DESIGN OPTIMIZATION OF BRAKE DISC GEOMETRY

by
SHUBHVARATA
DUTTA
MAE 598-DESIGN
OPTIMIZATION
Final Report

FALL, 2022

ABSTRACT

The motivation of this project is to determine the optimum dimensions of the brake disc for a four-wheeler vehicle using ANSYS Design Optimization Tool. These dimensions include the disc inner radius, outer radius and thickness. Structural, modal and thermal load cases for emergency braking conditions are individually considered to determine these dimensions. The optimization objective is to minimize the brake disc volume, whereas the other objectives are to minimize the stress, temperature and maximize the first natural frequency of the disc. These goals are accomplished using optimization algorithms in MATLAB and the results are correlated with the values obtained from ANSYS. Finally, system optimization is performed using MOGA by integrating all the load cases.

Contents

1 Design Problem Statement

Primary objective: To minimize the brake disc volume for emergency braking conditions

Secondary objectives:

- Minimize the maximum stress in the brake disc
- Maximize the first natural frequency of the brake disc
- Minimize the maximum temperature in the brake disc

1.1 Subsystem 1: Structural Analysis

The brake disc must sustain the pressure from the hydraulically actuated brake pads during sudden braking conditions. Stresses are induced due to friction between the brake pads and the disc. The disc also experiences centrifugal body forces due to its rotation. Resultant stresses generated due these forces can lead to material failure. Therefore, it is of prime importance to make sure that the stresses in the disc are minimized.

1.2 Subsystem 2: Modal Analysis

Free modal analysis is performed to ensure that the disc's first natural frequency is higher than the engine firing frequency. This guarantees that the disc does not experience failure due to resonance.

1.3 Subsystem 3: Thermal Analysis

Braking in a vehicle takes place due to friction between the brake pads and the rotor disc. This leads to heat flux generation in the disc which consequently results in increase in its temperature and thermal stresses. Emergency braking conditions induce high temperatures that damage the contact surfaces. It is therefore essential to minimize the temperature to prevent disc wear and tear.

The design optimization methodology flowchart is as shown in Figure 1.

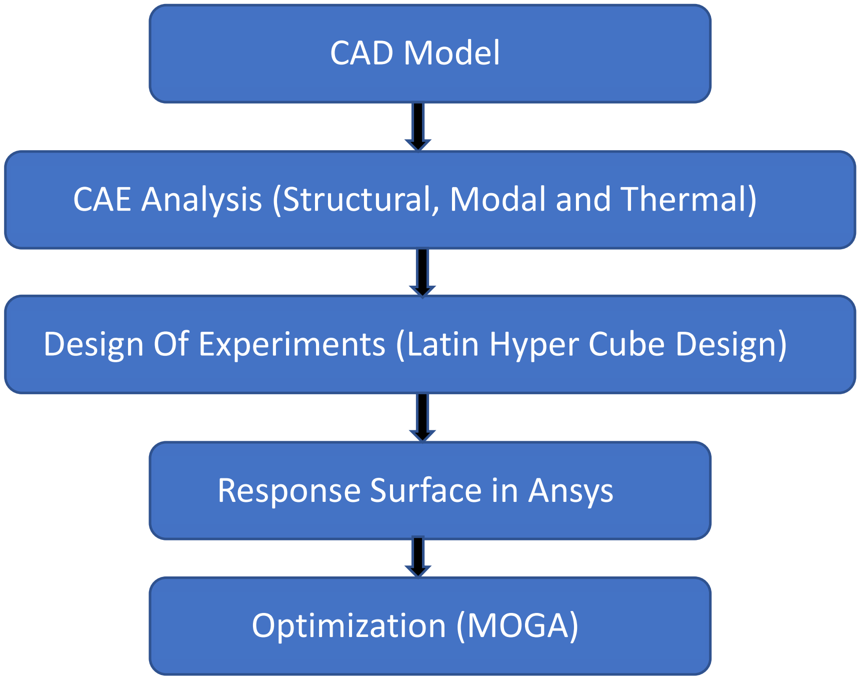

Figure 1: Design optimization process flowchart

2 Nomenclature

V: Volume (mm^3)

