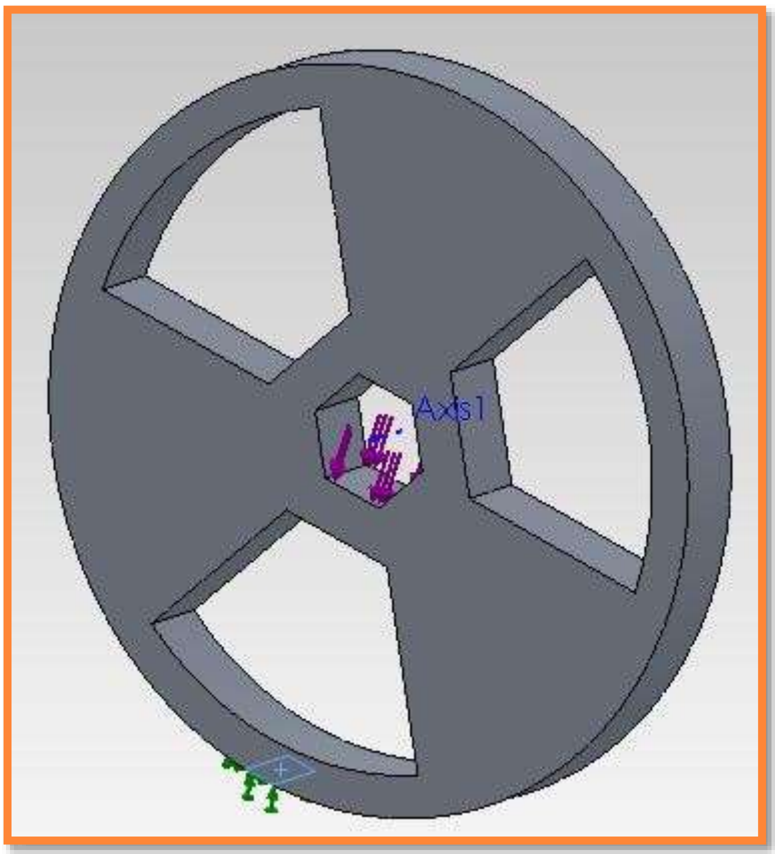
Finite Element Method (FEM) Analysis Report 2



Name: Wilson Lam

April 21, 2013

Table of Contents

LIST OF FIGURES

LIST OF TABLES

Objective

In this lab we used Finite Element Method (FEM) to test the flywheel using frequency and optimization testes. Afterward we will perform thermal test on a break disk and later a laser disk thermal test. By performing these testes we will obtain different modes of frequencies and its effect on the flywheel. Using the optimization test we will look for the best wheel thickness. As for the thermal test we can analyze the heat that the materials are heated up to. By completing these analyses we will become knowledgeable in frequency, optimization, and thermal analysis using Solidworks.

Flywheel Frequency Test

In this Finite Element Analysis (FEA) we are going to be using the Solidworks model as illustrated below to perform the analysis. This flywheel is assumed to be a wheel fixed on a car and experiencing resonance.

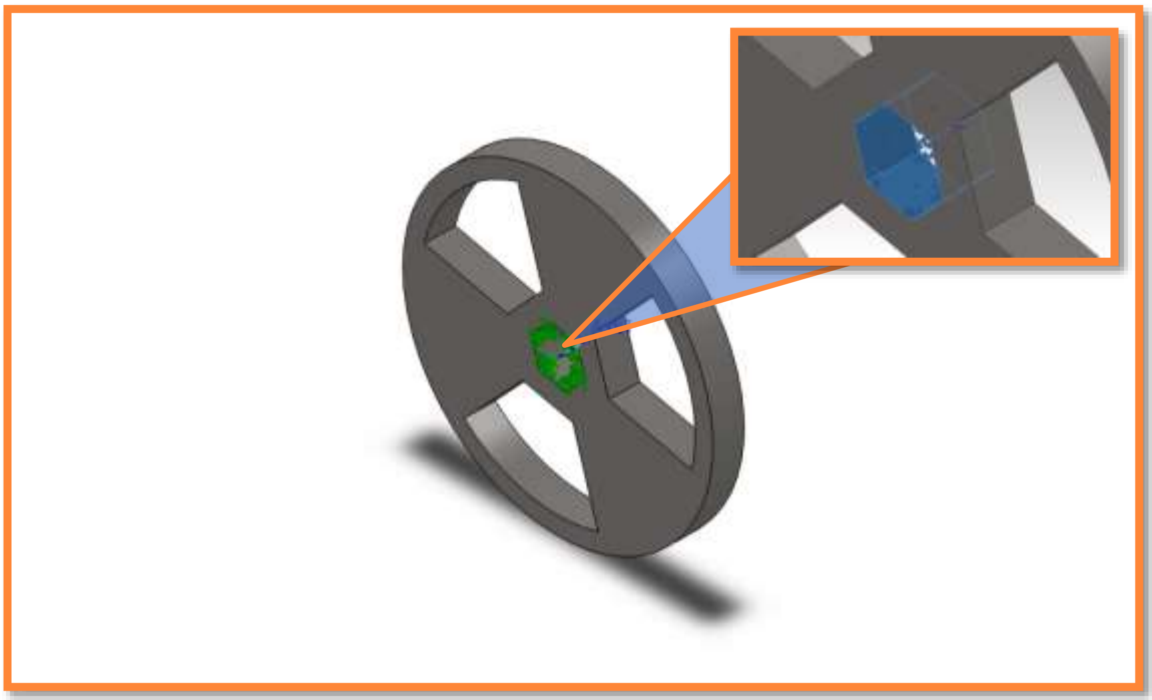


Figure 1: Solidworks Flywheel Model used in FEM analysis. (Fixtures shown in enlarged image.)

Assumptions

To begin the FEA we first have to make certain assumptions about the type of fixtures. We assume the restraints are geometric fixtures at the center of the flywheel preventing motion.

