MAE 598: Design Optimization

Project 2. Design Optimization of a Brake Disk

By

Rohit Iyer

Problem definition

The objective of this project is to optimize the design of a gray cast Iron brake disk with brake pads made of structural steel while maintaining it's structural, modal, and thermal integrity. The design variables of this brake disk are three geometric dimension parameters, i.e., inner radius, outer radius and thickness. The design is tested to ensure safety under the following cases: maximum stress under static loading, frequency of free vibration and maximum temperature. The design constrains mainly come from the geometry, since the dimensions cannot be either too large or too small. The objective of the design is to have less strain value, higher vibration frequency, and low temperature.

The software ANSYS R21 is used to perform the analysis and optimization.

Ansys

Ansys offers structural model, transit thermal and various other analysis. Basically, it is a software which provide 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 we have used ANSYS software to optimize an already existing brake pad design to simulate how a brake pad will hold up after years of its use.

Model setup

A brake disk part model is obtained from A. Durgude, A. Vipradas, S. Kishore, and S. Nimse provided by Dr. Yi Ren for the course of MAE 598: Design Optimization for the purpose of Project 2: Brake disk optimization.

The brake file (.agdb) is then used as the Input to ANSYS workbench which is later used by different analysis modules. Three modules are used for different analysis purposes are:

1. Static mechanical analysis
2. modal analysis
3. Transient thermal analysis.

The topology of the model setup is shown in Figure 1. The geometry of the part is generated from the input parameters. The part geometry and materials properties are shared by the three modules. Structural steel is assigned to the pads and grey cast Iron is assigned to the main body.

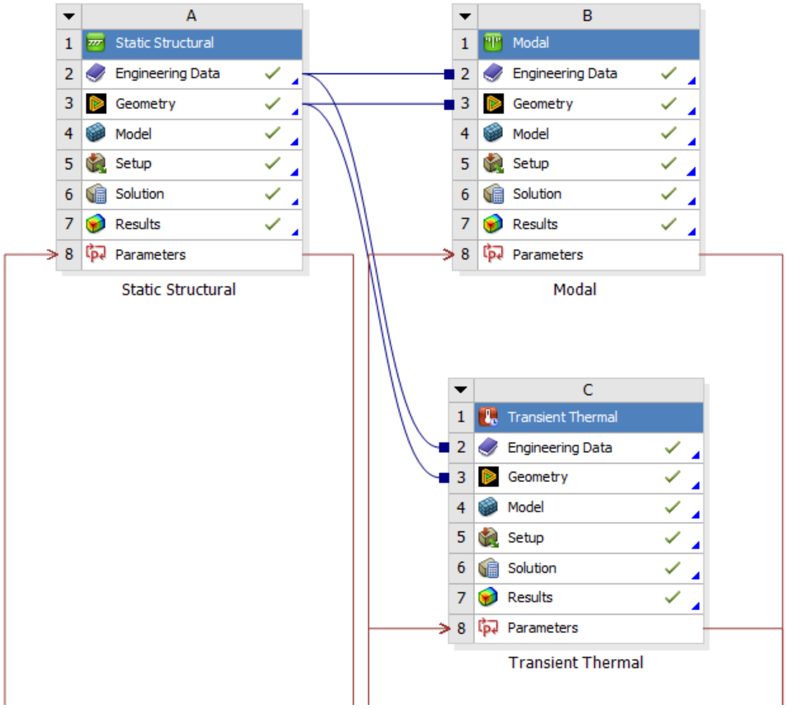


Figure 1: The setup of the design optimization process

Initial setup

The geometry of the disk brake is imported into the static structural analysis. The body has predefined dimensions which can be changed according to the need. The analysis is carried out in static structural, modal and transient thermal.

Static mechanical analysis

The static structural module is used for the analysis.

The mesh is carried out by the method 'tetrahedron' by using the Patch conforming algorithm. The inner face of the brake pad is meshed with 3mm element size.