α: Regression coefficients for volume

S: Stress (MPa)

β: Regression coefficients for stress

F: First natural frequency (Hz)

γ: Regression coefficients for frequency

T: Temperature (^oC)

δ: Regression coefficients for temperature

$P1$: Inner disc radius (mm)

$P2$: Outer disc radius (mm)

$P3$: Disc thickness (mm)

d: Braking distance (m)

t_e: Braking time (s)

u: Initial vehicle velocity ($kmph$)

m: Vehicle mass (kg)

ν: Coefficient of friction

w: Disc rotational velocity (rad/s)

R_{avg}: Average of P1 and P2 (mm)

KE: Vehicle kinetic energy (J)

F_b: Total braking force on the disc (N)

P: Pressure acting on the disc due to one brake pad (Pa)

A_p: Brake pad area (mm^2)

A_{sp}: Brake pad swept area (mm^2)

q: Heat flux (W/m^2)

T_{amb}: Ambient temperature (^oC)

3 Mathematical Model

The brake disc inner radius (P1), outer radius (P2) and thickness (P3) are the design variables in this optimization study. Initially, structural, modal and thermal analyses are performed in ANSYS. For the assumed geometric constraints, Design Of Experiment (DOE) points are generated. Mathematical model is then generated by performing a 2^{nd} order regression analysis on these DOE points to obtain the volume, stress, frequency and temperature quadratic functions. The optimization problem is designed as follows:

Primary objective:

$$Minimize : f1 : V = \sum_{n=1}^{n=3} \alpha_n x_n + \sum_{n=4}^{n=6} \alpha_n x_n^2 + \alpha_7; (R^2_{volume} = 0.98) \quad (1)$$

Secondary objectives:

$$f2 : Minimize : S = \sum_{n=1}^{n=3} \beta_n x_n + \sum_{n=4}^{n=6} \beta_n x_n^2 + \beta_7; (R^2_{stress} = 0.53)^1 \quad (2)$$

$$f3 : Maximize : F = \sum_{n=1}^{n=3} \gamma_n x_n + \sum_{n=4}^{n=6} \gamma_n x_n^2 + \gamma_7; (R^2_{frequency} = 0.92) \quad (3)$$

$$f4 : Minimize : T = \sum_{n=1}^{n=3} \delta_n x_n + \sum_{n=4}^{n=6} \delta_n x_n^2 + \delta_7; (R^2_{thermal} = 0.97) \quad (4)$$

Geometrical constraints for all subsystems:

$$g1 : -P1 \leq -66 \quad (5)$$

$$g2 : P1 \leq 90 \quad (6)$$

$$g3 : -P2 \leq -124 \quad (7)$$

$$g4 : P2 \leq 150 \quad (8)$$

$$g5 : -P3 \leq -5 \quad (9)$$

$$g6 : P3 \leq 27 \quad (10)$$

This is a multi-objective optimization problem. A multi-objective optimization algorithm can be readily solved using MOGA. But MOGA will not always give the best optimum value. Therefore, it is beneficial to convert the multi-objective problem into a single objective problem by assuming the rest of the objectives as constraints. In order to use single-objective optimization algorithms for this problem, it is thus necessary to provide upper or lower bounds on the secondary objective functions. This aids in plotting the desired Pareto curves. The bounds are determined by taking into consideration the brake disc failure.

Design constraints:

$$g7 : S \leq 14MPa \quad (11)$$

$$g8 : -F \leq -1200Hz \quad (12)$$

$$g9 : T \leq 400^oC \quad (13)$$

The sub-system level optimization problems are designed as follows:

3.1 Structural Analysis Model

$Min : f1, f2$

s.t.
$g1, g2, g3, g4, g5, g6$ and $g7$

[1] 5^{th} order polynomial fit gives a better R^2 value of 0.77. Kriging model may work better as well.