Table 1: Flywheel assumptions for FEA

Analysis type	Frequency
Material type	316 SS
Mesh type	Solid Mesh
Solver type	FFEPlus
Incompatible bonding options	Automatic
Fixtures	6 rectangular inner surfaces of flywheel

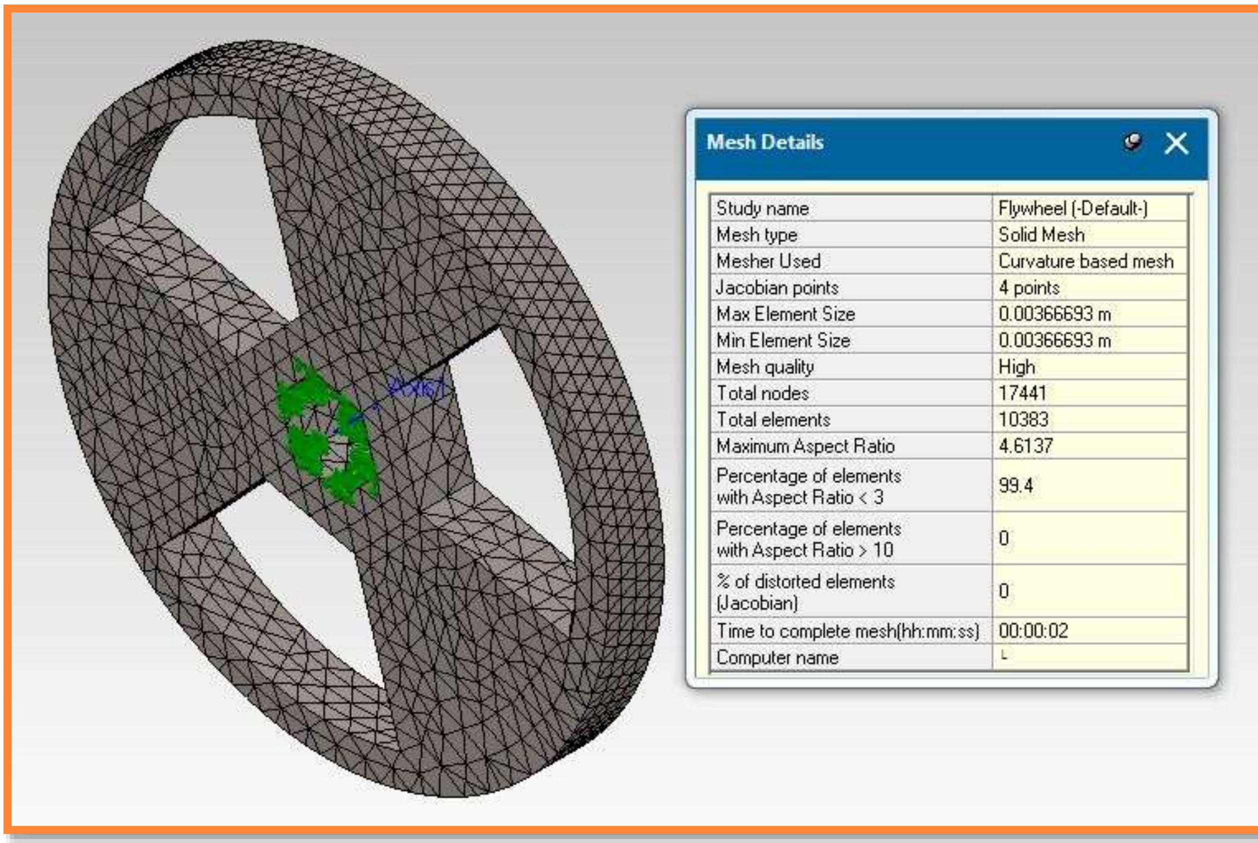


Figure 2: Meshed Model for frequency test and applied restraints (green).

FEA Results

Table 2: Mode and frequency tested data.

Frequency Number (Mode)	Hertz (Hz)
1	**2898.0**
2	**2898.9**
3	**3025.1**
4	**3766.9**
5	**4276.8**

