MAE 598: Design Optimization

Instructor: Dr. Max Yi Ren

Project 2: Ansys Design Optimization

Made By: Monark Bandishbhai Parekh

1222179426

2022

Abstract

ANSYS offers structural models, transit thermal and various other analysis. It is a software which provides solution that enables engineers of all levels and backgrounds to solve complex structural engineering problems faster and more efficiently. ANSYS mechanical finite element analysis software is used to simulate computer models of structures, electronics, or machine components for analyzing strength, toughness, electricity, temperature distribution, electromagnetism, fluid flow and other attributes and sizes used to determine how product will function with different specifications without building test products or conducting crash tests.

For example, in this project the ANSYS software is used to optimize an already existing brake pad design to simulate how a brake pad will hold up under varying conditions.

Project Objectives

The objective of this project is to minimize the volume of the provided brake disc geometry, while maintaining it's structural, modal, and thermal integrity. The design variables of this brake disk are three geometric dimension parameters, i.e., inner diameter of the disc, outer diameter of the disc and thickness of the disc. The objective of the design is to have less strain value, higher vibration frequency, and low temperature. The design is tested to ensure safety under the following cases:

- Maximum Stress under Static loading
- Frequency of Free Vibration
- Maximum Temperature.

The optimization is done in ANSYS which has a built-in optimization module. This feature utilizes parameters that can change the geometry of the design. The updated design is then solved with the finite element analysis modules. A response surface is created with the simulation runs conducted and this surface is used to optimize for the best design using the method of choice.

Constraints

The design constraints mainly come from geometry, since the dimensions cannot be either too large or too small. However, the design of the brake pas is supposed to be safe under three different conditions, which results in additional constraints such as the following:

i. Maximum Temperature: The melting temperature of the gray cast iron is around 1100 °C, so by considering a factor of safety, the maximum temperature was restricted below 500 °C.

ii. Maximum Stress: The brake pads experience both compressive stress and shear stress, but the compressive strength of the brake pads is higher than the shear strength. Hence, the constraint applied on the maximum stress is based on the applied shear stress. Therefore, the maximum stress was restricted to be less than 14 MPa.

iii. Minimum Volume: The optimization problem dictates reducing the volume of the given model to be as low as possible, however volume physically cannot be negative. Hence, the minimum volume of the model is restricted to be 0.

iv. Maximum Natural Frequency: The goal is to have a very high frequency for the 7th mode to prevent failure, hence the target value for the modal frequency is regulated to be 1300 (Hz).

Workspace Setup

This section discusses the workspace setup in Ansys, and the different modules.

A brake disk part model is obtained from A. Durgude, A. Vipradas, S. Kishore, and S. Nimse provided by Dr. Yi Ren for the purpose of this project. The brake file (.agdb) is then used as the input to the ANSYS workbench which is later used by different analysis modules. Three modules that are used for different analysis purposes are:

I. Static mechanical analysis
II. Modal analysis
III. Transient thermal analysis

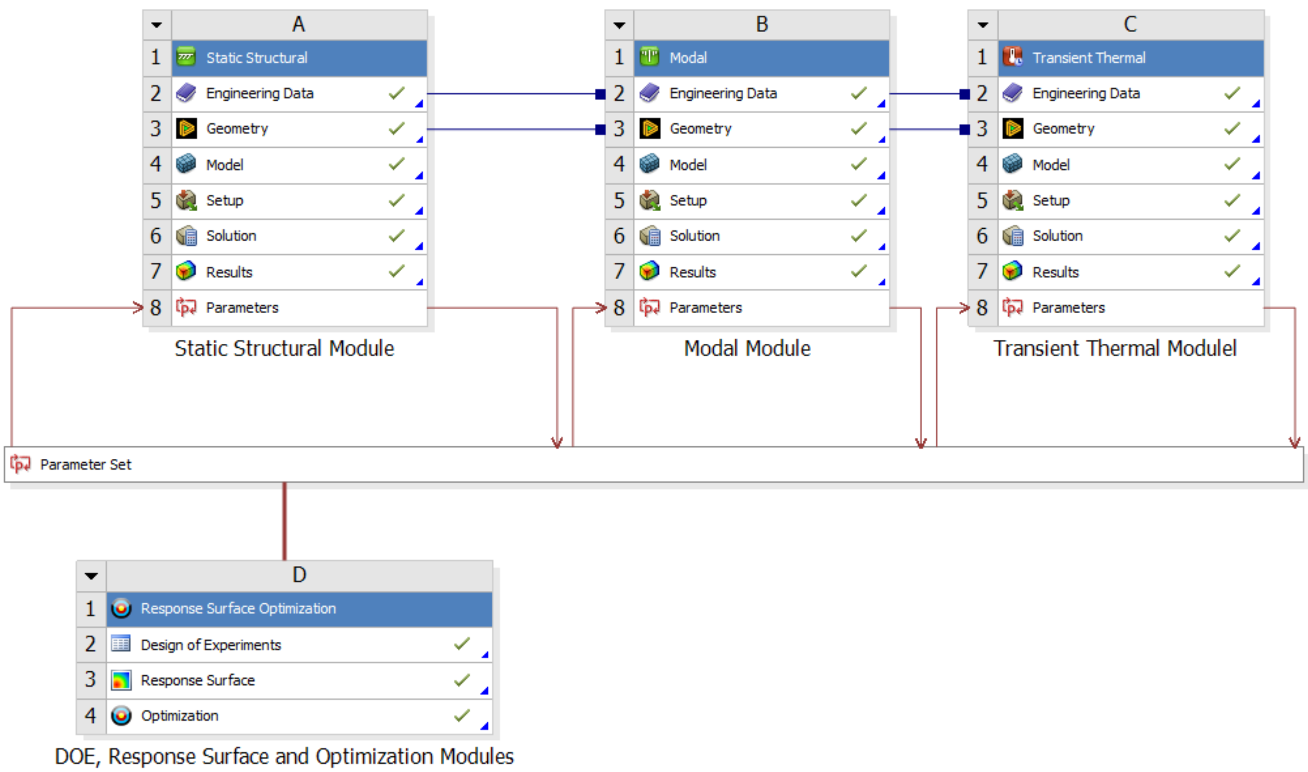

Figure 1: Workspace

Model Geometry

The model consists of three parts, namely the brake disc, upper brake pad and the lower brake pad. The given model has pre-defined dimensions that are to be optimized using the various modules. The geometry of the model is imported into the static structural analysis, by selecting "Geometry". This geometry is further re-used for analysis static structural, modal, and transient thermal modules.

