1 mm. The bond areas (surface areas with nodal restraints) in the finite element model were placed with the lower edge tangent to the centerline and so are about 20 mm too high compared to the intended design. I doubt that this will affect the magnitude of the results very much; however the deformation pattern might shift relative to the center of the optic[5].

It was found that the asymmetry of free (automatic) meshing caused significant quasi-rigid body pitch and roll deformations. A mapped mesh ensured a purely vertical quasi-rigid-body global motion in response to a vertical gravity vector.

---

[1] H. Armandula, G. Billingsley, G. Harry, B. Kells, "Core Optics Components: Conceptual Design Document", LIGO-T000098-02. Also LIGO RODA M050397-00 in review.

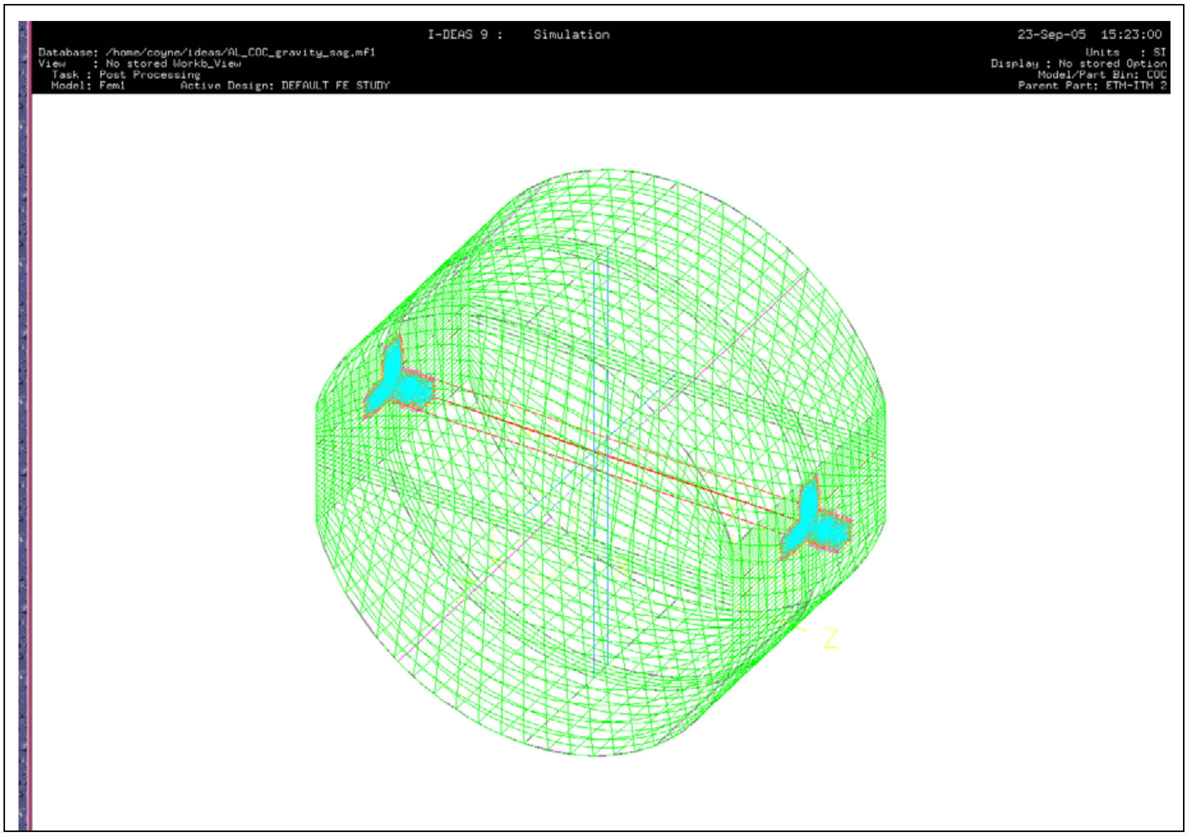
[2] I could not readily find formal documentation on the planned dimensions/geometry of the ITM/ETM ears. I took the dimensions from Matthieu Musso's study as representative. Musso's shows a design which improves upon the original GEO ear geometry:
M. Musso, "Test Masses Suspensions Modeling", E030392-00.

[3] M. Perreur-Lloyd, "Pendulum Parameter Descriptions and Naming Conventions", LIGO-T040072-01, 20 Jul 2004.

[4] N. Robertson, et. al., "Parameters for current ETM/ITM main chain noise prototype design", LIGO-T040214-01, 12 Nov 2004.

[5] The model can be revised and the analysis rerun, of course.

The ITM and ETM are actually pitched relative to the local gravity vector in order to align to the long Fabry-Perot optical cavities[6]. This nominal pitch angle is varies from –0.619 mrad to +0.326 mrad. In the analysis reported here only a vertical gravity vector is considered.



I-DEAS 9 : Simulation
23-Sep-05 15:23:00
Database: /home/coyne/ideas/AL_COC_gravity_sag.mf1
View : No stored Workb_View
Task : Post Processing
Model: Fem1
Active Design: DEFAULT FE STUDY
Units : SI
Display : No stored Option
Model/Part Bin: COC
Parent Part: ETM-ITM 2

Figure 1: Finite Element Mesh

The mesh consists of 13,440 parabolic, brick elements and 59,041 nodes.

