Optical Aberrations due to a 1W laser

Eduardo Marin

Gemini Observatory
+56 51 2205 648

December 11, 2016

I Introduction

In an ideal case an optical element is held in thermal equilibrium and has the same temperature at all point. In practice though a thermal gradient might be seen. This influx of thermal power will deform the surface of the optical element for most common used materials. One such source of this thermal power can be from a laser. From a design perspective it is essential that the designer study the thermal effects from the laser, and decide on a material and coating that will withstand the power without excessively deforming..

II Effects of Thermal change

In order to see why we have to consider these effects lets examine what changes materials undergo as the temperature changes. All materials have a coefficient of thermal expansion α which is given in parts per million per degree centigrade. This is inherent to the material and it is an important element of design to select a material that will work for you. The higher this value the more your material will deform based on a temperature change. For optical systems we also have to look at the change of index of refraction with temperature, but for this tutorial will we only look at examining the thermal distortion using a Finite Element Analysis. Equation 1 shows the simple liner deformation of a material when exposed to a temperature shift.

$$\Delta L = L\alpha\Delta T \qquad (1)$$

As the coefficient of thermal expansion is different for each material it is important to model how different materials change with different temperatures. In our case we will look at a 3mm diameter 1W laser beam and its effects on a flat mirror

III Introduction to SolidWorks

Solidworks is a commercially available 3D modeling software. Within this software tool it is possible to create opto-mechanical systems from scratch or to import models that are made commercially. Additionally you can select the material of the design elements in order to use them in a design analysis.

IV FEA introduction

FEA (Finite Element Analysis) is a process of turning one large problem into a series of small problems. This is done by dividing an object into many smaller objects (elements) connoted by nodes, the nodes act as connectors between the object elements. The amount of elements you divided into is finite but it is many more than can be done by hand. The result of this division is a matrix equation given as equation 2

$$[K]\{u\} = \{F\} \qquad (2)$$

With K being the parts stiffness u being the displacement and F being force. This can be solved for the displacement as:

$$\{u\} = [K]^{-1} \bullet \{F\} \qquad (3)$$

Doing this kind of analysis allows us to model complex devices to see how the change due to thermal effects. Figure 1 shows an example of the elements used. In solidworks it is possible to use tetrahedron and shell solid elements.

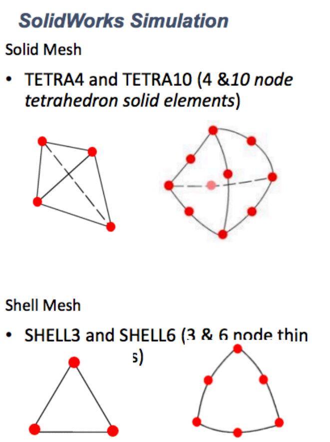

Figure 1: Solidworks FEA nodes

V FEA

For the purpose of this tutorial we will look at a flat mirror made from fused silica and see how it deforms from a laser heat power of 1W and 3mm radius on a 100mm diameter and 15mm thick Fused Silica flat mirror. First we will make a model of the mirror made of Fused Silica in solid works. Lets first design the flat mirror. First design the mirror. The resultant mirror is shown in figure 2

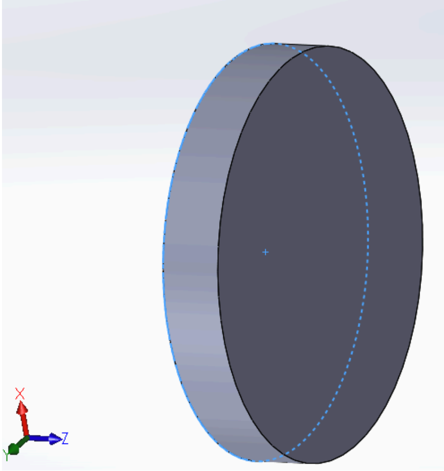

Figure 2: Flat mirror of radius 50mm thickness 15mm

Now that the mirror has been made the next step is to set the appropriate boundary constraints in order to do the FEA. Lets start by making a new study in solidworks and doing a thermal study to add our thermal gradient. We will set the laser power to be 1W with a 3mm diameter on the front surface and set the back surface fixed to room temperature of 295K.

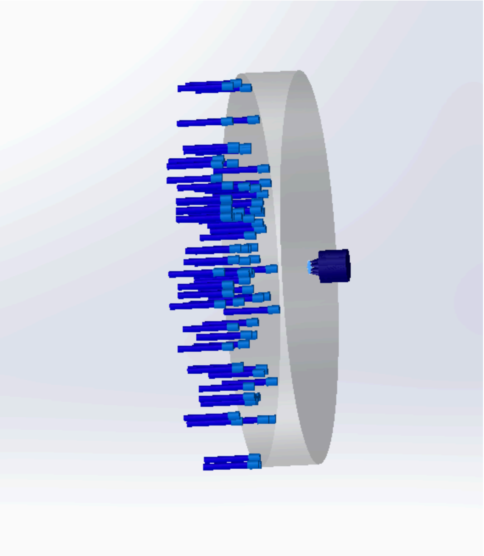

Figure 3: Applied Laser Power

The next step is to make a proper mesh for the FEA. This is making the elements and the nodes that were outlined in the above section on FEA. Using the default settings we can crate the following mesh.