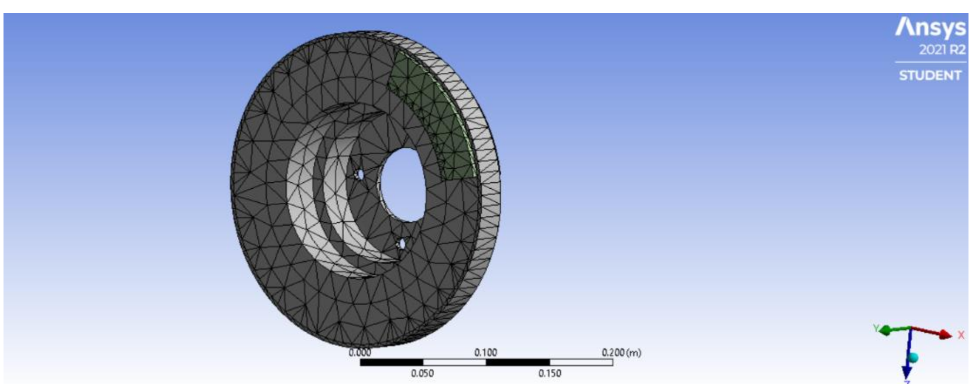


Figure 2: Mesh of the brake disk assembly model

The main body of the brake disk is assigned to rotate along the y axis with a speed of 250 rad/s. Frictional contact is applied between the main body and the two pads. The pads are constrained in the x and z direction, and pressures of 10.5 MPa are applied on the outer faces of the pads. The setup of the mechanical model is shown in Figure 2. The maximum Von Mises stress and the volume of the main body are calculated and used as the output values.

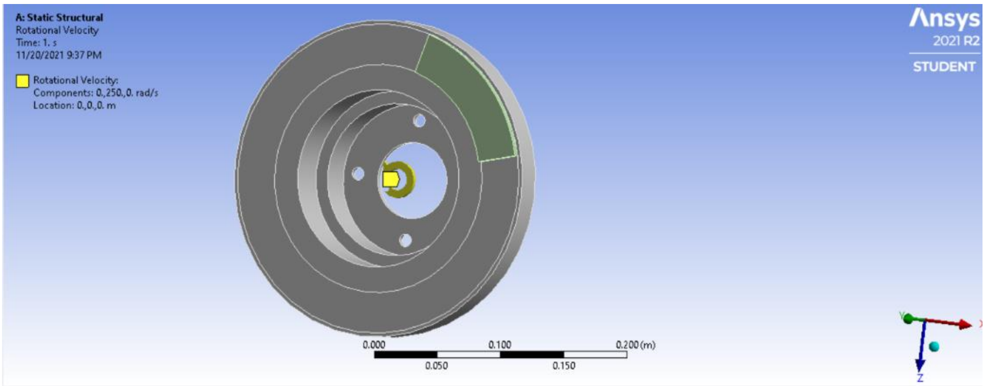


Figure 3: Setup of the brake disk with rotational velocity

Modal analysis

The modal module is used for this analysis. The same geometry and materials assignment is adopted from the static structural analysis but the two brake pads are suppressed. No boundary conditions are needed applied to the part since we are solving the free vibration response. The maximum number of modes to find is set to 10. The 7th vibration mode is selected for deformation calculation since the first six modes are rigid-body vibration modes. The frequency of the mode is used as the output for modal analysis.

Thermal analysis

The transient thermal module is used for this analysis. The same geometry and materials assignment is adopted from the static structural analysis, while the two brake pads are removed from the simulation. Convection boundary conditions are applied to all the faces of the geometry, with factor set as 5 W/m2. Heat flux of 1500 kW/m2 is applied to the surface of main body contacting with the pads. The maximum temperature is extracted for output value.

Initial design performance

The input parameters of the initial design are set as:

1. Inner diameter = 75 mm
2. outer diameter = 125 mm
3. Thickness = 25mm

The volume of the initial design is After performing static structural analysis, the Von Mises stress distribution is shown in Figure 4. The maximum Von Mises stress of the initial design is 14.2 MPa.

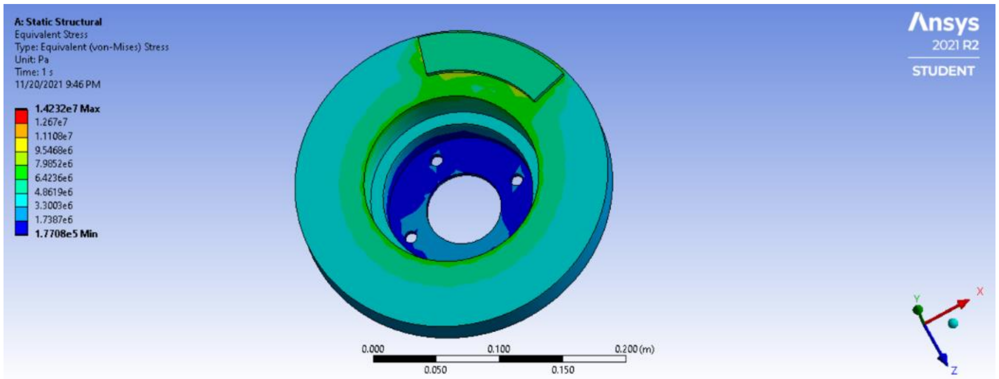


Figure 4: Distribution of the Von Mises stress

After performing vibration modal analysis, the deformation mode of the 7th mode of the initial design is shown in Figure 4. The corresponded frequency is 2079.8 Hz.