3 Calculated Stress & Deformation Field

The stress contours (Figures 2, 3 and 4) show that only the region immediately adjacent to the bond area has significant stress, as expected. The penetration depth of the stress field is on the order of the bond area width, h, which also seems reasonable. One might expect then that the peak transverse displacement due to Poisson's effect would be approximately

$$\delta_T = \frac{\nu m g}{4hE} = 13\, nm$$

where $\nu = 0.17$, Poisson's ratio for fused silica; $E = 7.0 \times 10^{10}$, Young's modulus for fused silica; $h = 0.018$ m is the bond area width; $m = 40$ kg is the optic mass and $g = 9.8$ m/s^2. The finite element

---

[6] W. Althouse, L. Jones, A. Lazzarini, "Determination of Global and Local Coordinate Axes for the LIGOSites", LIGO-T980044-10, 07 Feb 2001.

analysis indicates a peak transverse (normal to the optic face) deformation of +/- 4 nm, or 8 nm total (as shown in Figures 4 and 5).

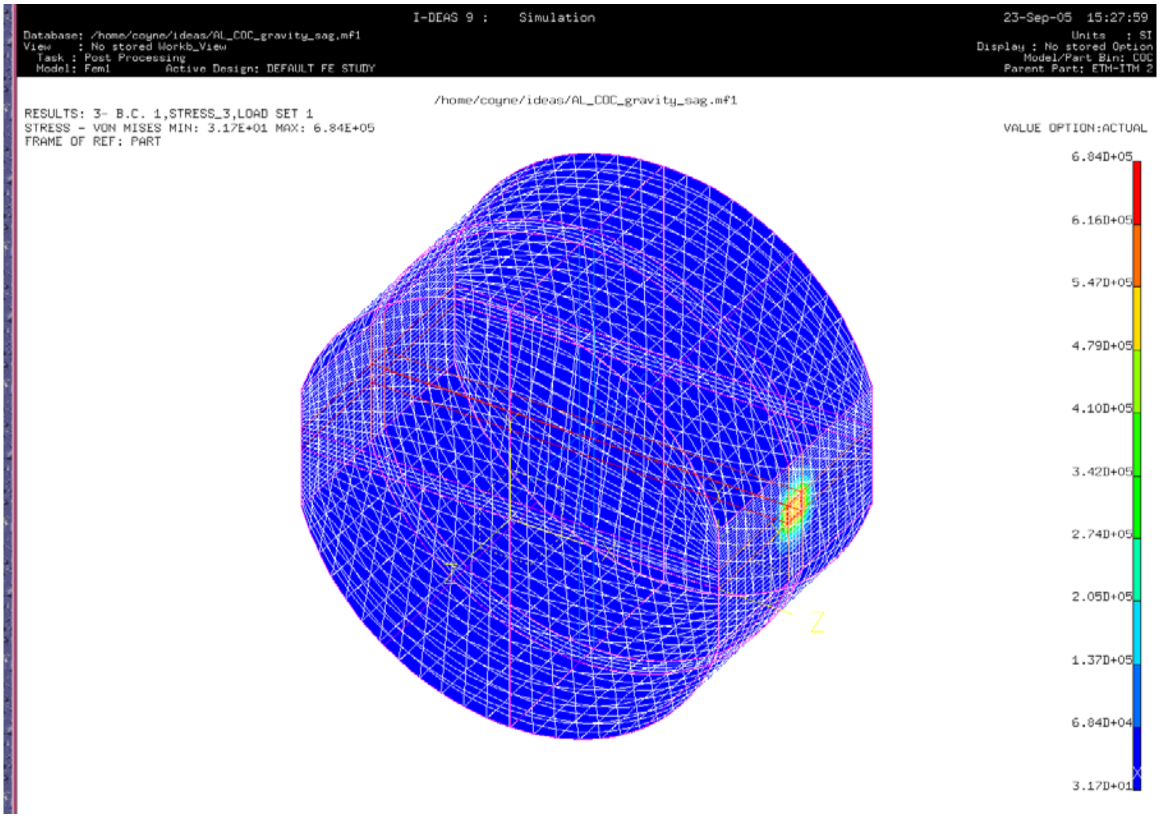
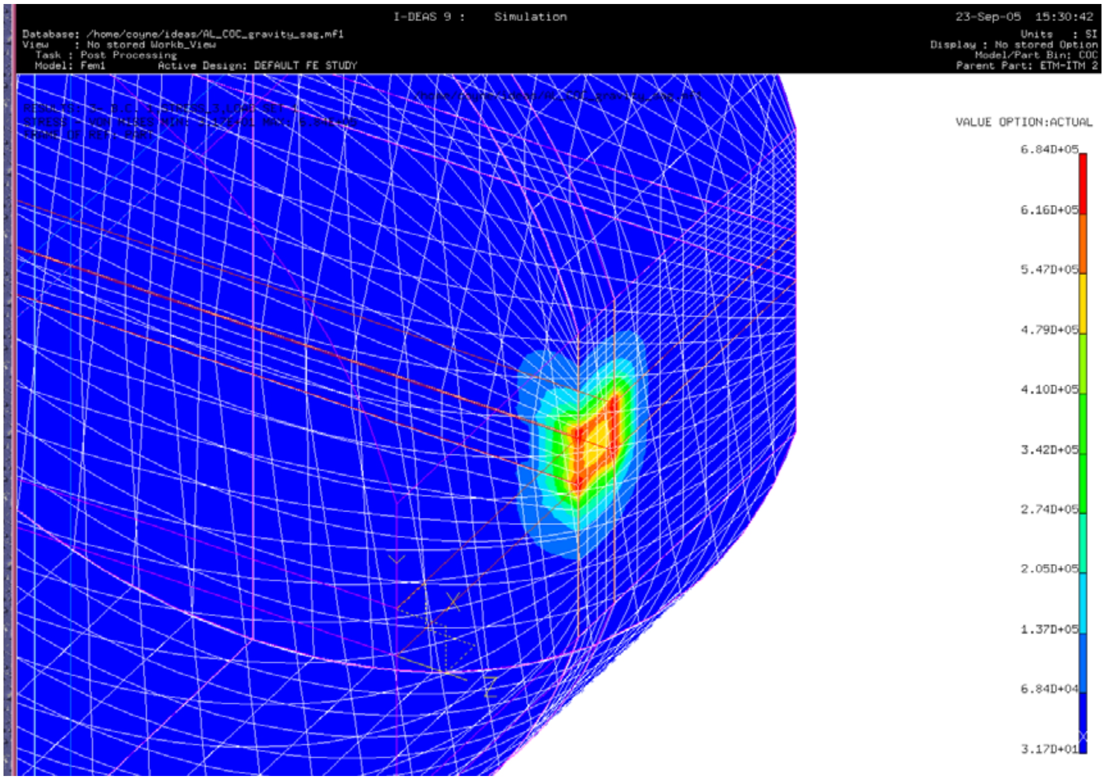