3.2 Modal Analysis Model

$Min : f1, -f3$

s.t.
$g1, g2, g3, g4, g5, g6$ and $g8$

3.3 Thermal Analysis Model

$Min : f1, f4$

s.t.
$g1, g2, g3, g4, g5, g6$ and $g9$

4 Model Analysis

The flowchart for optimization in ANSYS is shown in Figure 2. This figure pertains to the flowchart for structural analysis. Similar procedure is followed for modal and thermal analyses.

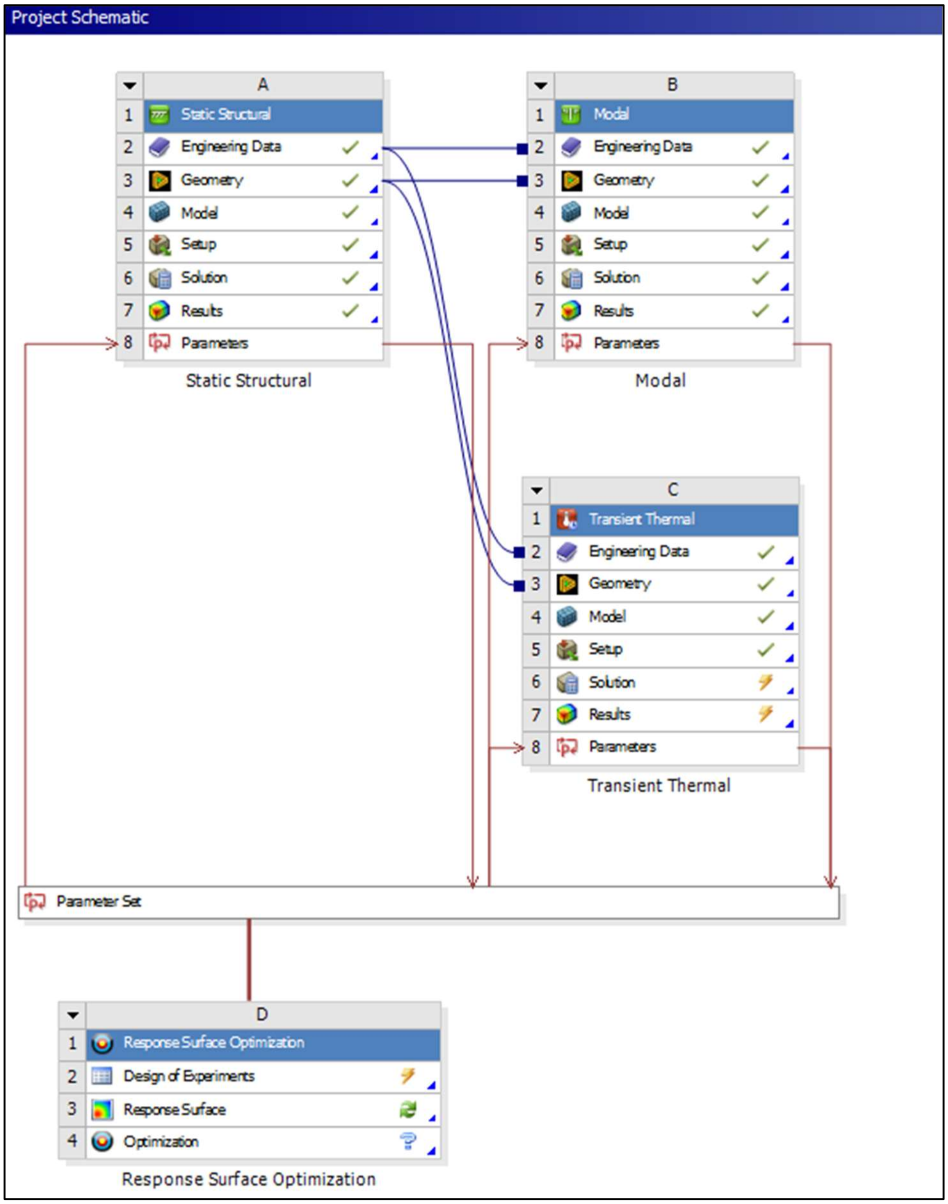

Figure 2: ANSYS Optimization Flowchart

4.1 FE Model

The brake disc geometry is prepared in ANSYS Design Modeler as shown in Figure 3. The initial values for P1, P2 and P3 are considered to be 75 mm, 125 mm and 25 mm respectively. These values are obtained from the literature survey by considering disc dimensions in different vehicles.

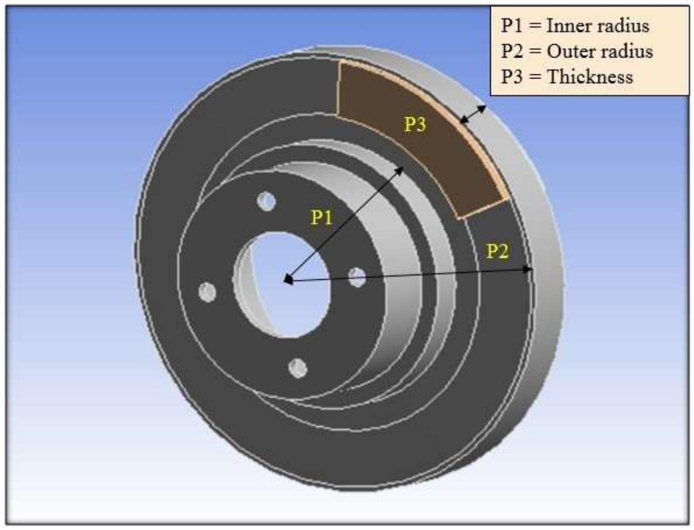