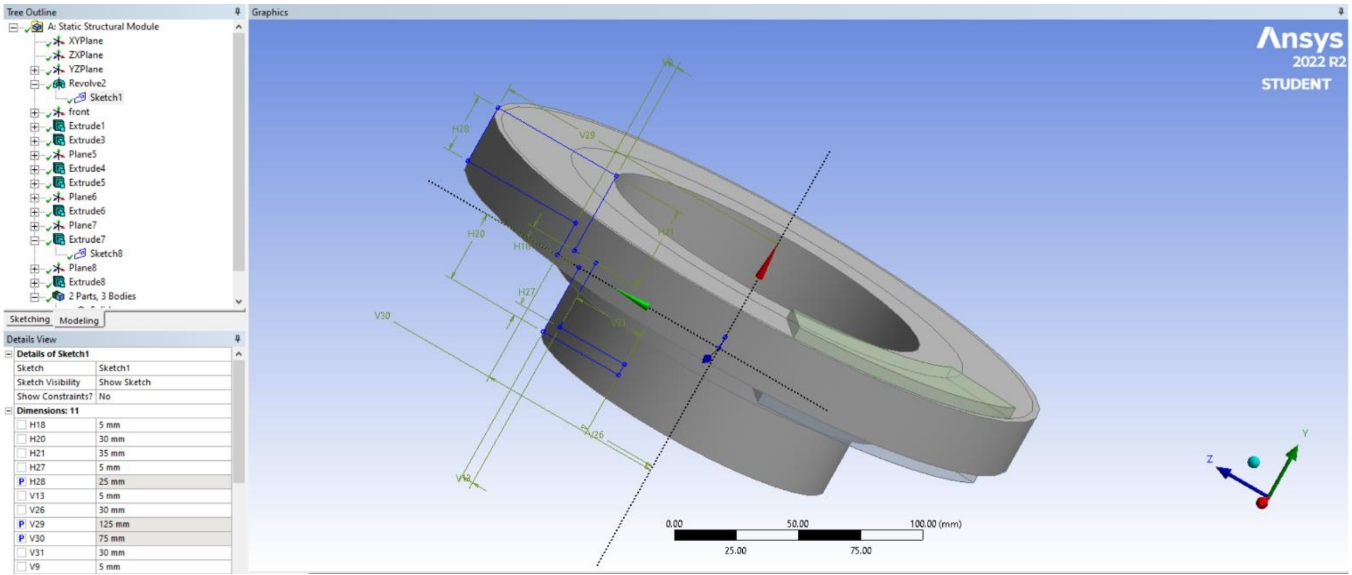

Figure 2: Model Geometry

Static Structural Analysis

Material Assignment

As described in the tutorial, the brake pads are assigned structural steel, while the brake disk is made of gray cast Iron.

Mesh

The meshing of the given model includes a lot of separate sizing for different sections in order to accommodate the size differences of different sections of the geometry. In order to obtain better results, patch conforming algorithm is used for the mesh with the tetrahedrons method. The following mesh is used for the different sections:

- The effective area is a very important zone since most load is applied in that region. Hence an element size of 0.005 m is chosen for the disc, and similarly, an element size of 0.004 m is chosen for brake pads
- The region that is not in the effective area is not of high importance, thus it is meshed with an element size of 0.01 m. Since this part is not in the region, which is subject to applied load, we can take liberty to increase the mesh size, thereby effectively decreasing the computation time with little to no effect on the results.

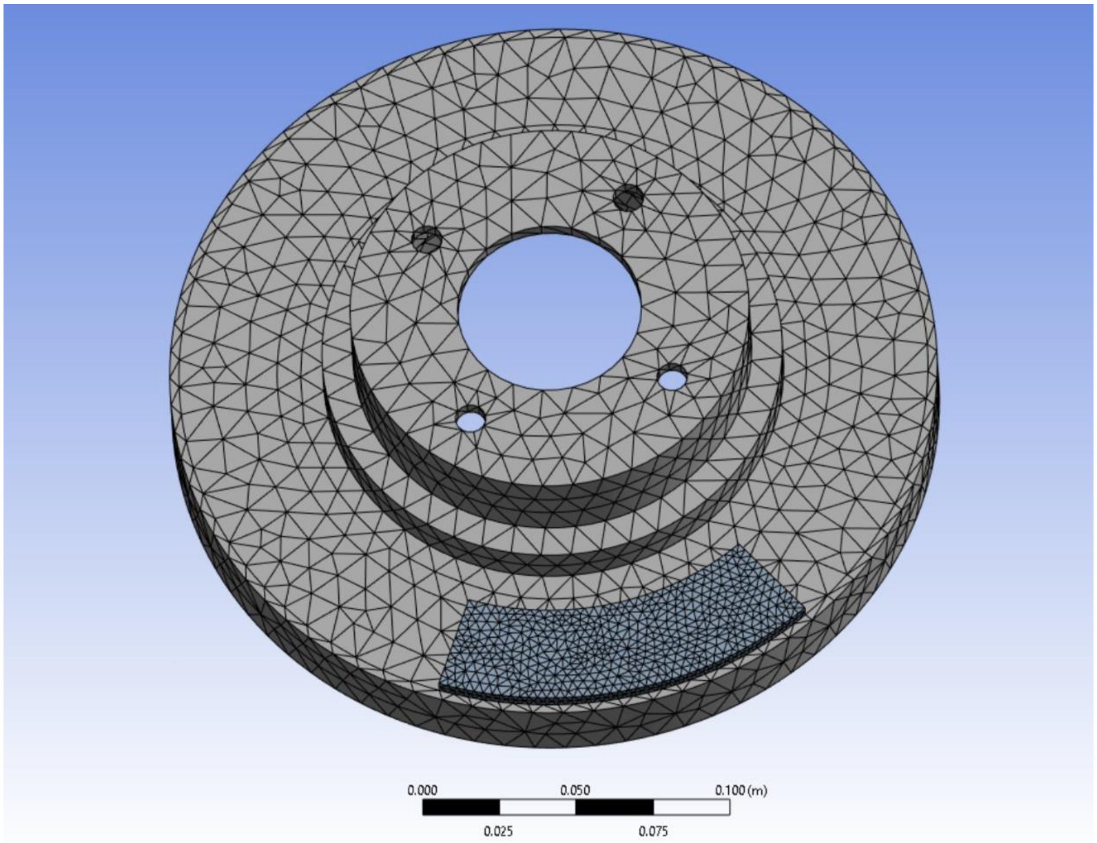

Figure 3: Mesh

Parameter Values

As described in the given tutorial the following parameter values are selected:

- Rotational Velocity → 250 [rad/s]
- Pressure → 10495000 [Pa] (ramped)
- Output Parameter → Equivalent Von-Mises Stress

Results

The static structural analysis was used to determine the Equivalent Von Mises stress on the brake disc. The stress distribution is shown below:

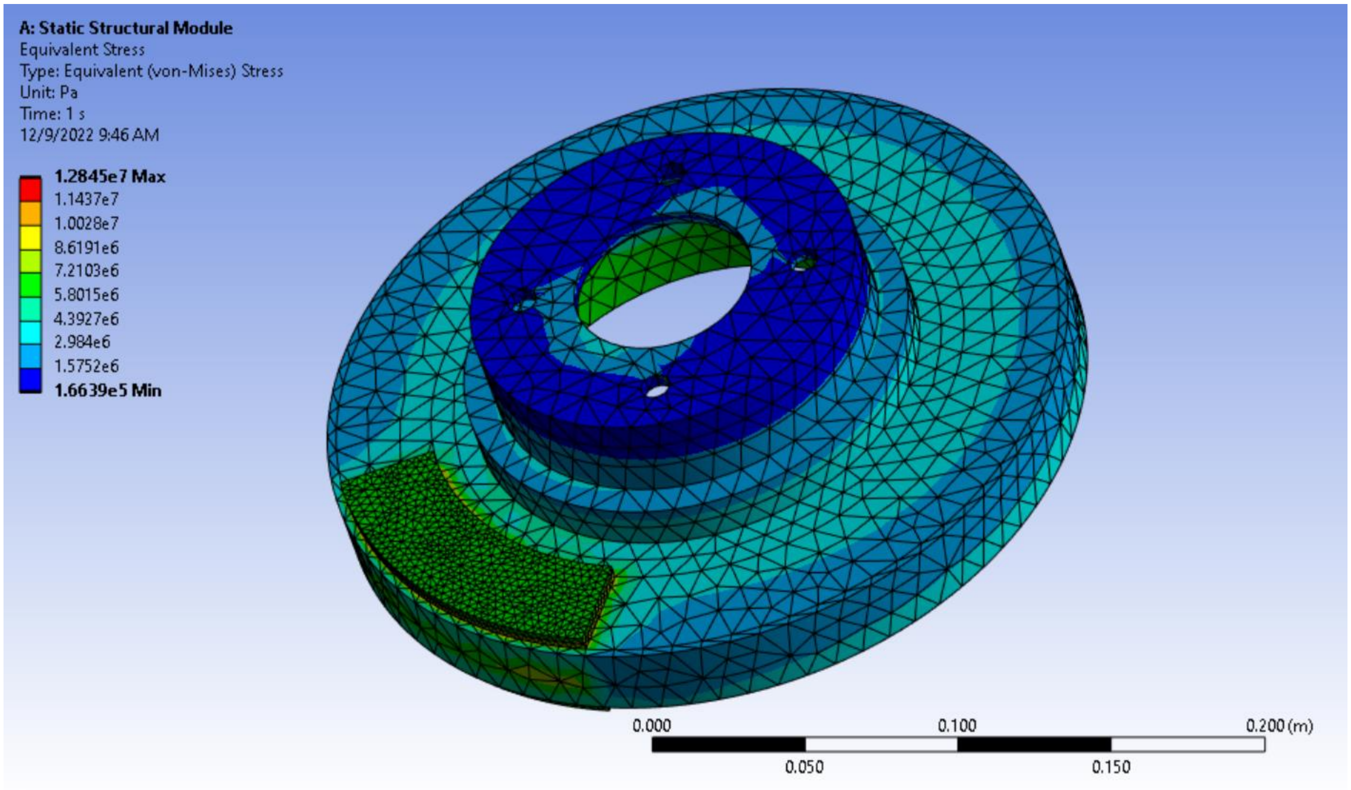

Figure 4: Static Structural Analysis Result

As seen in the above figure, the Equivalent Von Mises stress on the brake disc is 12.845 [MPa], before optimization.

Modal Analysis

Geometry Change

To undertake modal analysis, we only require the brake disc geometry. Hence, the brake pads are suppressed in the model.

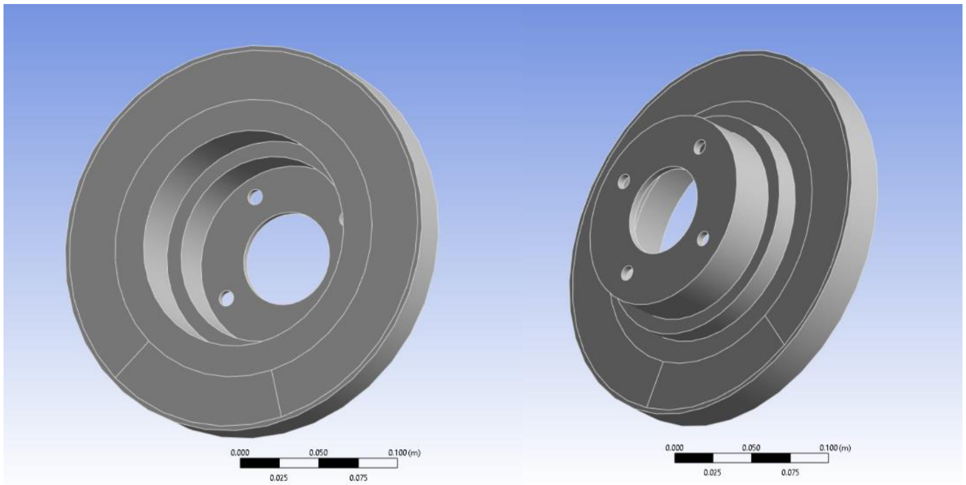

Figure 5: Modal Analysis Geometry

Parameter Values

As described in the given tutorial the following parameter values are selected:

- Max Modes to Find → 10
- Output Mode Number → 8

Results

In order to avoid the condition of resonance, where the natural frequency of the brake disc is equal to the frequency of the engine, we need to determine the frequency modes of the brake disc. To accomplish this we conducted the Modal Analysis. The following is the deformation produced under Modal stress:

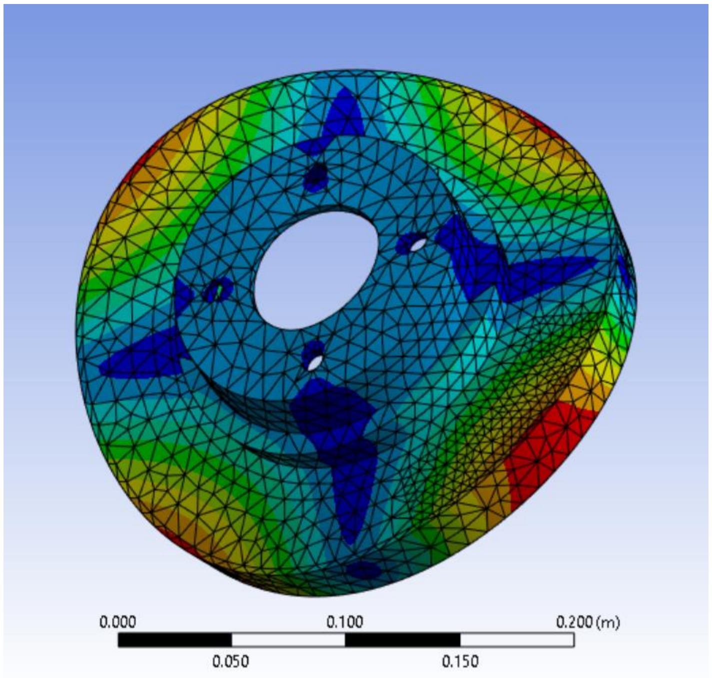

Figure 6: Modal Analysis Result

As seen in the above figure, the lowest non-rigid body frequency of the brake disc is 1592.3 [Hz], before optimization.

Transient Thermal Analysis

Parameter Values

As described in the given tutorial the following parameter values are selected:

- Initial Temperature → 35 °C
- Film Coefficient → 5 W/m^2°C
- Heat Flux → 1539500 W/m^2 (on both sides)
- Output → Temperature

Results

Other than modal load and stress acting on the given model, a tremendous amount of heat is also generated because of the friction between the brake disc and the brake pads. Thus, transient thermal response is important because the brakes are under a huge thermal load. This can then lead to problems like melting of brakes, brake fade or decrease in usable life of the brakes.