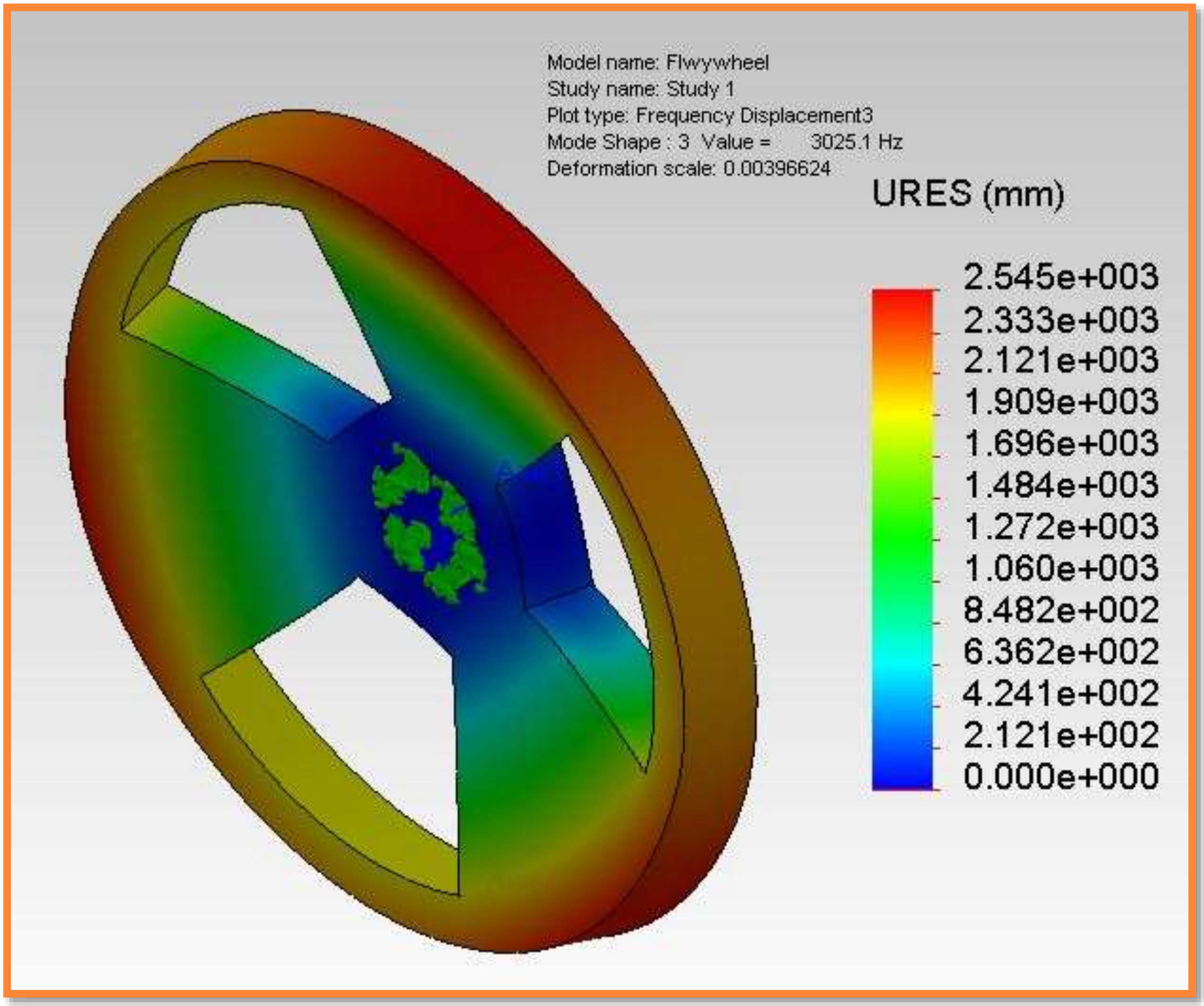


Figure 3: Flywheel Displacement for Frequency FEA at Mode 3.

Flywheel Optimization Study

In this Finite Element Analysis (FEA) we are going to be using the Solidworks model as illustrated below to perform the analysis. In this lab we assumed certain restraints and loads conditions.

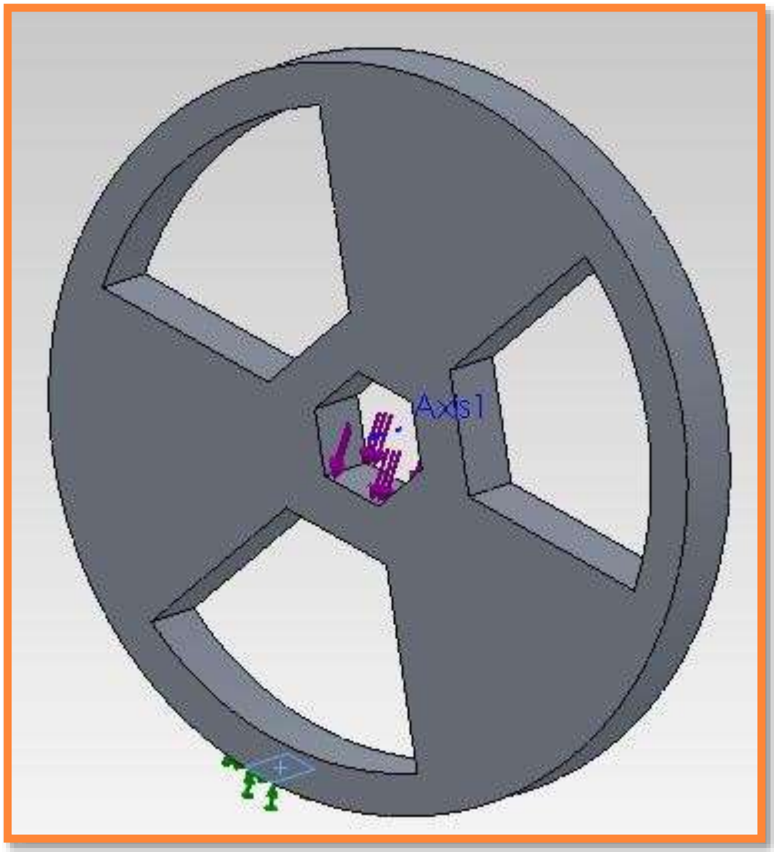


Figure 4: Flywheel Model used in FEM analysis with fixtures at the base.

Assumptions

To begin the FEA we first have to make certain assumptions about the type of fixtures, loads, and element analysis type. We assume the restraints are at the base of the flywheel and load of 70N to one of the inner surface. In this analysis we will be using solid mesh.

Table 3: Flywheel assumptions for FEA

Analysis type	Frequency
Material type	316 SS and Al-6061 (2 cases)
Mesh type	Solid Mesh
Solver type	FFEPlus
Incompatible bonding options	Automatic
Fixtures	Base of flywheel (green)
Loading	70 N on one inside surface (purple arrows)