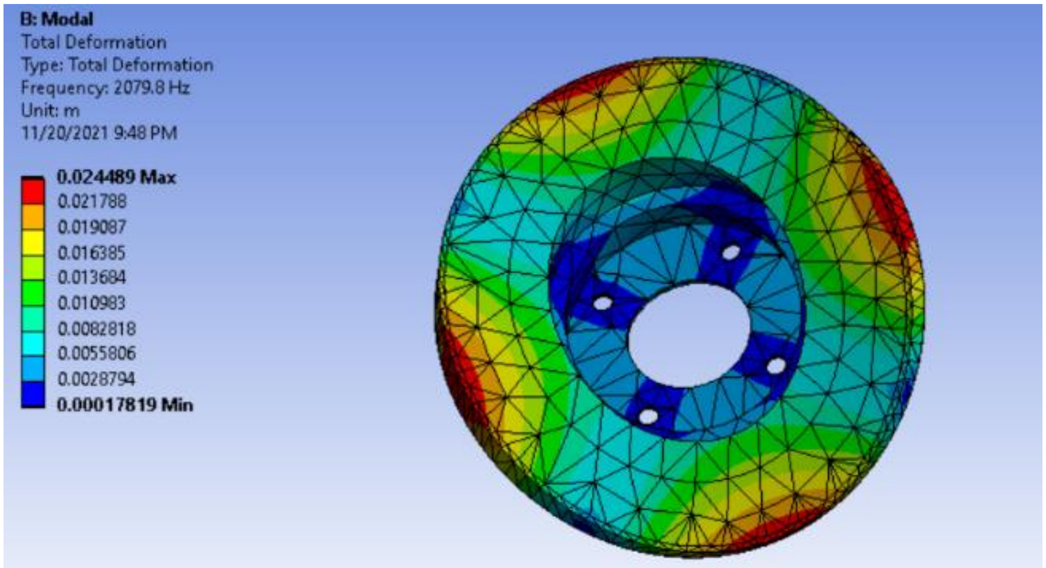


Figure 5: Deformation of the 7th vibration mode

After performing transient thermal analysis, the temperature distribution of the initial design is shown in

Figure 6. The maximum temperature is 311.6 ºC, and it is located at the contacting surface with the pads.