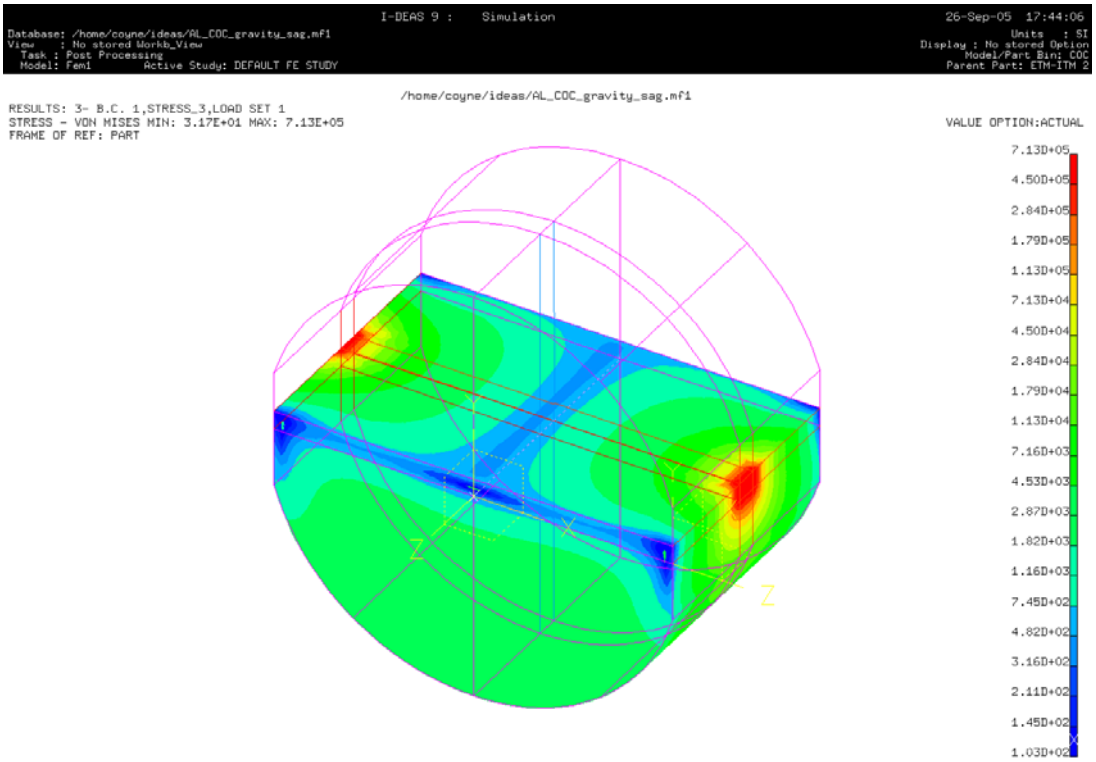
Figure 2: Stress Contours (Von Mises stress, linear scale)



```
I-DEAS 9 :     Simulation                                  23-Sep-05  15:30:42
Database: /home/coyne/ideas/AL_COC_gravity_sag.mf1                  Units    : SI
View    : No stored Workb_View                        Display : No stored Option
  Task : Post Processing                                  Model/Part Bin: COC
  Model: Fem1         Active Design: DEFAULT FE STUDY     Parent Part: ETM-ITM 2

/home/coyne/ideas/AL_COC_gravity_sag.mf1
RESULTS: 3- B.C. 1,STRESS_3,LOAD SET 1
STRESS - VON MISES MIN: 3.17E+01 MAX: 6.84E+05
FRAME OF REF: PART
VALUE OPTION:ACTUAL
```