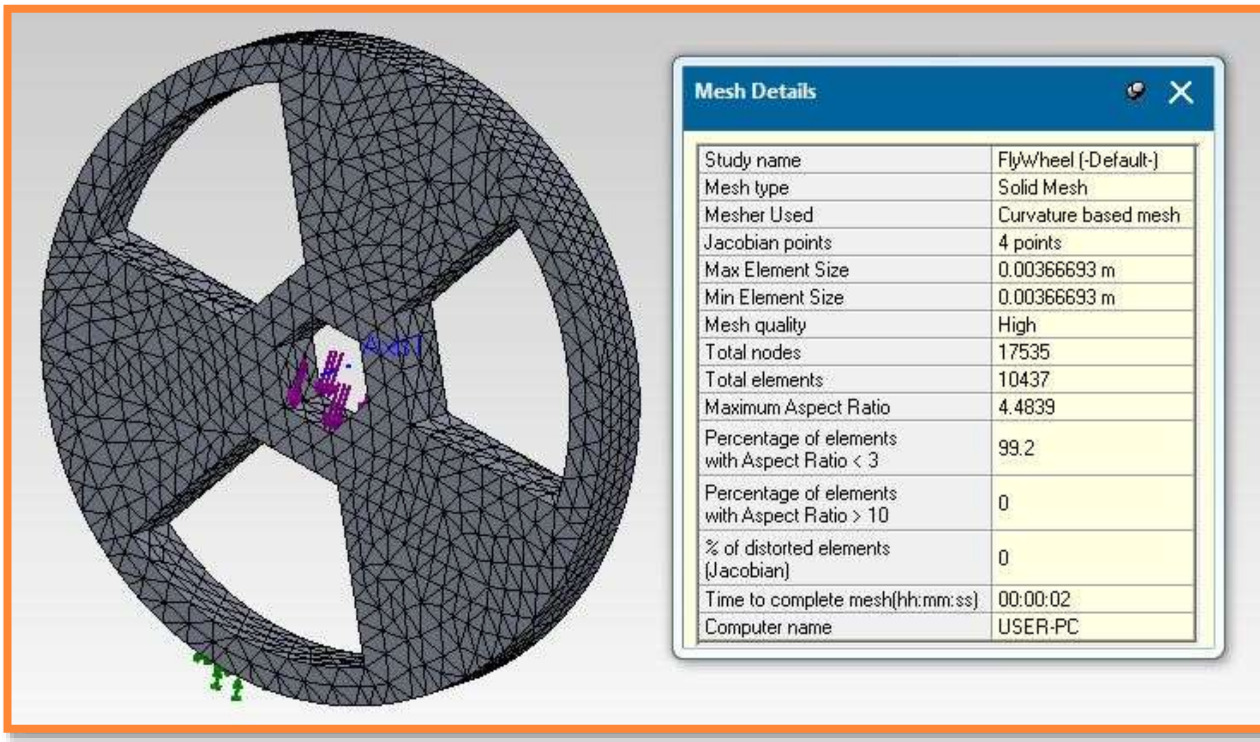


Figure 5: Flywheel meshed model with applied loads (purple) and restraints (green). (Meshed Details Table)

FEA Results

From performing the FEA optimization analysis I was able to obtain the following table below. It took my computer quite a long time to obtain these values. First I did it at step size around 5 mm then took it down to a lower range once I know that that the optimum thickness value is around 1mm where the material stress approaches the yielding stress for AL-6061. I perform the analysis for the 316 SS material in a similar fashion.

Table 4: Mode and frequency tested data.

Material	Thickness (mm)	Weight (g)
316 SS	**0.5 mm**	**19.7 g**
Al 6061	**0.9 mm**	**11.9 g**

In this analysis I used the thickness with the stress closest to the yielding stress so it calculate faster but overall it still took a long time. Apparently going to lower value crashes my computer (316SS) for some reason in general it is done similar to the Al-6061 which seem to work out fine.

Optimal (0)	Scenario 3	Scenario 4
0.0005m	0.0005m	0.0007m
88.27 N/m	88.27 N/m	65.748 N/
0.019703	0.019703	0.027584

Figure 6: FEM Optimization for 316SS at different thickness.