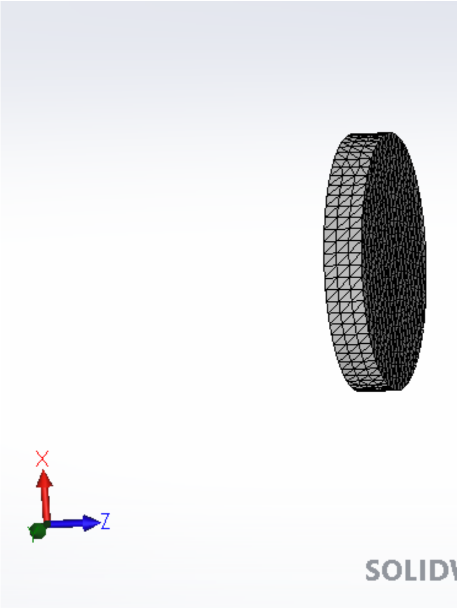

Figure 4: Mirror meshed for FEA

Then we can run the study to get the temperature in the mirror as a function of location.

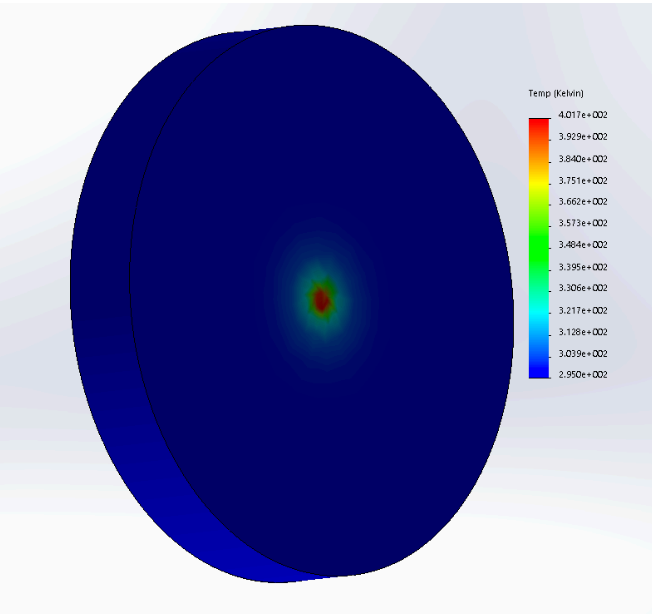

Figure 5: Results of Thermal FEA

With this information we are ready for mounting of the mirror. For the sake of simplicity we will use a 3-point support on the back face of the mirror. This kind of support provides rigidity in all degrees of freedom. After setting the correct mounting points and constraints you should now have an image in the back of the mirror as shown in figure 6

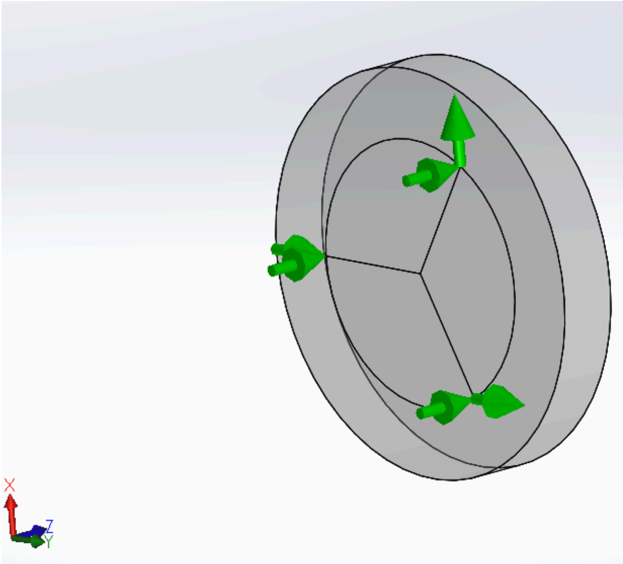

Figure 6: Mirror with mounting constraints

We again mesh the mirror to prepare for the FEA, and then run the FEA. We want to look at the mirror displacement in the X, Y, and Z planes.