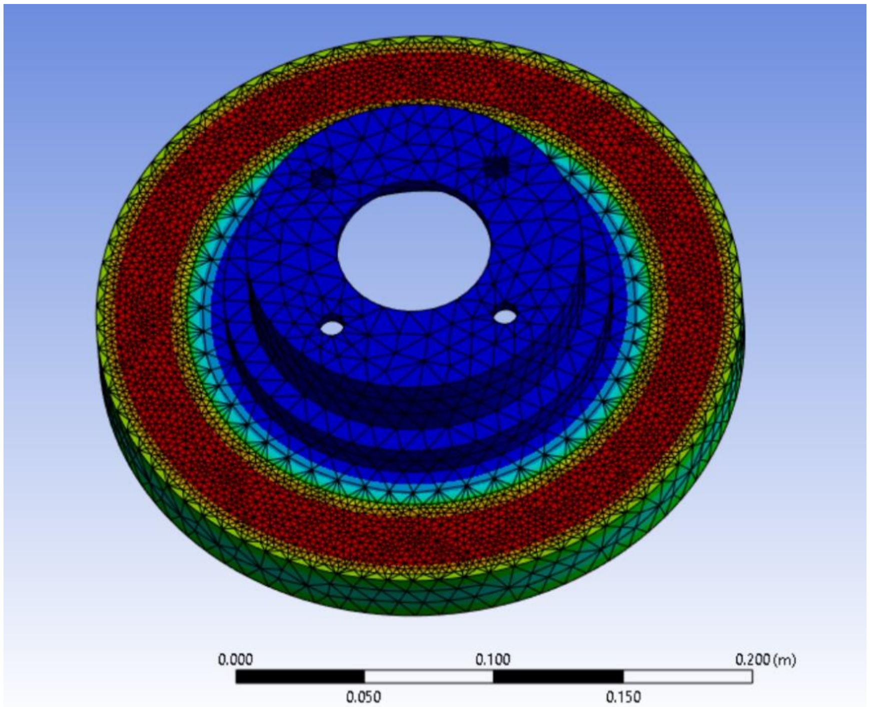

Figure 7: Transient Thermal Analysis Result

As seen in the above figure, the maximum temperature reached by the brake disc is 334.48 °C, before optimization.

Design of Experiments

For this project I have used the Latin Hypercube Sampling (LHS), to sample the data points and to create the appropriate response surface. The LHS has all the sample points with varying nature. Whereas, for sampling methods like Central Composite Design (CCD) and the full factorial method several runs are required. On the other hand, Latin Hypercube Sampling requires the same number of runs, even with an increasing number of parameters. The number of sample points is chosen to be 150, after trying with 50 and 100 points. The reasoning behind the requirement for so many sample points is that the mesh is a relatively refined mesh, and so whenever the geometry is altered, the mesh changes are drastic which could lead to some unwanted spikes which were causing diversion.

<u>Parameter Values</u>

As described in the given tutorial and the model geometry, the following parameter values are selected

- Disc Outer Diameter: Minimum Value → 122 mm and Maximum Value → 132 mm
- Disc Inner Diameter: Minimum Value → 67.5 mm and Maximum Value → 87.5 mm
- Disc Thickness: Minimum Value → 10 mm and Maximum Value → 30 mm

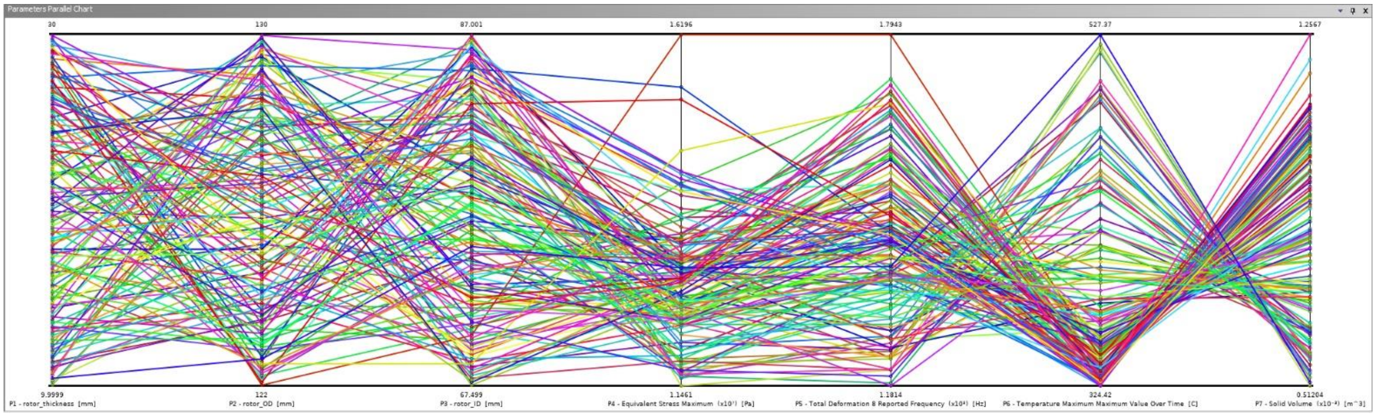

Figure 8: Parameters Parallel Chart

Response Surface

The standard response surface, full 2nd order polynomial meta-model is used to generate the response surface using all the inputs and output parametrized values. For verification 30 points are used, since the number of sample points is 150, thus taking number of verification points = 150/5 = 30 points. The below figures represent the created response surface outputs, which is the interpreted between the output variable thickens and the outer diameter of the disc.

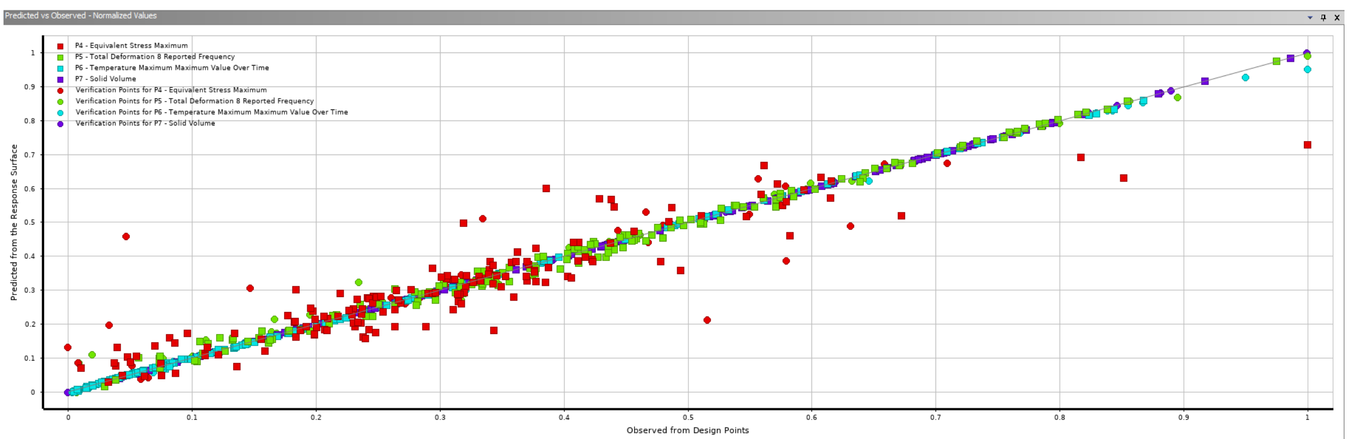

Figure 9: Goodness of Fit

The goodness of fit is calculated for the design of experiment data points and verification points as shown in the figure above. It can be observed that all the points lie near the fitted curve.