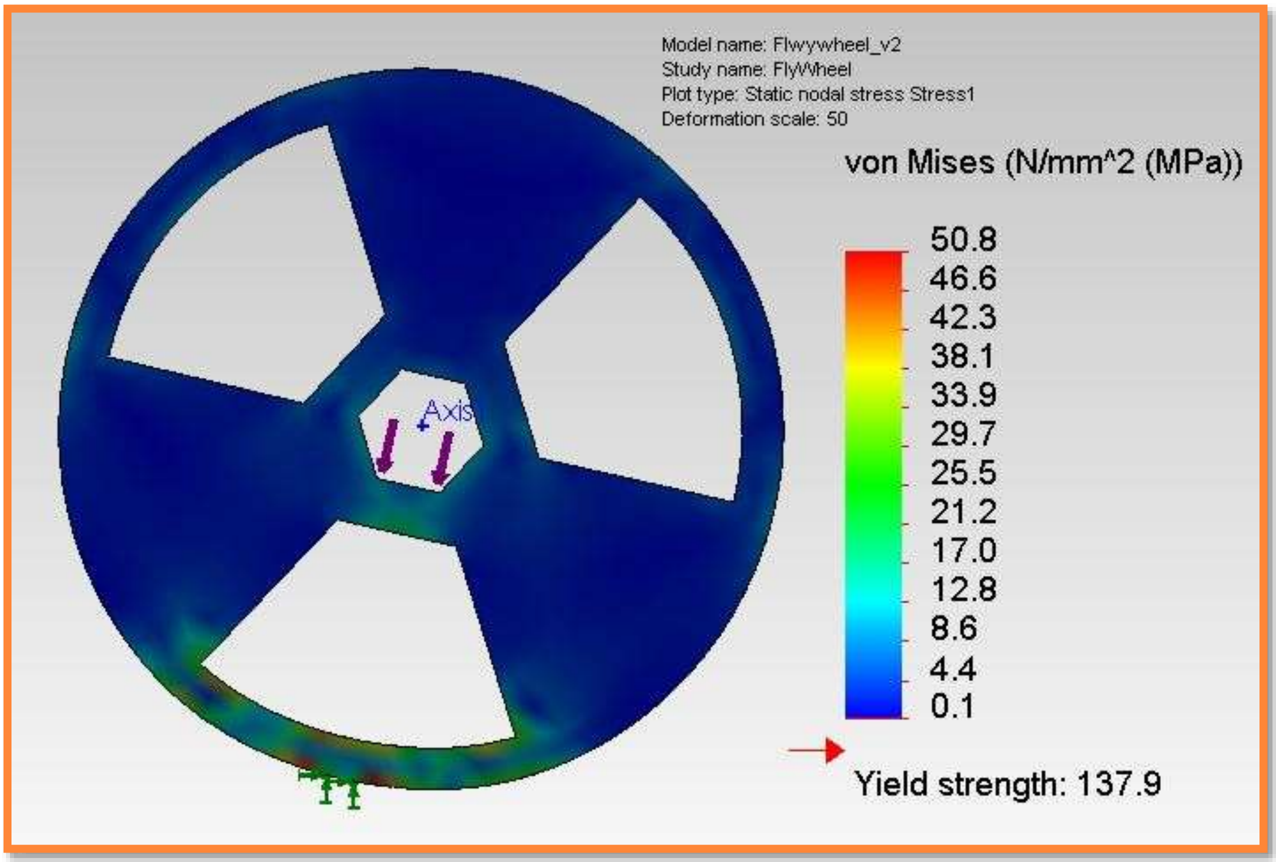


Figure 7: 316 SS stresses at 1mm thickness for Static FEA

Optimal (3)	Scenario 2	Scenario 3	Scenario 4	Scenario 5	Scenario 6
0.0009m	0.0008m	0.0009m	0.001m	0.0011m	0.0012m
54.323 N/mm^2	214.33 N/mm^2	54.323 N/mm^2	50.54 N/mm^2	46.777 N/mm^2	43.54 N/mm^2
0.0119697 kg	0.0106397 kg	0.0119697 kg	0.0132996 kg	0.0146296 kg	0.0159596 kg

Figure 8: FEM Optimization for Al-6061 at different thickness.

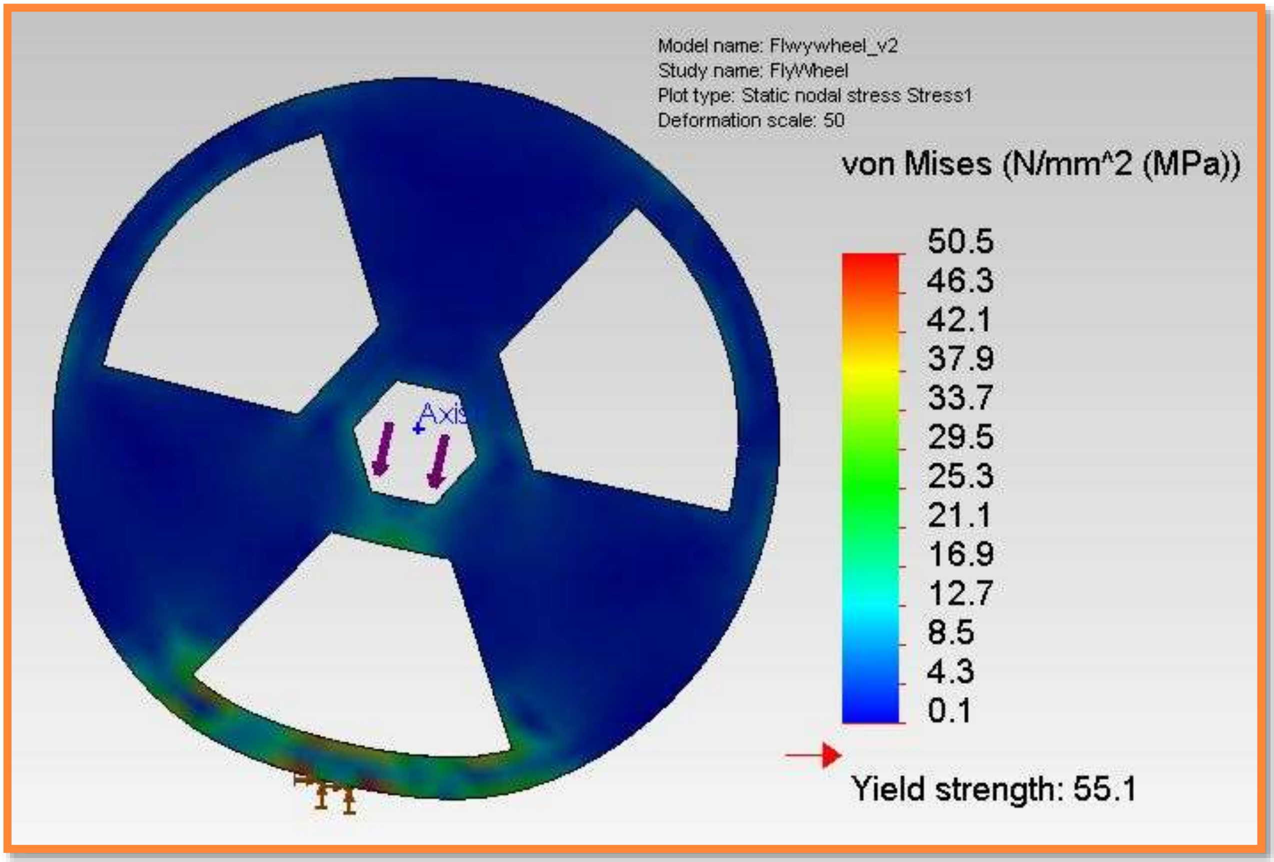


Figure 9: Al 6061 stresses at 1mm thickness for Static FEA

One important note during the analysis is that the stress for the Al-6061 is approaching its yield strength meaning the thickness would not be able to get any smaller by much. For 316 SS its yield strength is higher so the thickness could still get much smaller compare to Al-6061. The other note we need to make is that the mass for the 316 SS is larger than Al-6061 at the same thickness.

Break Disk Thermal Analysis

To begin the FEA we first have to make certain assumptions about the location of the heat flux and convection. In this analysis we are looking for possible the ranges of temperature.