Figure 3: Disc Brake CAD Model

The CAD geometry is meshed using 6 mm sized tetrahedral quadratic elements as shown in Figure 4.

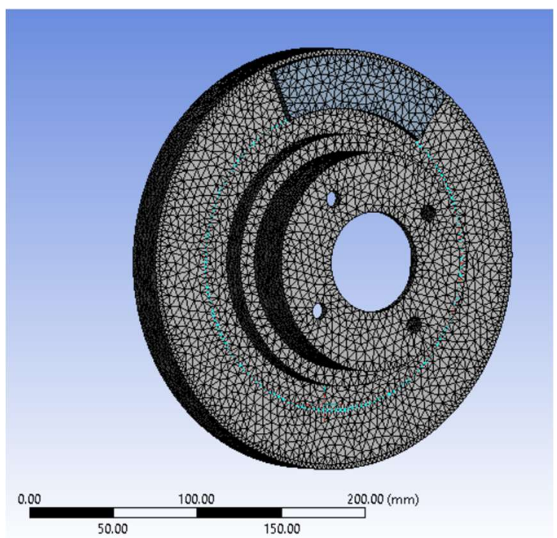

Figure 4: Disc Brake Mesh Model

The brake disc is made of gray cast iron. The material properties of gray cast iron are shown in Table 1.

Property	**Value**	**Unit**
Density	7200	kg/m^3
Young's modulus	110	GPa
Poisson's ratio	0.28	-

Thermal conductivity	52	W/m^oC
Specific heat	447	J/kg^oC

Table 1: Gray cast iron material properties

The assumptions made for performing the FEM analysis are as follows:

- The braking torque distribution between the front and rear axles is 70:30.
- Natural convection takes place due to the ambient air.
- The disc brake considered is of the solid type.
- Heat flux on the disc brake acts on both sides of the disc.