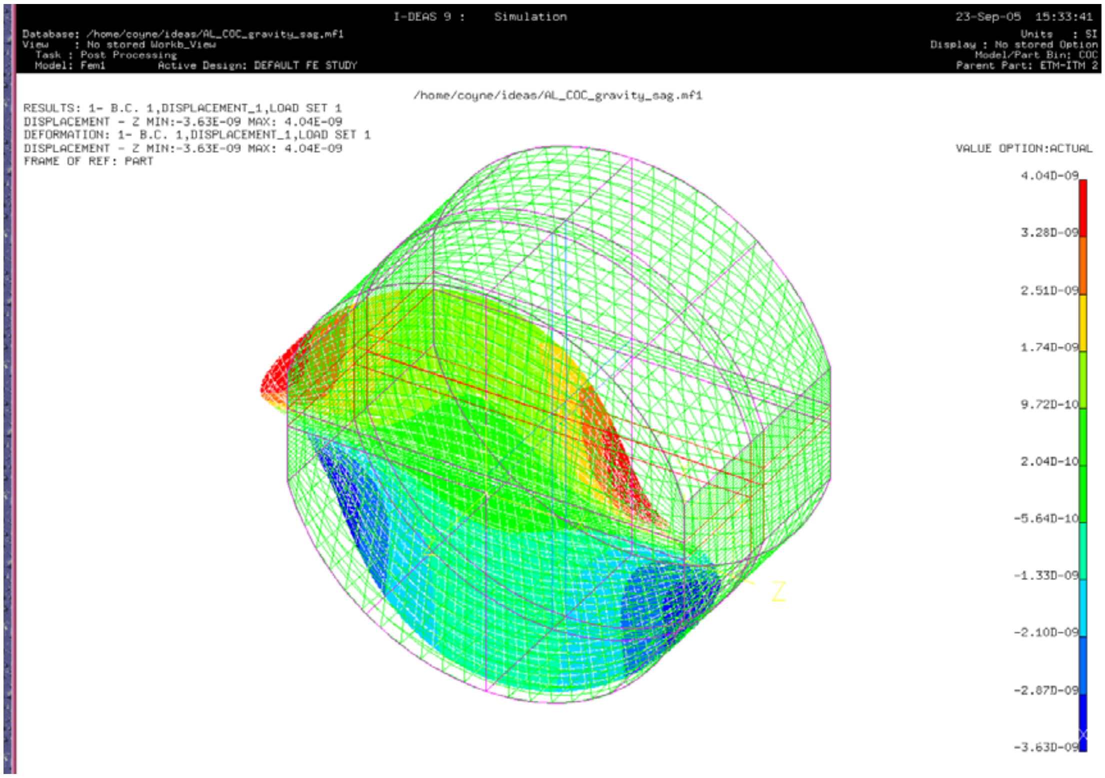
Figure 3: Close-up and cut-away of the stress contours (Von Mises stress, linear scale)



```
I-DEAS 9 :     Simulation                                  26-Sep-05  17:44:06
Database: /home/coyne/ideas/AL_COC_gravity_sag.mf1                  Units    : SI
View    : No stored Workb_View                        Display : No stored Option
  Task : Post Processing                                  Model/Part Bin: COC
  Model: Fem1         Active Study: DEFAULT FE STUDY      Parent Part: ETM-ITM 2

/home/coyne/ideas/AL_COC_gravity_sag.mf1
RESULTS: 3- B.C. 1,STRESS_3,LOAD SET 1
STRESS - VON MISES MIN: 3.17E+01 MAX: 7.13E+05
FRAME OF REF: PART
VALUE OPTION:ACTUAL
```

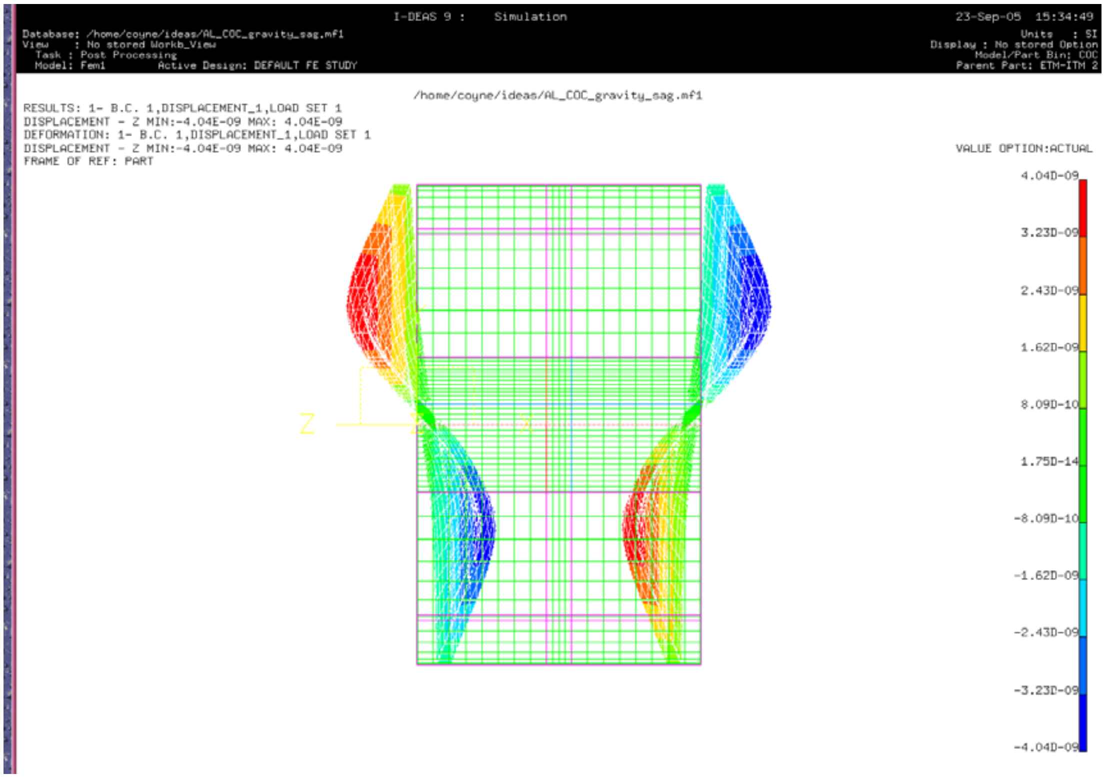
Figure 4: Stress contours for cut-away of lower half of optic (Von Mises stress, log scale)



```
                         I-DEAS 9 :    Simulation                       23-Sep-05  15:33:41
Database: /home/coyne/ideas/AL_COC_gravity_sag.mf1                               Units   : SI
View    : No stored Workb_View                                  Display : No stored Option
  Task : Post Processing                                            Model/Part Bin: COC
  Model: Fem1         Active Design: DEFAULT FE STUDY               Parent Part: ETM-ITM 2

                        /home/coyne/ideas/AL_COC_gravity_sag.mf1
RESULTS: 1- B.C. 1,DISPLACEMENT_1,LOAD SET 1
DISPLACEMENT - Z MIN:-3.63E-09 MAX: 4.04E-09
DEFORMATION: 1- B.C. 1,DISPLACEMENT_1,LOAD SET 1
DISPLACEMENT - Z MIN:-3.63E-09 MAX: 4.04E-09
FRAME OF REF: PART
                                                                  VALUE OPTION:ACTUAL
```