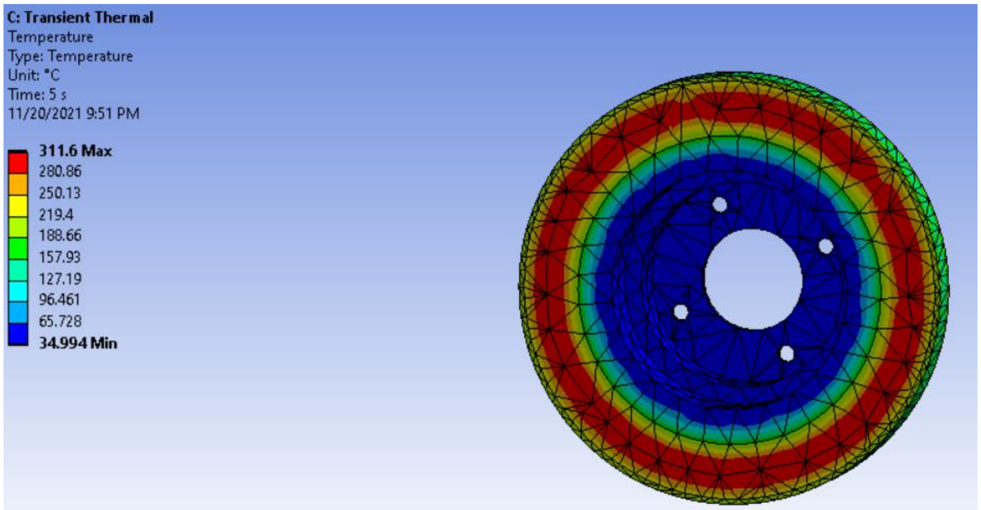
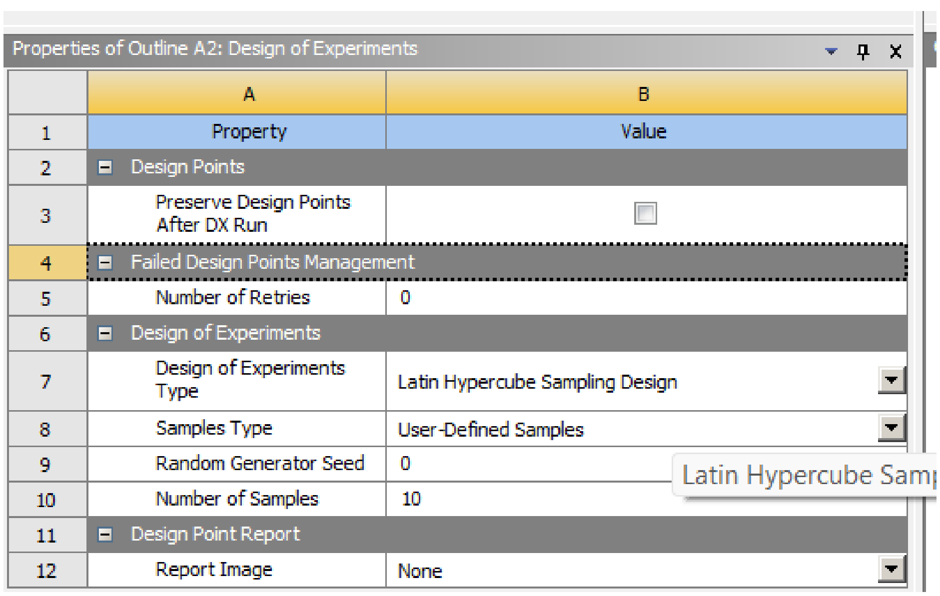


Figure 6: Temperature distribution of the Initial design

Response surface

Since the design variables are all dimensional parameters, they are all continuous.

For the sake of computing time, 10 design points are generated using the Latin Hypercube Sampling method. Neural networks are used to fit the relationship between the input parameters and the responses. Five verification points are used to estimate the accuracy of the response surface.


Properties of Outline A2: Design of Experiments

	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	Design of Experiments	
7	Design of Experiments Type	Latin Hypercube Sampling Design
8	Samples Type	User-Defined Samples
9	Random Generator Seed	0
10	Number of Samples	10
11	Design Point Report	
12	Report Image	None

Latin Hypercube Samp

Figure 7: Design of Experiments outline

Table of Outline A2: Design Points of Design of Experiments

	A	B	C	D	E	F	G
1	Name	P1 - rotor_thickness (mm)	P2 - rotor_OD (mm)	P3 - rotor_ID (mm)	P4 - Total Deformation Reported Frequency (Hz)	P5 - Temperature Maximum (C)	P6 - Equivalent Stress Maximum (MPa)
2	1	24.7	128.58	72	2058.9	312.43	15.153
3	2	26.5	123.18	86.4	1794.6	309.52	21.228
4	3	22.3	133.98	84	1686.8	320.22	15.904
5	4	25.9	131.28	69.6	2026.3	311.71	15.43
6	5	22.9	129.93	74.4	1955.2	317.17	15.291
7	6	27.7	132.63	76.8	1936.9	309.14	15.7
8	7	25.3	135.33	81.6	1770.1	311.46	16.047
9	8	27.1	127.23	79.2	1968.5	306.08	15.135
10	9	24.1	125.88	88.8	1655.2	309.99	14.383
11	10	23.5	124.53	67.2	2142.6	313.93	14.428