4.2 Structural Analysis

Considering the emergency braking distance of 10 m with stopping time of 5 seconds for a vehicle with initial velocity of 90 km/hr, the braking pressure required is calculated. This pressure will act as one of the boundary conditions for the simulation. It is assumed that 70% of the braking power is in the front axle of a four-wheeler vehicle. The force acting on a single disc brake is calculated by multiplying the front axle braking power by 0.5. Using this, the pressure on each pad is calculated by considering the coefficient of friction. Here, $d = 10m; t_e = 5s; u = 90kmph; m = 1500kg; \nu = 0.22; A_p = 3552mm^2$.

Calculating the disc angular velocity (ω),

$$\omega = u/R_{avg} = 250rad/s \tag{14}$$

Evaluating the total braking force on the disc (F_b),

$$F_b = (KE * 0.5 * 0.7)/d = 16.406kN \tag{15}$$

Formulating the pressure exerted by one brake pad on the disc (P),

$$P = F_b/(2 * A_p * \nu) = 10.495MPa \tag{16}$$

Static structural analysis is performed on the aforementioned geometry due to the brake pad actuating load acting on the disc. The boundary conditions imposed on the brake geometry are as follows. See Figure 5.

- The inner circumference of the wheel hub has a revolute joint applied.
- The brake caliper pads are constrained in X and Z directions. They are kept free to move in the Y direction.
- The rotational velocity of 250 rad/s is applied on the disc at the wheel hub attachment point.
- Frictional contact is provided between the brake caliper pads and brake rotor.

- Actuating pressure of 10.496 MPa is applied on the brake pads.

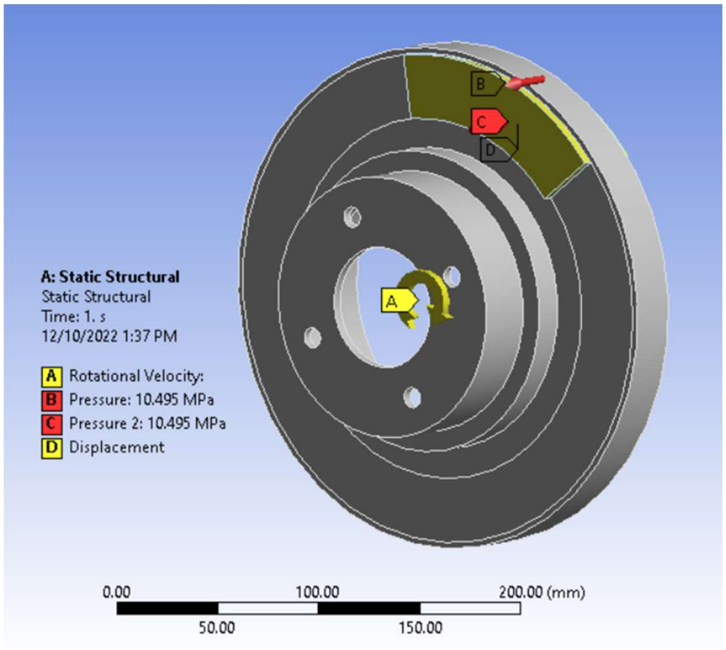

Figure 5: Static structural boundary conditions

The stress plot obtained is as shown in Figure 6. As observed, the maximum stress of 14.262 MPa is obtained for initial conditions on the design variables.