Figure 5: Front Surface Normal Displacement



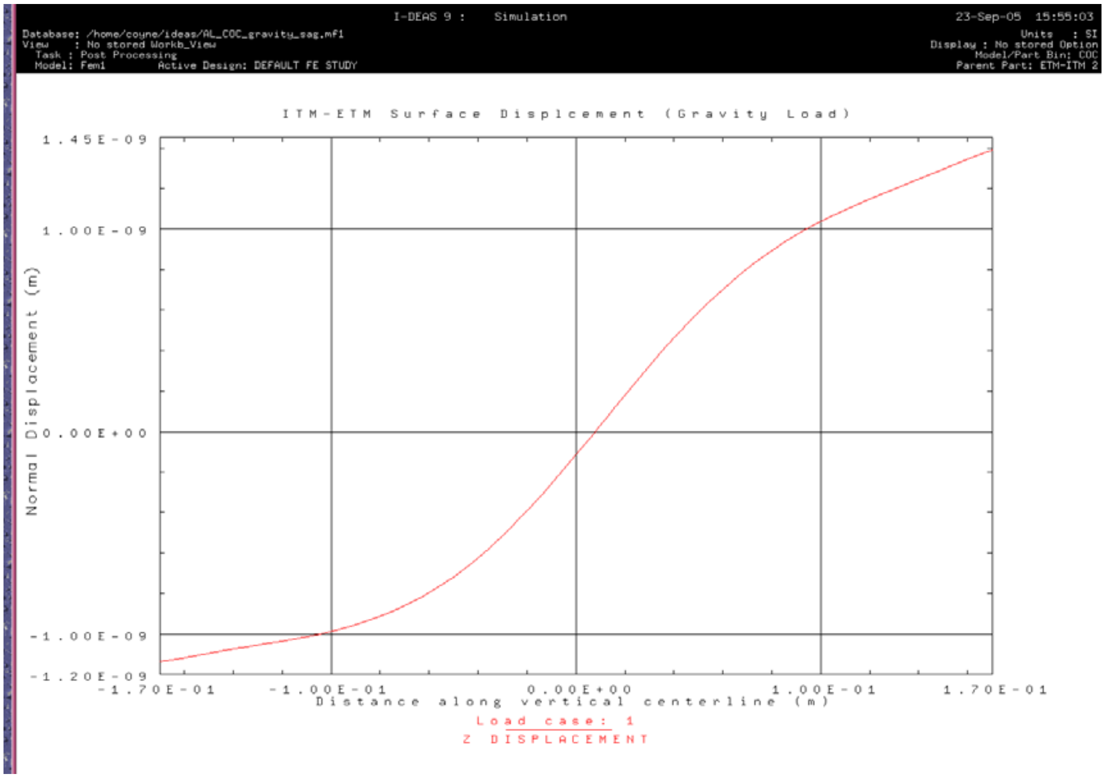
```
                         I-DEAS 9 :    Simulation                       23-Sep-05  15:34:49
Database: /home/coyne/ideas/AL_COC_gravity_sag.mf1                               Units   : SI
View    : No stored Workb_View                                  Display : No stored Option
  Task : Post Processing                                            Model/Part Bin: COC
  Model: Fem1         Active Design: DEFAULT FE STUDY               Parent Part: ETM-ITM 2

                        /home/coyne/ideas/AL_COC_gravity_sag.mf1
RESULTS: 1- B.C. 1,DISPLACEMENT_1,LOAD SET 1
DISPLACEMENT - Z MIN:-4.04E-09 MAX: 4.04E-09
DEFORMATION: 1- B.C. 1,DISPLACEMENT_1,LOAD SET 1
DISPLACEMENT - Z MIN:-4.04E-09 MAX: 4.04E-09
FRAME OF REF: PART
                                                                  VALUE OPTION:ACTUAL
```

Figure 6: Front & Rear Surface Normal Deformation

4 Comparison to optical surface figure requirements

The draft polishing specifications[7] call for a sagitta over the central 215 mm diameter of 2862 nm with a surface error of no more than 0.75 nm rms over the central 120 mm diameter. As can be seen from the vertical and horizontal centerline profiles (Figures 6 and 7), the predicted deformation due to gravity loading is ~1.2 nm p-v over the central 120 mm diameter. However, much of this deformation is local tilt (pitch).