Assumptions

In our analysis we will assume that the disk pad will create a heat flux on the surface of the break disk. The rest of the surface will then be exposed to convection. The only surface not exposed is the cylindrical side of the disk. We assume the worst case scenario where that location is insulated. Details of the surface with convection and heat flux are in the table below.

Table 5: Break disk assumptions for FEA

Analysis type	Thermal
Material type	316 SS
Mesh type	Solid Mesh
Solver type	FFEPlus
Incompatible bonding options	Automatic

Table 6: Thermal convection and heat flux surfaces details.

Thermal	Applied Surface	Details
Convection		**Entities:** 2 **face(s)** **Convection Coefficient:** **746 W/(m^2.K)** **Bulk Ambient Temperature:** **300 Kelvin**
Heat Flux		**Entities:** **2 face(s)** **Heat Flux Value:** **10000 W/m^2**

FEA Results

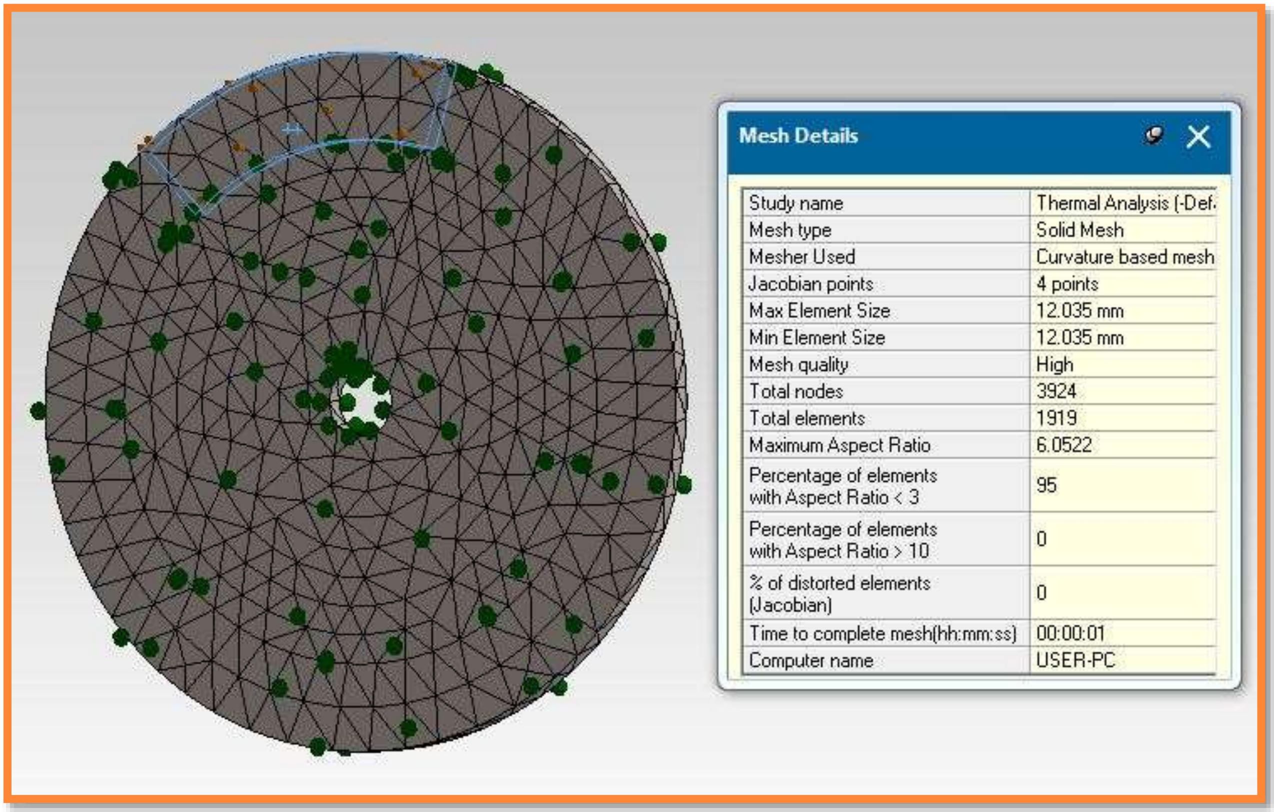


Figure 10: Meshed Model for thermal test heat flux (orange in blue outline) convection (green).