Sensitivity Analysis

The thickness is positively correlated with frequency, stress, and volume, whereas it is negatively correlated with temperature, as shown in the figure below. On the other hand, is positively correlated with volume, while it is negatively correlated with all the other output variables. It can also be inferred that the outer diameter is highly sensitive to temperature while the inner diameter has a negligible effect on temperature. Moreover, the increase in inner radius leads to a decrease in natural frequency. Thus, by the local sensitivity analysis, it can be concluded that the change in thickness and outer diameter will provide a solution for the objective function, but it may not be optimal. Therefore, to find the optimal solution, optimization algorithms are required to be executed.

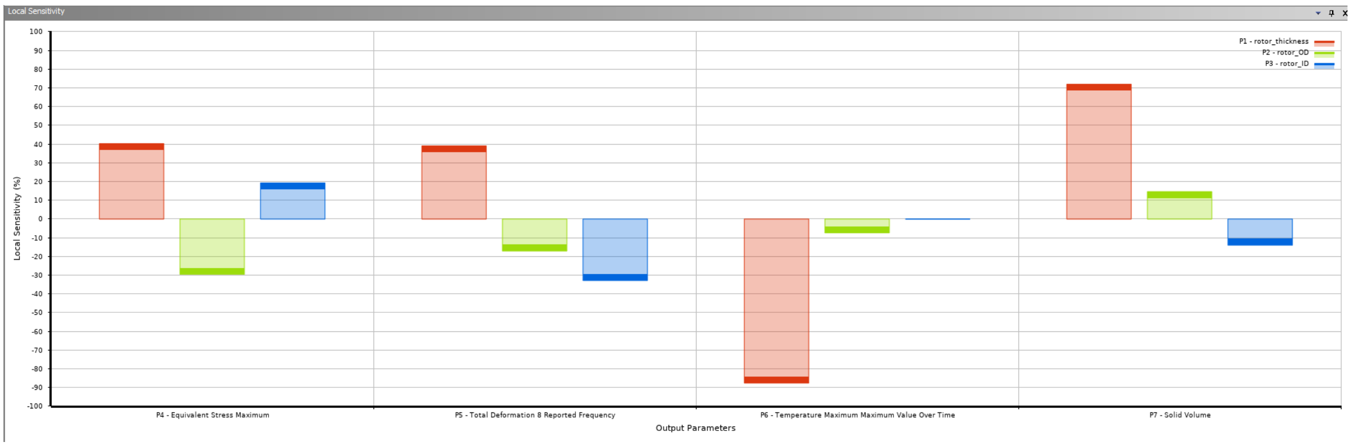

Figure 10: Sensitivity Analysis Result

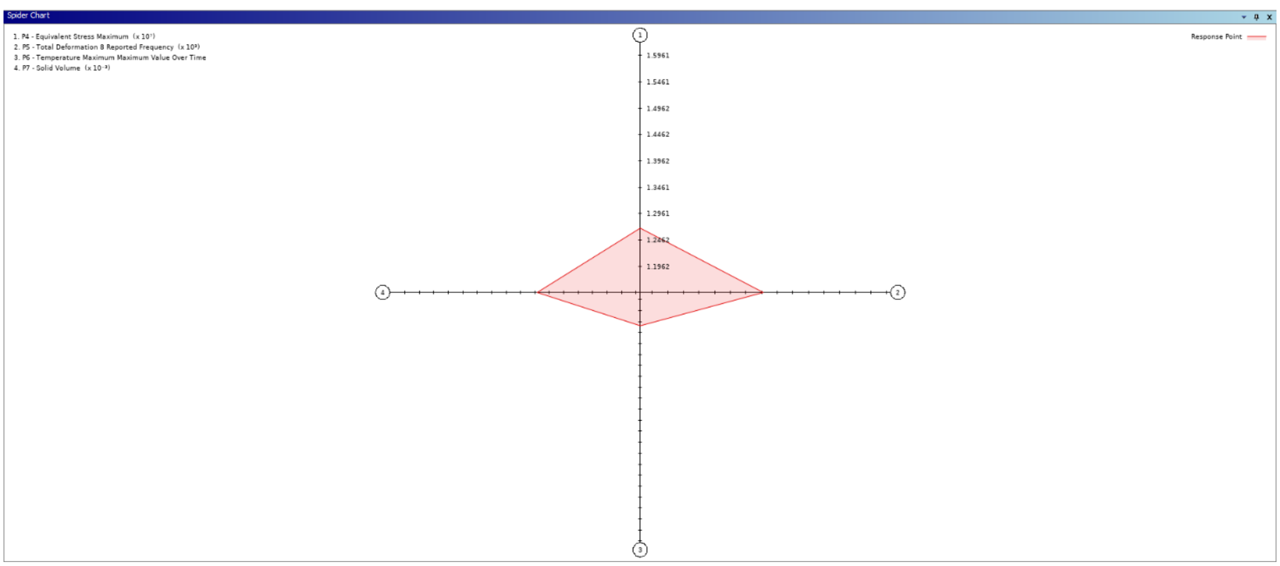

Figure 11: Spyder of Parameters on Response Point

Optimization

As described in the given tutorial, the optimization method used for the analysis was Multi Objective Genetic Algorithm (MOGA). This method was selected because the given problem is composed of multiple objectives, namely, minimize the volume, maximum temperature, and stress, while maximize the lowest non-rigid mode frequency.

Parameter Values

As described in the given tutorial and the model geometry, the following parameter values are selected

Properties of Outline A2: Optimization

	A	B
1	Property	Value
2	Design Points	
3	Preserve Design Points After DX Run	☐
4	Failed Design Points Management	
5	Number of Retries	0
6	Optimization	
7	Method Selection	Manual
8	Method Name	MOGA
9	Estimated Number of Evaluations	93200
10	Tolerance Settings	☑
11	Verify Candidate Points	☑
12	Number of Initial Samples	5000
13	Number of Samples Per Iteration	1800
14	Maximum Allowable Pareto Percentage	70
15	Convergence Stability Percentage	2
16	Maximum Number of Iterations	50
17	Maximum Number of Candidates	3
18	Optimization Status	
19	Converged	Yes
20	Pareto Percentage	0.055556
21	Stability Percentage	1.5033
22	Number of Iterations	7
23	Number of Evaluations	14366
24	Number of Failures	0
25	Size of Generated Sample Set	1800
26	Number of Candidates	3

Figure 12: Optimization Setup

Table of Schematic D4: Optimization

	A	B	C	D	E	F	G	H	I
1	Name	Parameter	Objective			Constraint			
2			Type	Target	Tolerance	Type	Lower Bound	Upper Bound	Tolerance
3	Minimize P7; P7 >= 0 m^3	P7 - Solid Volume	Minimize	0		Values >= Lower Bound	0		0.001
4	Minimize P6; P6 <= 500 C	P6 - Temperature Maximum Maximum Value Over Time	Minimize	300		Values <= Upper Bound		500	0.001
5	Minimize P4; P4 <= 1.4E+07 Pa	P4 - Equivalent Stress Maximum	Minimize	1E+07		Values <= Upper Bound		1.4E+07	0.001
6	Maximize P5; P5 >= 1100 Hz	P5 - Total Deformation 8 Reported Frequency	Maximize	1300		Values >= Lower Bound	1100		0.001

Figure 13: Objectives and Constraints

Results

The results converged in 7 iterations as shown below:

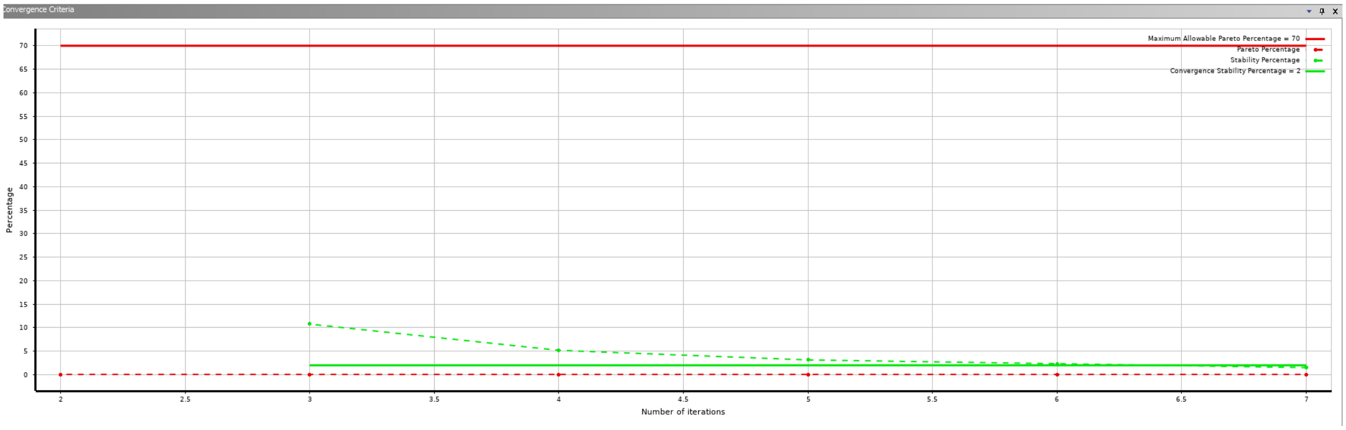

Figure 14: Convergence Analysis

As shown in the figures below, the optimized volume is 0.000798 m^3 which previously was 0.00099667 m^3, thus producing a decrement of 19.93 %.

Candidate Points	Candidate Point 1	Candidate Point 1 (verified)	Candidate Point 2	Candidate Point 2 (verified)	Candidate Point 3	Candidate Point 3 (verified)
P1 - rotor_thickness (mm)	19.063		19.035		19.055	
P2 - rotor_OD (mm)	125.02		125.12		124.99	
P3 - rotor_ID (mm)	76.418		76.012		77.106	
P4 - Equivalent Stress Maximum (Pa)	★ 1.2571E+07	★ 1.2781E+07	★ 1.2557E+07	★ 1.2803E+07	★ 1.2591E+07	★ 1.265E+07
P5 - Total Deformation 8 Reported Frequency (Hz)	★★★ 1468	★★★ 1474.9	★★★ 1472.2	★★★ 1479.7	★★★ 1457.1	★★★ 1462.9
P6 - Temperature Maximum Maximum Value Over Time (C)	★★ 360.88	★★ 361.9	★★ 360.92	★★ 361.95	★★ 361	★★ 362.03
P7 - Solid Volume (m^3)	★ 0.00080261	★ 0.00080256	★ 0.00080542	★ 0.00080539	★ 0.000798	★ 0.00079793

Figure 15: Optimization results 1

Name	P1 - rotor_thickness (mm)	P2 - rotor_OD (mm)	P3 - rotor_ID (mm)	P4 - Equivalent Stress Maximum (Pa)		P5 - Total Deformation 8 Reported Frequency (Hz)		P6 - Temperature Maximum Maximum Value Over Time (C)		P7 - Solid Volume (m^3)	
				Parameter Value	Variation from Reference	Parameter Value	Variation from Reference	Parameter Value	Variation from Reference	Parameter Value	Variation from Reference
Candidate Point 1	19.063	125.02	76.418	★ 1.2571E+07	-0.16%	★★★ 1468	0.75%	★★ 360.88	-0.03%	★ 0.00080261	0.58%
Candidate Point 1 (verified)				★ 1.2781E+07	1.51%	★★★ 1474.9	1.22%	★★ 361.9	0.25%	★ 0.00080256	0.57%
Candidate Point 2	19.035	125.12	76.012	★ 1.2557E+07	-0.27%	★★★ 1472.2	1.03%	★★ 360.92	-0.02%	★ 0.00080542	0.93%
Candidate Point 2 (verified)				★ 1.2803E+07	1.69%	★★★ 1479.7	1.55%	★★ 361.95	0.27%	★ 0.00080539	0.93%
Candidate Point 3	19.055	124.99	77.106	★ 1.2591E+07	0.00%	★★★ 1457.1	0.00%	★★ 361	0.00%	★ 0.000798	0.00%
Candidate Point 3 (verified)				★ 1.265E+07	0.47%	★★★ 1462.9	0.40%	★★ 362.03	0.29%	★ 0.00079793	-0.01%

Figure 16: Optimization results 2

Table 1: Comparison of Parameters before and after Optimization

Parameters	Before Optimization	After Optimization
Max. Temperature [°C]	334.48	361
Lowest Mode Frequency [Hz]	1592.3	1457.1
Equivalent Stress [MPa]	12.845	12.591