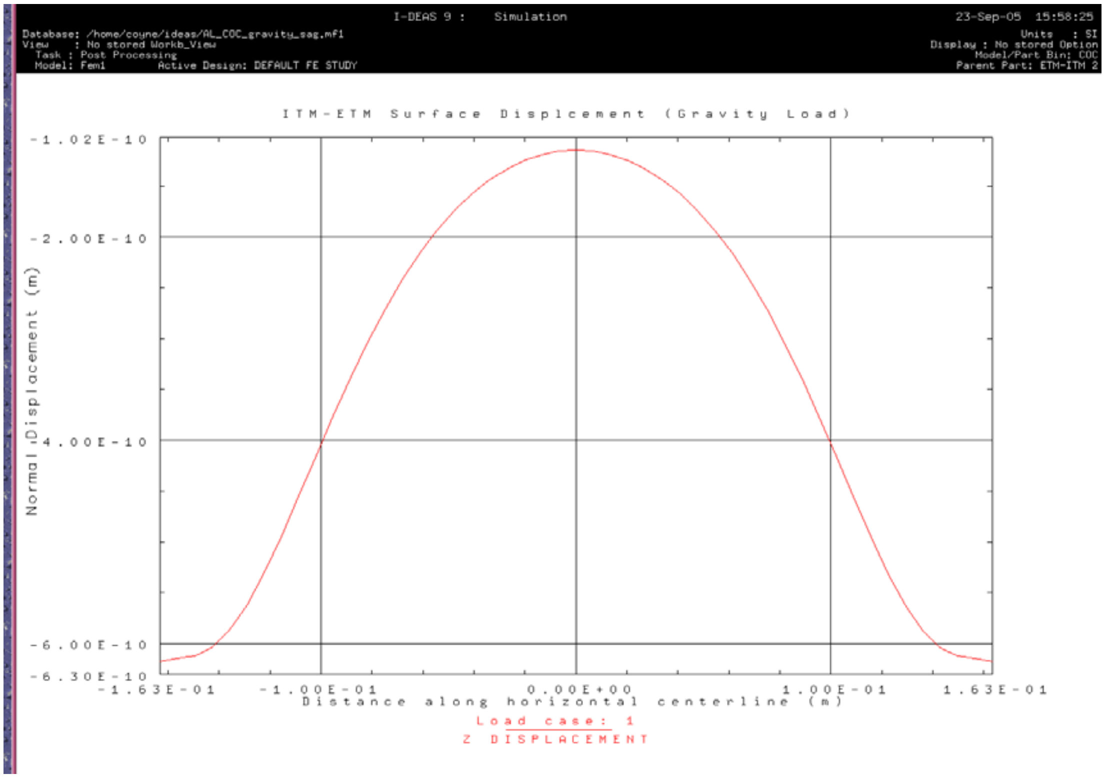


```
                         I-DEAS 9 :    Simulation                 23-Sep-05  15:55:03
Database: /home/coyne/ideas/AL_COC_gravity_sag.mf1                         Units   : SI
View    : No stored Workb_View                          Display : No stored Option
  Task : Post Processing                                         Model/Part Bin: COC
  Model: Fem1          Active Design: DEFAULT FE STUDY           Parent Part: ETM-ITM 2
```

Figure 7: Front Surface Normal Deformation Along the vertical centerline

---

[7] Current specification parameters are listed here: spectable.html
Also see Tables 1 and 2 of H. Armandula, et. al., COC CDD, LIGO-T000098-02, 20 Jun 2004.



```
                                   I-DEAS 9 :    Simulation                                      23-Sep-05  15:58:25
Database: /home/coyne/ideas/AL_COC_gravity_sag.mf1                                                         Units   : SI
View     : No stored Workb_View                                                              Display : No stored Option
  Task : Post Processing                                                                        Model/Part Bin: COC
  Model: Fem1          Active Design: DEFAULT FE STUDY                                        Parent Part: ETM-ITM 2
```

Figure 8: Front Surface Normal Deformation along the horizontal centerline

The results of the finite element model have been exported for use in further optical analysis, as described in the appendix.

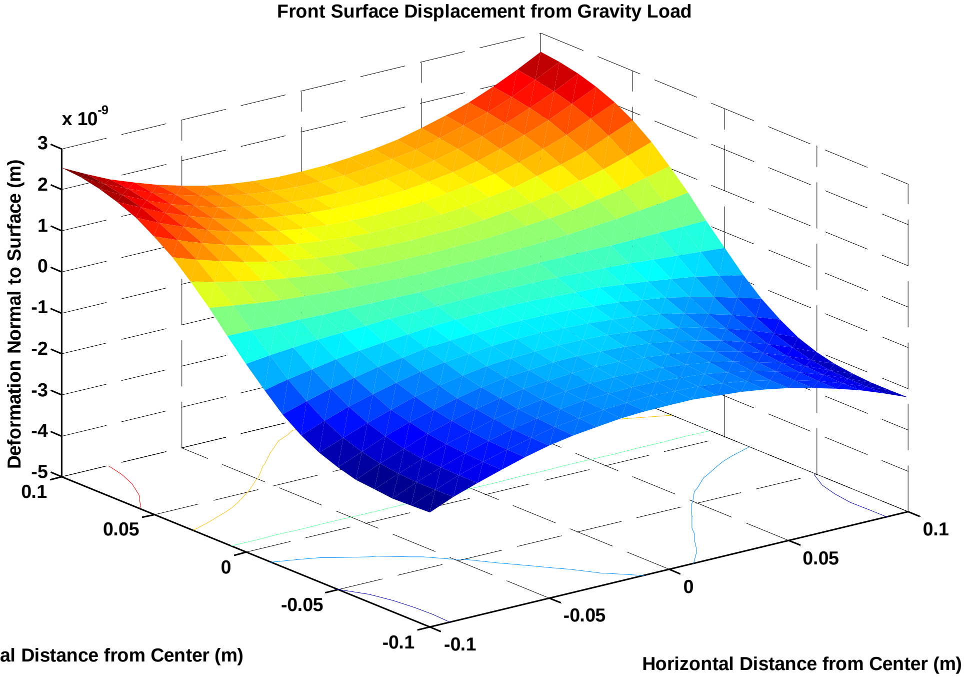


Figure 9: Interpolated optic surface deformation map

The irregular finite element nodal grid was interpolated to a regularly spaced grid for Zernike fitting in Matlab.