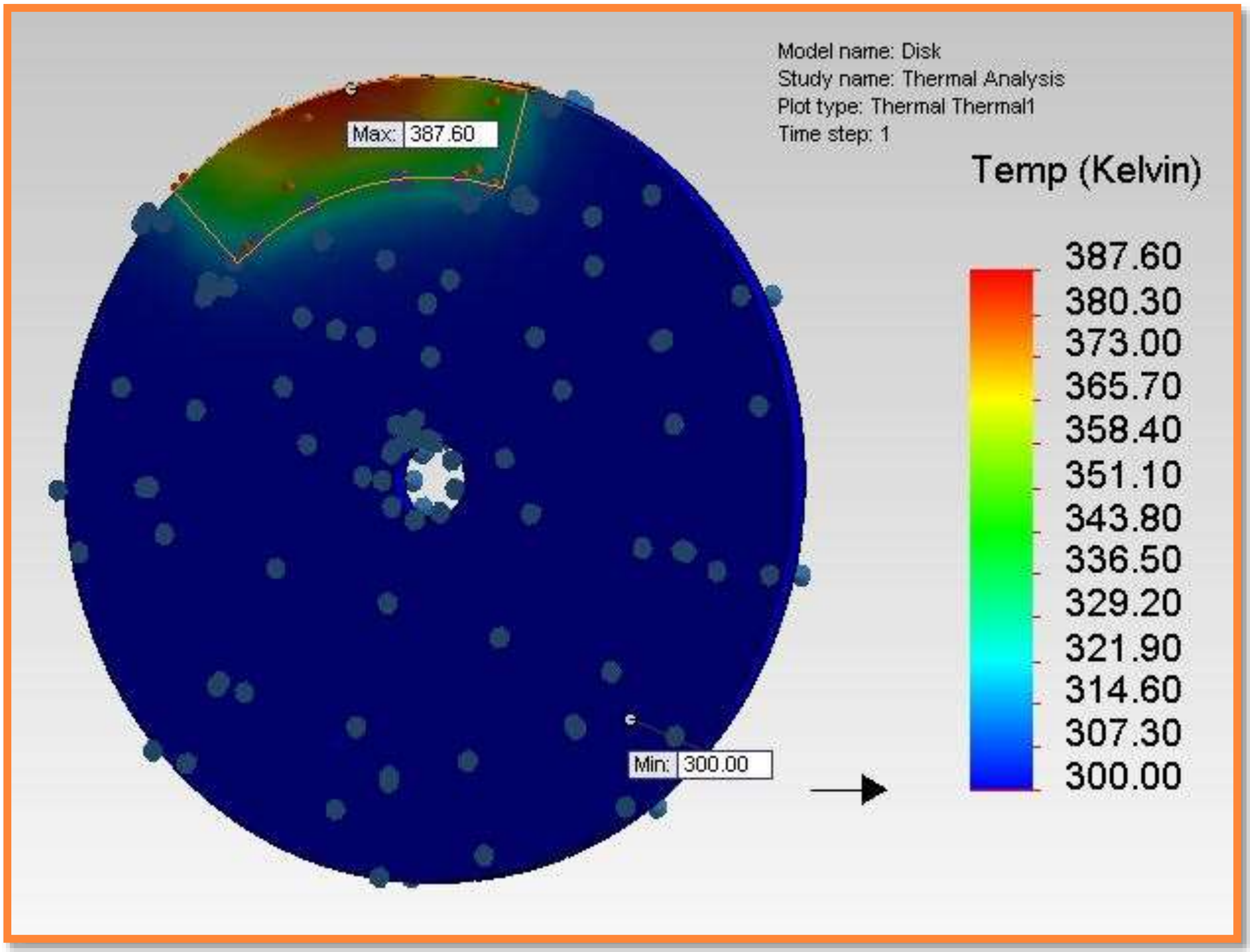


Figure 11: Thermal test heat flux (orange area) convection (light blue).

From the thermal analysis we obtain a maximum temperature of 387.60°K. This friction energy is enough to boil water!

Laser Disk Thermal Analysis

To begin the FEA we first have to make certain assumptions about the location of the heat flux and convection. In this analysis we are looking for possible the ranges of temperature and test to see if the materials will actually melt.

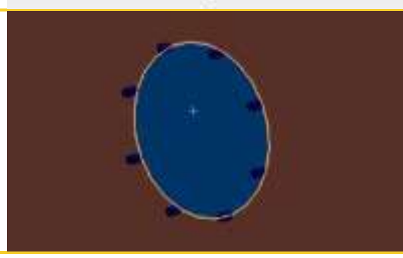
Assumptions

In our analysis we will assume that the laser will create a heat flux on the surface of the disk. The back surface will then be exposed to convection. The rest of the surface is assumed to be insulated. Details of the surface with convection and heat flux are in the table below.