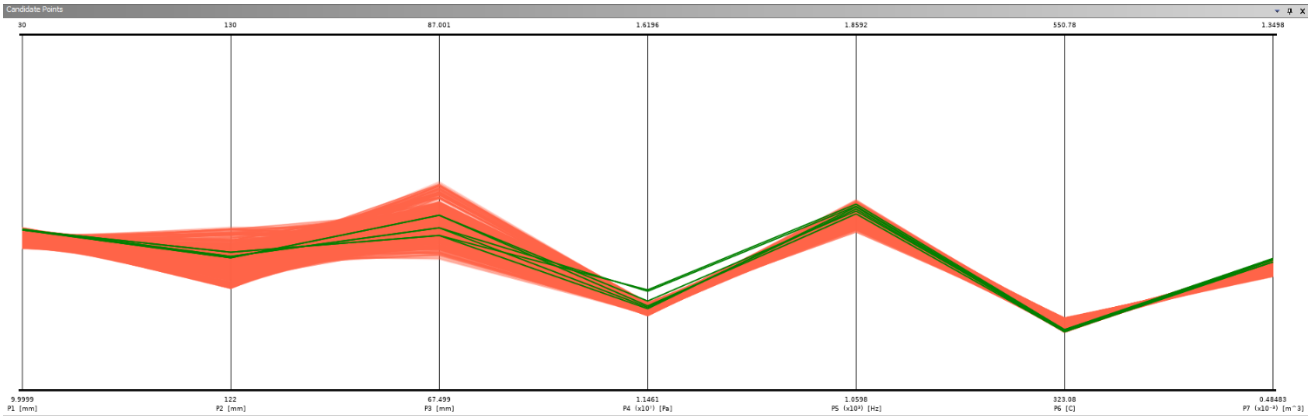

Figure 19: Candidate Points

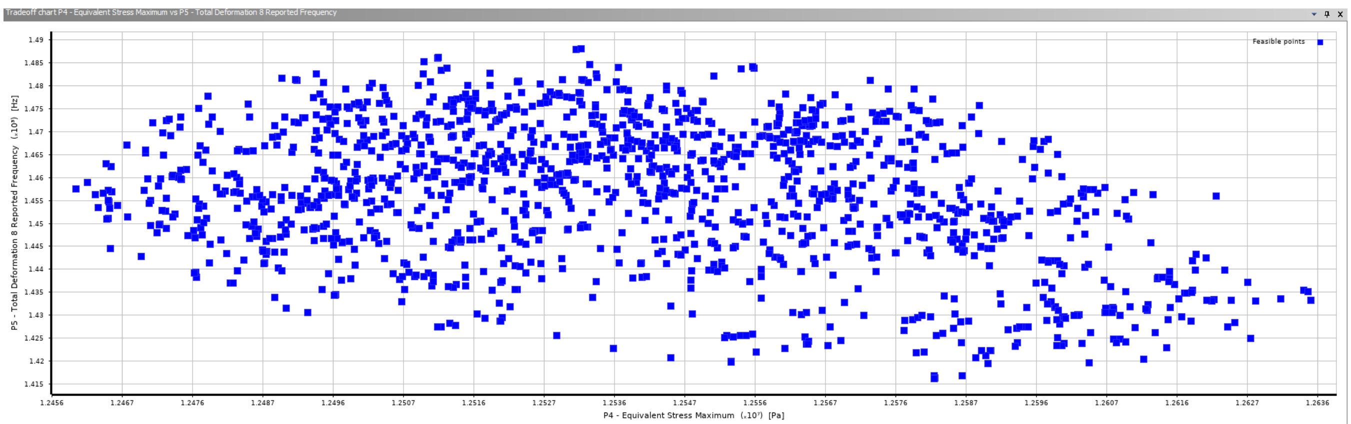

Figure 18: Tradeoff

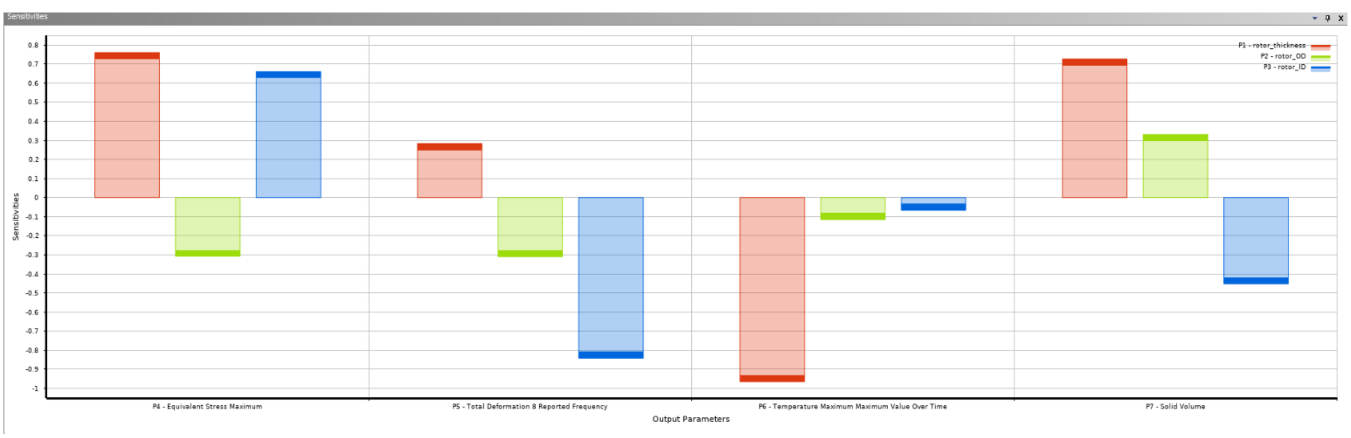

Figure 17: Sensitivities

Thus, the best possible design was obtained based on the given the constraints.

Discussion and Conclusion

- **What are your design variables, constraints, and objectives?**
  - Variables: Volume, equivalent maximum stress, maximum temperature, and first natural frequency of the disc
  - Constraints: Volume ≥ 0, Maximum Stress ≤ 14 [MPa], Maximum Temperature ≤ 500 °C and First Natural Frequency ≥ 1300 [Hz].
  - Objectives: Minimize Volume, Maximum Temperature and Equivalent Maximum Stress, while maximize first natural frequency of the disc.
- **What are the potential trade-offs between your objectives?**
  - As observed from the optimization results, the maximum natural frequency is not completely maximized in order to obtain minimum volume of the disc. However, the constraints are still satisfied, and we obtain perfectly fit results. Hence there are tradeoffs between the total volume, maximum stress, maximum temperature, and first natural frequency. Larger the surface area between the brake pads and the brake disk, lower the stress and temperature in the disk but that would increase volume and affect cost and the first natural frequency of the of the brake disk.
- **Are your variables continuous? Or are they discrete/integer?**
  - All my output and input variables are continuous.
- **Do you have analytical objective/constraint functions? And are they differentiable?**
  - All the objectives are differentiable.
- **Based on the above answers, what optimization methods will you choose?**
  - I chose Multi Objective Genetic Algorithm (MOGA). This method was selected because the given problem is composed of multiple objectives.
- **Perform a sensitivity analysis and comment on the importance of your variables? Also, do you observe monotonicity (i.e., the objective always goes up or down with a variable)?**
  - The sensitivity analysis is conducted in the section under Response Surface.
  - There is monotonicity with each objective.
- **Compare your optimal design against the initial one (e.g., see the following comparison on the brake disc design) AND comment on whether the optimal design is reasonable.**
  - Before Optimization
    - The below figures represent the geometry of the brake model based on the given data, before conducting the optimization. The Volume of the model is 0.00099667 m^3.

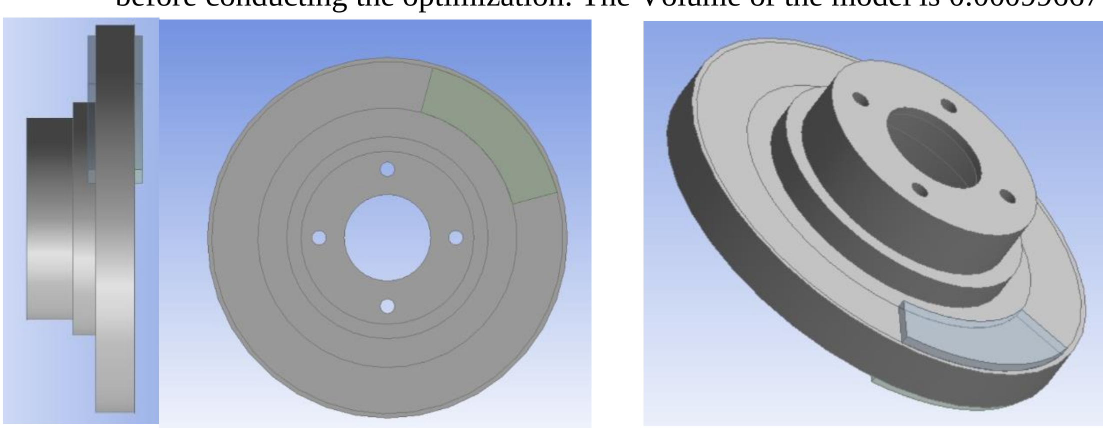

Figure 20: Geometry Before Optimization

- After Optimization geometry:
    - The below figures represent the geometry of the brake model based on the optimized output parameters. The Volume of the brake model is 0.000798 m^3 which previously was 0.00099667 m^3, thus producing a decrement of 19.93 %