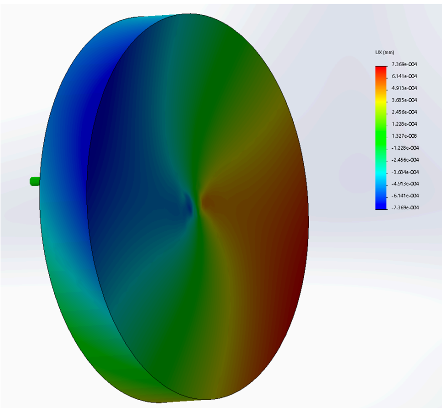
Figure 6: Resultant X displacement

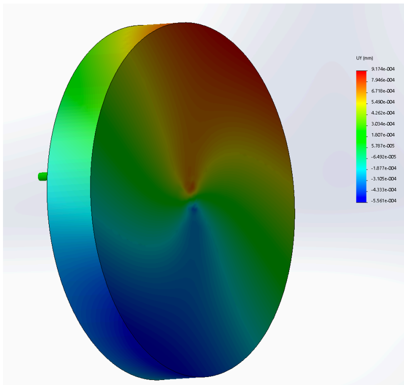
Figure 8: Resultant Y displacement

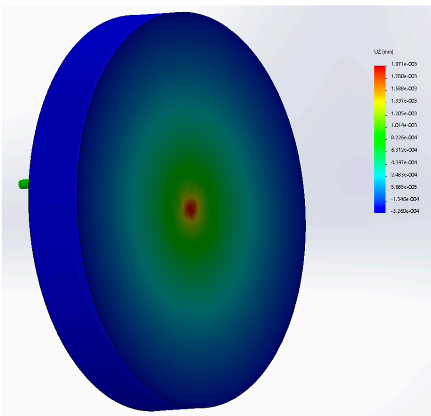
Figure 9: Resultant Z displacement

VI: Zernike analysis

Using an data processing code we can then use the displacement information to determine the shape of the mirror surface. For this tutorial the SUGUARO tool was used. Looking at the results we see that nearly 0.71 microns surface error of added to the flat mirror due to this laser power.

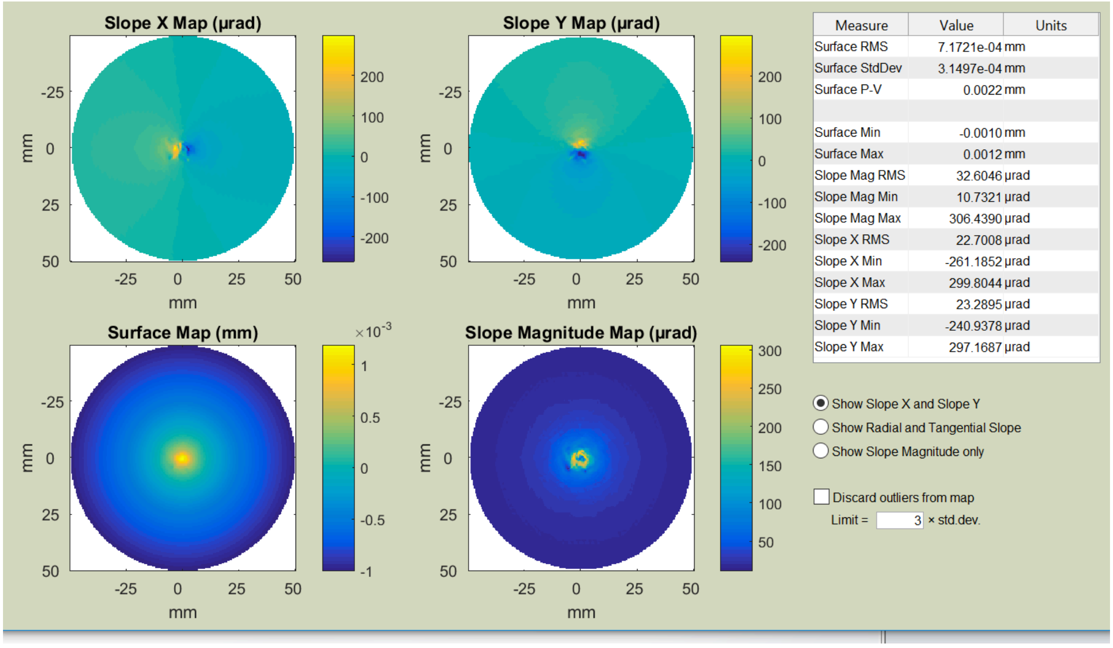

Figure 10: Resulting mirror surface

VII Conclusions

As we can see from above a laser on an optical element changes the nature of the surface and can add in additional aberrations that were not accounted for in the original model. In the very simple case that was presented in this tutorial only one material was tested. For a full model many different materials and coating should be looked at, as well as different mounting options. Also while this tutorial focused on the thermal deformation it is important to make sure the material can handle the laser power with out cracking or breaking.