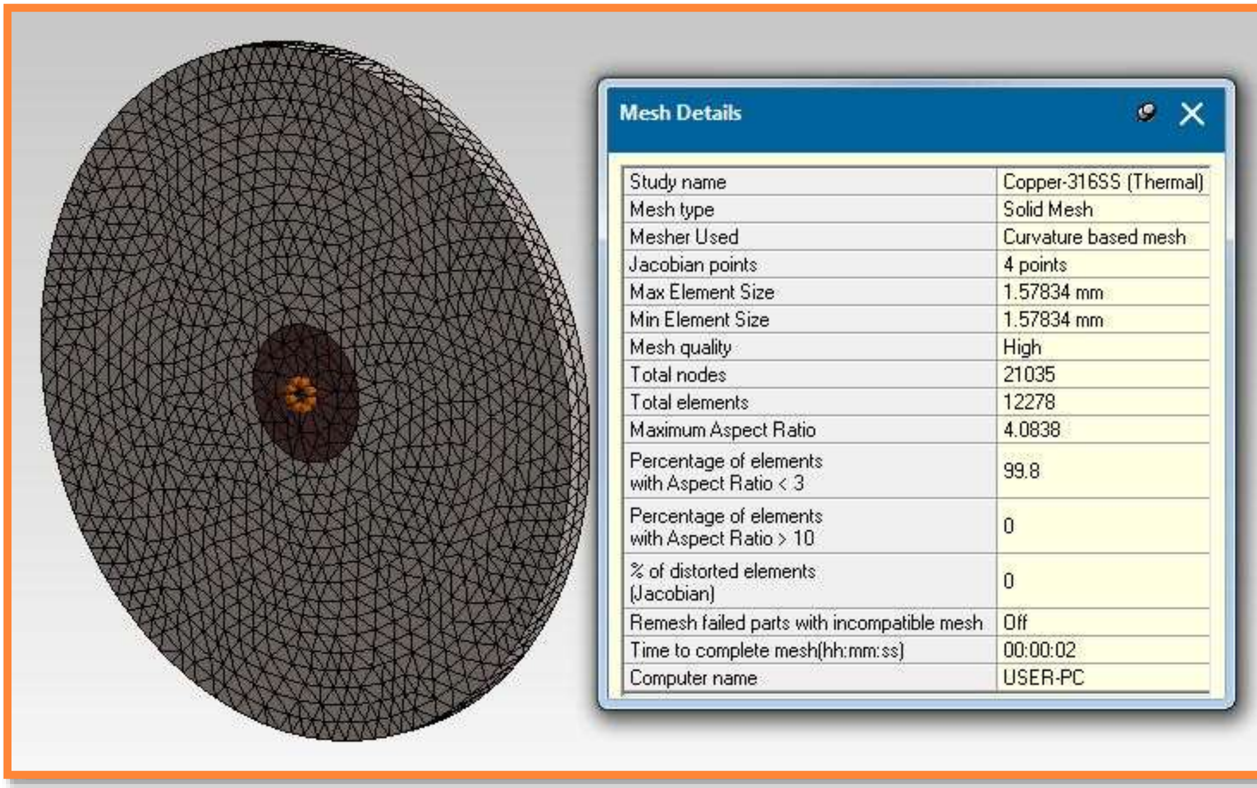
Table 7: Laser disk assumptions for FEA

Analysis type	Thermal
Material type	Copper (center disk) 316 SS (outer disk)
Mesh type	Solid Mesh
Solver type	FFEPlus
Incompatible bonding options	Automatic

Table 8: Thermal convection and heat flux surfaces details.

Thermal	Applied Surface	Details
Convection		**Entities:** 2 faces inner & outer disk **Convection Coefficient:** 500 W/(m^2.K) **Time variation:** Off **Temperature variation:** Off **Bulk Ambient Temperature:** 300 Kelvin **Time variation:** Off
Heat Flux		**Entities:** 1 face(s) **Heat Flux Value:** 300000 W/m^2 Note: The heat flux of 1.5 MW/m^2 is reduced by 80% due to reflectivity of the copper plate.

FEA Results



Study name	Copper-316SS (Thermal)
Mesh type	Solid Mesh
Mesher Used	Curvature based mesh
Jacobian points	4 points
Max Element Size	1.57834 mm
Min Element Size	1.57834 mm
Mesh quality	High
Total nodes	21035
Total elements	12278
Maximum Aspect Ratio	4.0838
Percentage of elements with Aspect Ratio < 3	99.8
Percentage of elements with Aspect Ratio > 10	0
% of distorted elements (Jacobian)	0
Remesh failed parts with incompatible mesh	Off
Time to complete mesh(hh:mm:ss)	00:00:02
Computer name	USER-PC

Figure 12: Meshed Model for thermal test heat flux (orange).

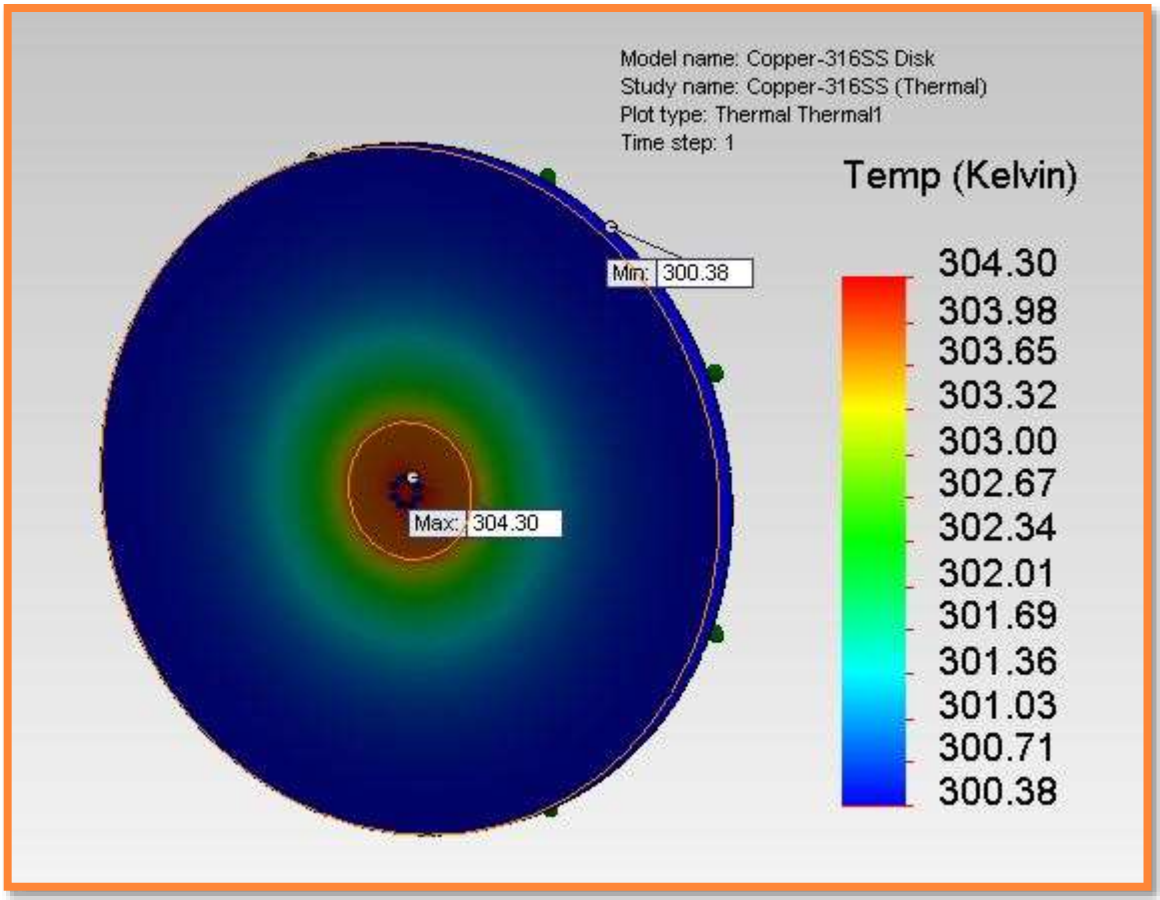


Figure 13: Thermal test heat flux (blue) convection (green).

From the analysis a rise in about $4°K$ is not enough to melt copper and 316 SS as the surrounding materials. The claim made by the friend is completely off. The laser is way too weak to produce enough heat to melt the materials.