Figure 8: Design of Experiments obtained points

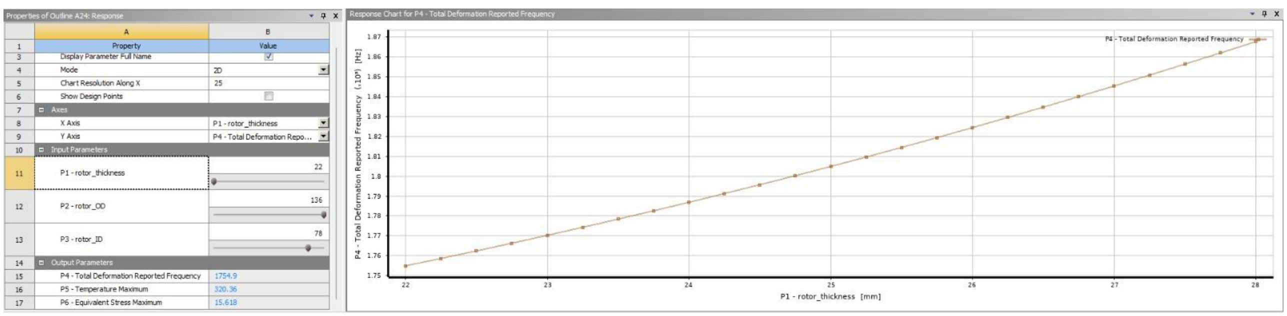


Figure 9: Response surface

Sensitivity

The sensitivity between design variables and responses can be calculated by the ANSYS and it is plotted below. From the sensitivity analysis results, the rotor thickness has the largest positive effect on the maximum stress and increasing the inner and outer diameters will both increase the stress value. Temperature is most sensitive to the thickness of the brake disk, and it can be reduced by increasing the thickness. Reducing thickness and outer diameter will reduce the volume of the disk and increasing the inner diameter will reduce the volume of the disk.