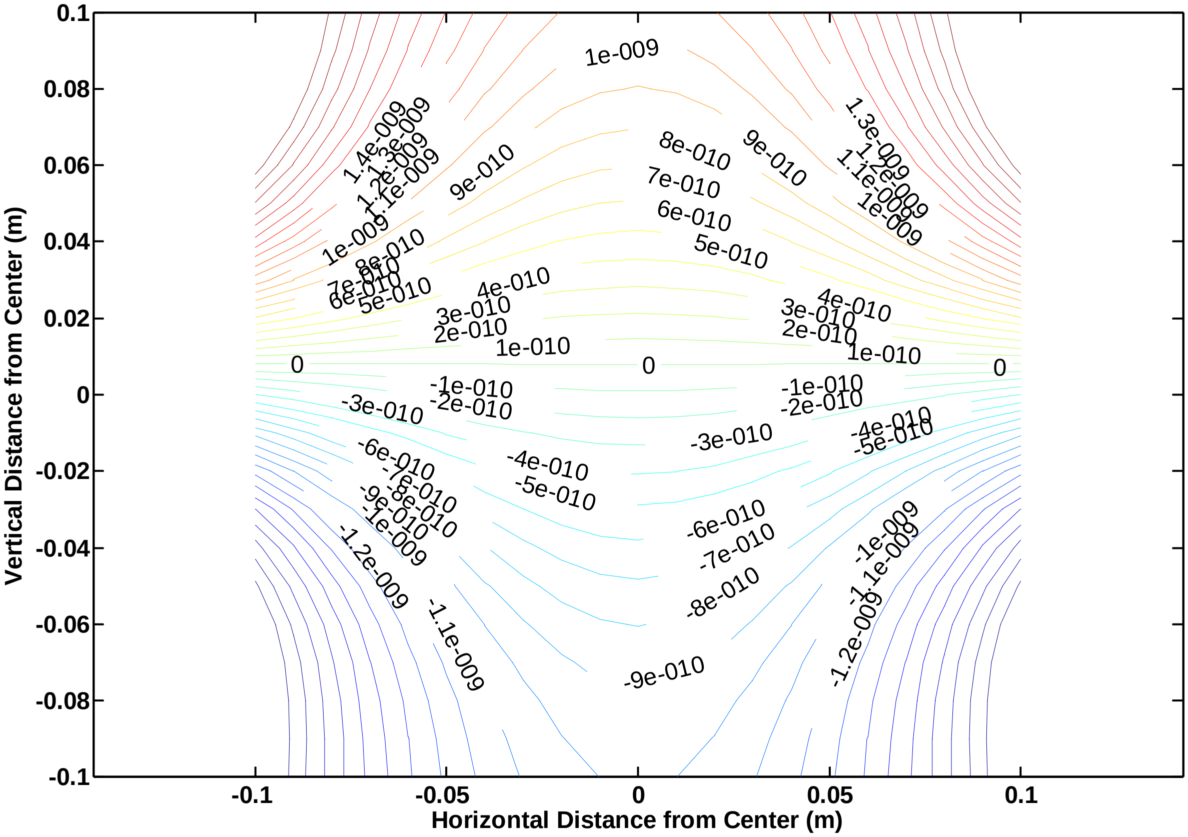


Figure 10: Contour plot of the surface deformation

Using Matlab to fit the surface deformation to Zernike aberration functions yields the following Zernike fits (nm):

```
Z1, piston           = -0.11455
Z2, yaw              = -6.1246e-007
Z3, pitch            = 0.90346
Z4, focus            = 0.0010491
Z5, astigmatism (0)  = -0.082277
Z6, astigmatism (45) = 8.7543e-008
Z7, x-Coma           = 2.897e-008
Z8, y-Coma           = 0.019566
Z9, spherical        = 0.0012452
```

The rms of the surface deformation, in the central 120 mm diameter, is 0.468 nm (1.555 nm p-v). However, after removing piston, tip & tilt, the rms is only 0.071 nm (and 0.420 nm p-v). Since this predicted value is small compared to the total allowable figure error (0.75 nm rms), the gravity induced surface deformation is not a problem.

5 Appendix: Surface Deformation Map from the FEA

The Matlab m-files for reading the finite element results from I-DEAS in the Universal File Format (UFF), described in T050125-00[8], were extended for this analysis, to handle static results. The readuff.m file is included in the T050184-01.zip file associated with this memo. Also included in the zip file is the Matlab m-file, importFEAnodal.m, used to call uffread.m and create the interpolated grid plots and perform the Zernike fit. importFEAnodal.m is listed below.

The results for the stress, strain and deformation fields (at all nodal points) are embedded in the associated universal file. From this file, with suitable extensions to readuff.m, one could calculate the birefringence as well.