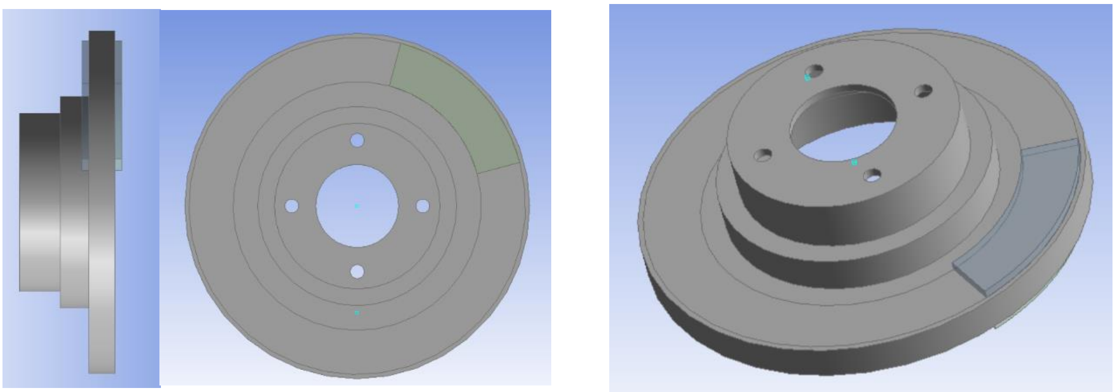

Figure 21: Geometry After Optimization

- Static Structural Analysis after Optimization
    - The below figure represents the solution of Static Structural Analysis based on the optimized parameter values. The Maximum Equivalent von-Mises Stress on the brake model is reduced to 12.603 [MPa] from 12.845 [MPa].

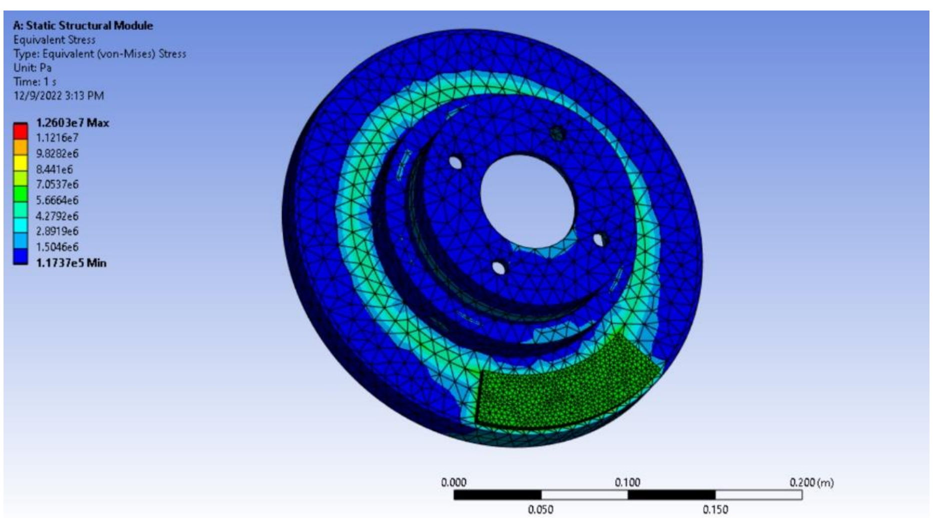

Figure 22: Static Structural Analysis after Optimization

- Modal Analysis after Optimization
    - The below figure represents the solution of Modal Analysis based on the optimized parameter values. The Free Natural Frequency of the brake model is reduced to 1463.4 [Hz] from 1592.3 [Hz].

- Although we wish maximize the natural frequency, the optimized output natural frequency is lower than the original because of some tradeoff. However, the optimized output natural frequency still satisfies the given constraints and we have a considerable lower optimized output volume.

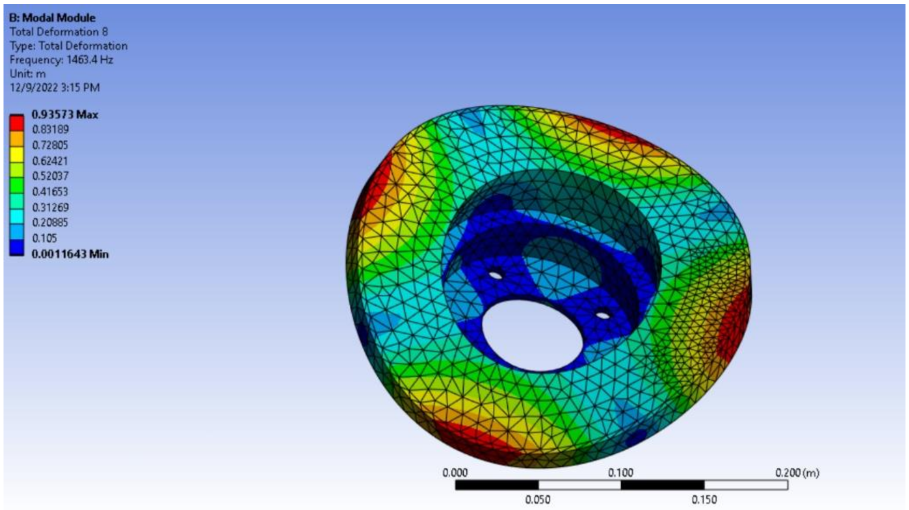

Figure 23: Modal Analysis after Optimization

- Transient Thermal Analysis after Optimization
  - The below figure represents the solution of Transient Thermal Analysis based on the optimized parameter values. The Maximum Temperature of the brake model is increased to 362.02 from 334.48 °C.

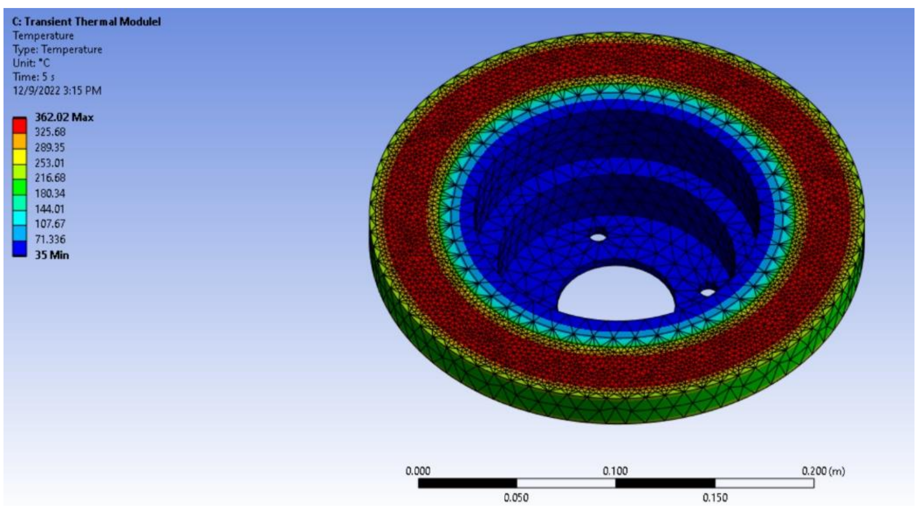

Figure 24: Transient Thermal Analysis after Optimization

Since all the output parameters satisfy the given constraints and the main objective of reduction in volume has been satisfied, we can conclude that the optimal design is reasonable. The ANSYS file is at : https://github.com/monarkparekh/MAE-598__Design-Optimization/tree/Project-2