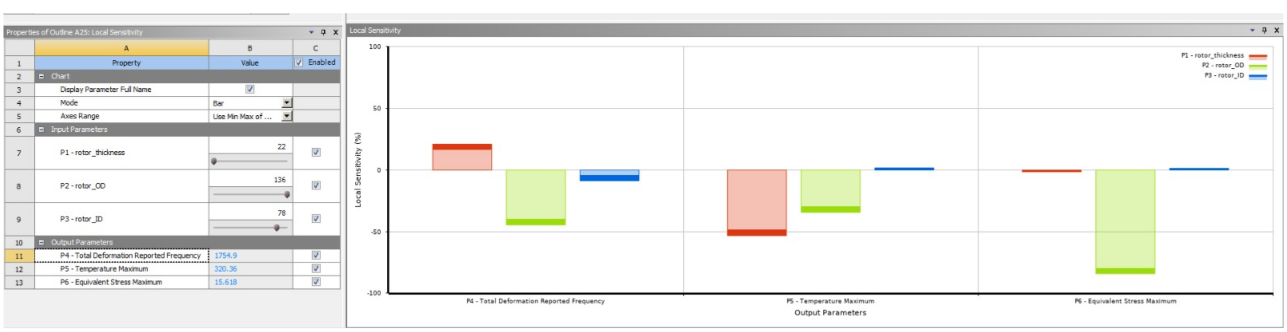


Figure 10: Sensitivity between different design parameters and objectives

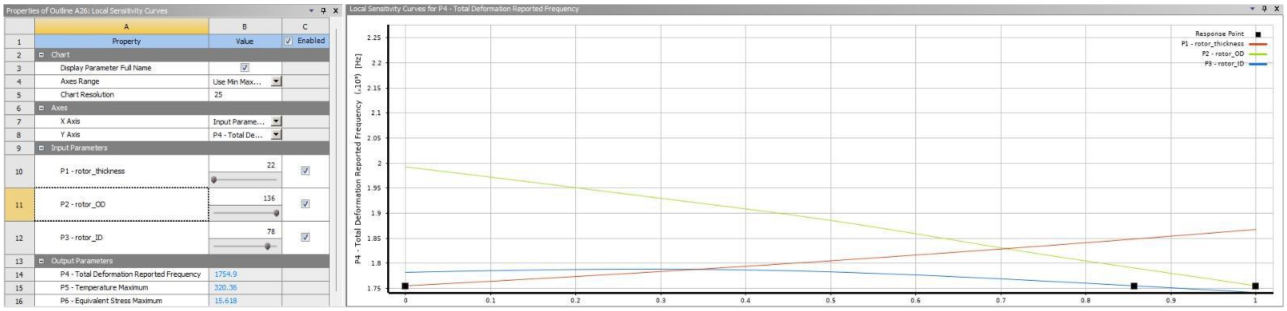


Figure 11: Local sensitivity curves

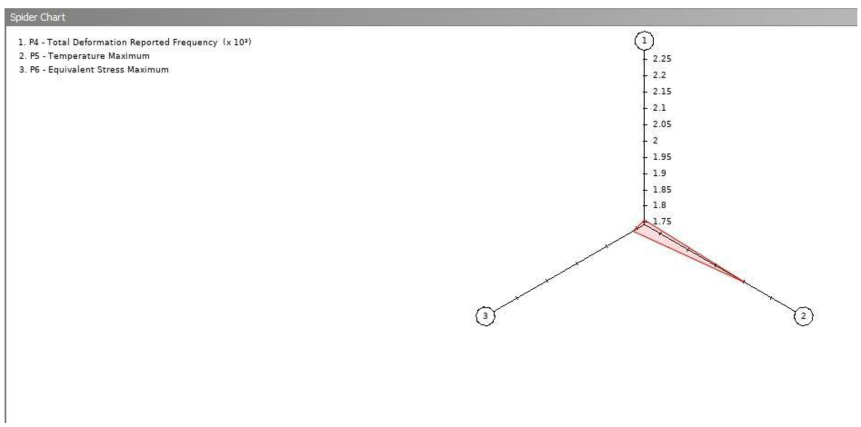


Figure 12: Spyder plot with response point

Optimization and final design performance

As the main objective of this project was to optimize the already existing brake pad and to find the minimum volume, stress, temperature and to maximize the frequency. Compared with the design provided by Professor Yi Ren, a slight reduction in temperature and performance on stress relief are better in this case. Based on engineering parameters and response surfaces, the final design can be explained by this model. If we increase the thickness the temperature will slightly be reduced and a slight change in frequency will be observed. Altering lower and upper bounds will also play a vital role on this optimization project.

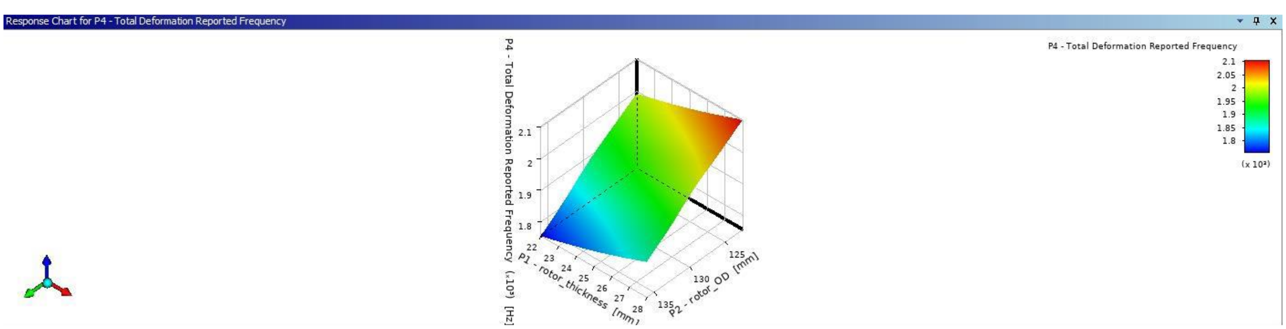


Figure 13: Response for deformation

Because there are multiple objectives in this design, the Multi-objective Genetic Algorithm (MOGA) is used to find the best design candidates. As mentioned earlier, the objective of the optimization is to minimize volume, stress, and temperature, and maximize frequency. Default importance is applied to each response. The best design candidates are shown below. Compared with the initial design, the optimized versions have better performance on stress relief, higher frequency, and better temperature control although, the reduction of volume is not much.

Table of Schematic E4: Optimization

	A	B	C	D
1	Optimization Study			
2	Seek P4 = 0 Hz	Goal, Seek P4 = 0 Hz (Default Importance)		
3	Seek P6 = 0 MPa	Goal, Seek P6 = 0 MPa (Default Importance)		
4	Optimization Method			
5	MOGA	The MOGA method (Multi-Objective Genetic Algorithm) is a variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports multiple objectives and constraints and aims at finding the global optimum.		
6	Configuration	Generate 3000 samples initially, 600 samples per iteration and find 3 candidates in a maximum of 20 iterations.		
7	Status	Converged after 5584 evaluations.		
8	Candidate Points			
9		Candidate Point 1	Candidate Point 2	Candidate Point 3
10	P1 - rotor_thickness (mm)	22.073	22.049	22.187
11	P2 - rotor_OD (mm)	135.58	134.78	133.67
12	P3 - rotor_ID (mm)	78.883	78.353	78.942
13	P4 - Total Deformation Reported Frequency (Hz)	1751.6	1773.5	1786.5
14	P6 - Equivalent Stress Maximum (MPa)	15.548	15.106	14.691

Figure 14: Best design

Thus, we obtained the best possible design given the constraints and hardware restrictions.