---

[8] D. Coyne, "Transforming Finite Element Eigensolutions to State Space Models", LIGO-T050125-00, 23 Jul 2005.

```
% interpolateFEAresults.m
% Import Finite Element Analysis (FEA) nodal results
% Import Universal File Format (UFF) or UNV file data from I-DEAS
% 2005-09-24, D. Coyne
% Notes:
% 1) Using an extension of the UFF Read m-file, readuff.m, dated 9/24/2005
% 2) This version is written for the results of a specific ITM FEA;
%    adaptation/generalization for another finite element model requires
%    some editing.

[UffDataSets,Info,errmsg] = readuff('ITM_gravity_sag.unv');

if Info.nErrors ~= 0
    for ii=1:Info.nErrors
        disp(Info.errorMsgs{ii});
    end
end

% echo header information
iHeader=find(Info.dsTypes==151);
UffDataSets{iHeader}

% echo units information
iUnits=find(Info.dsTypes==164);
UffDataSets{iUnits}

% nodal information
iNodal=find(Info.dsTypes==2411);
UffDataSets{iNodal}

% permanent groups
iGroups=find(Info.dsTypes==2452);
UffDataSets{iGroups}
% in this case the dataset I want is the the 8th
% labeled "front surface nodes"
UffDataSets{iGroups}.groupName{8}
frontSurfaceNodes = UffDataSets{iGroups}.entityTag{8};
nFrontSuraceNodes = size(frontSurfaceNodes,2);

% nodal analysis information
iNodalResults=find(Info.dsTypes==2414);
UffDataSets{iNodalResults}
% in this case the first dataset has the displacements
% and is the one I want
nNodalSets = length(iNodalResults);

% node numbers
nodeNumbers = UffDataSets{iNodalResults(1)}.nodeNum;
nNodes = length(nodeNumbers);
```

```matlab

% extract front surface node group positions
for i=1:nFrontSuraceNodes
   pos(i)=find(UffDataSets{iNodal}.nodeLabel==frontSurfaceNodes(i));
   frontSurfaceNodePositions(i,:)=[UffDataSets{iNodal}.x(pos(i)),UffDataSets{iNodal}.y(pos(i)),UffDataSets{iNodal}.z(pos(i))];
end

% Plot Front Surface Nodal Pattern
figure(1)
plot3(frontSurfaceNodePositions(:,1),frontSurfaceNodePositions(:,2),frontSurfaceNodePositions(:,3),'go')
axis equal
title('Surface Nodal Pattern')
%print surface_nodes.ps -dpsc2

% extract front surface node group deformation
for i=1:nFrontSuraceNodes
   pos(i)=find(UffDataSets{iNodalResults(1)}.nodeNum==frontSurfaceNodes(i));
   frontSurfaceNodeDeform(i,:)=[UffDataSets{iNodalResults(1)}.r1(pos(i)),UffDataSets{iNodalResults(1)}.r2(pos(i)),UffDataSets{iNodalResults(1)}.r3(pos(i))];
end

% Plot Front Surface Deformation Shape
scale=0.05;
figure(2)
plot3(frontSurfaceNodePositions(:,1)+scale*frontSurfaceNodeDeform(:,1), ...
    frontSurfaceNodePositions(:,2)+scale*frontSurfaceNodeDeform(:,2), ...
    frontSurfaceNodePositions(:,3)+scale*frontSurfaceNodeDeform(:,3),'go')
axis square
grid on
title('Surface Deformation')
rotate3d on

% Interpolate from nonuniform nodal grid to a uniform grid & plot
    ngrid=51;
    w = 0.20;
    dx = w/(ngrid-1);
    xi = -w/2:dx:w/2;
    yi=xi';
    [xi,yi,zi]=griddata(frontSurfaceNodePositions(:,1),frontSurfaceNodePositions(:,2),frontSurfaceNodeDeform(:,3),xi,yi);
    figure;
    surfc(xi,yi,zi)
    shading interp
    title(strcat('Front Surface Displacement from Gravity Load'))
    xlabel('Horizontal Distance from Center (m)');
    ylabel('Vertical Distance from Center (m)');
    zlabel('Deformation Normal to Surface (m)')
```

```
    rotate3d on

    figure;
    v = -2e-9:0.1e-9:2e-9;
    [C,h] = contour(xi,yi,zi,v);
    axis equal;
    clabel(C,h)
    xlabel('Horizontal Distance from Center (m)');
    ylabel('Vertical Distance from Center (m)');

% least squares fit to Zernikes
    xf=reshape(xi,1,ngrid^2);
    yf=reshape(yi,1,ngrid^2);
    zf=reshape(zi,1,ngrid^2);
    rad = sqrt(xf.^2 + yf.^2);
    theta = atan2(yf, xf);
    % choose only points in the central region !!!!
    radiusCentral = 0.06;
    disp(['central region radius = ',num2str(radiusCentral,3),' (m)']);
    include=find(rad <= radiusCentral);
    radiusNormalize = min(radiusCentral,max(rad));
    r = rad(include)/radiusNormalize;
    t = theta(include);
    z = zi(include);

    % calculate the mean and rms in the central region
    zMean = mean(z);
    zPV = max(z) - min(z);
    zRms = norm(z)/sqrt(length(z));
    disp(['mean surface deformation in the central region = ',num2str(zMean,5),' (m)'])
    disp(['p-v surface deformation in the central region = ',num2str(zPV,5),' (m)'])
    disp(['rms surface deformation in the central region = ',num2str(zRms,5),' (m)'])

    % Zernike decomposition
    A = [ones(size(r)); r.*cos(t); r.*sin(t); -1+2*r.^2; r.^2.*cos(2*t); r.^2.*sin(2*t); ...
            (3*r.^2 - 2).*r.*cos(t); (3*r.^2 - 2).*r.*sin(t); 6*r.^4 - 6*r.^2 + 1]';
    Ainv = pinv(A);
    zernf = z*Ainv';
    disp('Zernike fits (nm):')
    disp(['Z1, piston           = ',num2str(zernf(1)*1e9,5)])
    disp(['Z2, yaw              = ',num2str(zernf(2)*1e9,5)])
    disp(['Z3, pitch            = ',num2str(zernf(3)*1e9,5)])
    disp(['Z4, focus            = ',num2str(zernf(4)*1e9,5)])
    disp(['Z5, astigmatism (0)  = ',num2str(zernf(5)*1e9,5)])
    disp(['Z6, astigmatism (45) = ',num2str(zernf(6)*1e9,5)])
    disp(['Z7, x-Coma           = ',num2str(zernf(7)*1e9,5)])
    disp(['Z8, y-Coma           = ',num2str(zernf(8)*1e9,5)])
```

```
disp(['Z9, spherical        = ',num2str(zernf(9)*1e9,5)])

% remove piston, tip, tilt
zCorrected = z - zernf(1:3) * A(:,1:3)';
zCorrectedMean = mean(zCorrected);
zCorrectedPV = max(zCorrected) - min(zCorrected);
zCorrectedRms = norm(zCorrected)/sqrt(length(zCorrected));
disp(['mean (piston,tip & tilt removed) = ',num2str(zCorrectedMean,5),' (m)'])
disp(['p-v (piston,tip & tilt removed)  = ',num2str(zCorrectedPV,5),' (m)'])
disp(['rms (piston,tip & tilt removed)  = ',num2str(zCorrectedRms,5),' (m)'])

```