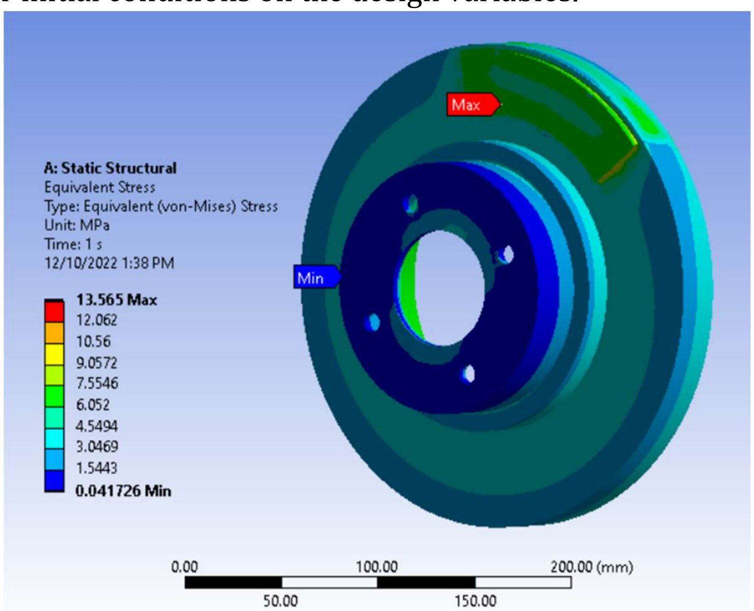

Figure 6: Stress plot for initial conditions on the design variables

4.3 Modal Analysis

Modal analysis is performed on the brake disc to determine its free natural frequency. The brake caliper pad geometry is suppressed while performing this analysis because the natural frequency of the brake disc is to be determined. Figure 7 shows the first natural frequency and its mode shape.

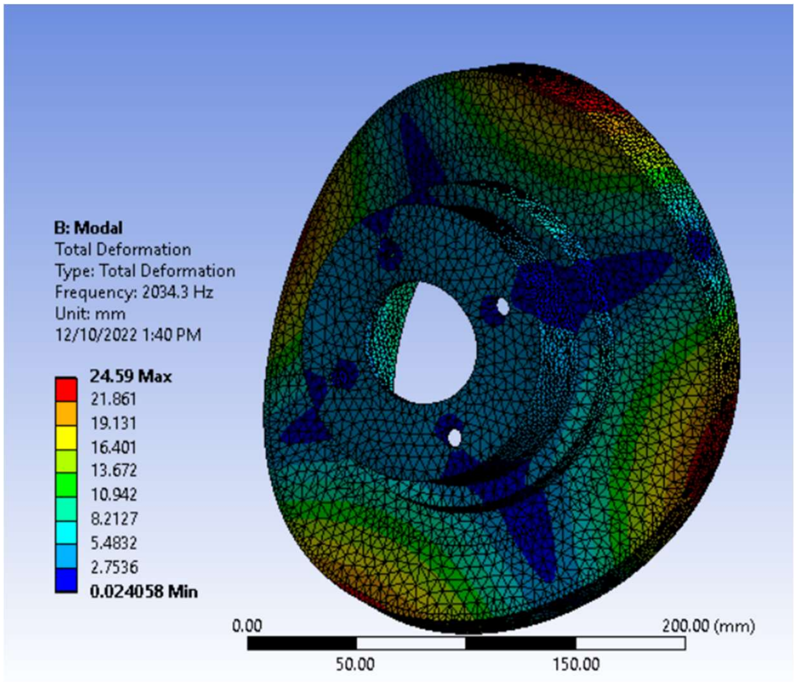

Figure 7: Mode shape plot for initial conditions on the design variables

4.4 Thermal Analysis

Transient thermal analysis is performed on the disc to observe the maximum temperature rise after the braking operation. The heat flux is calculated as shown below. It is assumed that 70% of the braking power is in the front axle of a four-wheeler vehicle. The total heat flux is also multiplied by 0.5 to get the flux generated by a single pad on the disc brake. Here, $t_e = 5s$;$A_{sp} = 0.021m^2$;$T_{amb} = 35^oC$.

Calculating the heat flux (q) generated on each face of the disc,

$$q = (KE * 0.5 * 0.7)/(t_e * A_{sp}) = 1.5395e6W/m^2 \tag{17}$$

Boundary conditions applied for the transient thermal analysis are as follows:

- Heat flux of $1.5395e6W/m^2$ is applied on the swept area of both the pads while the direction of heat flow is towards the disc.

- Convection is applied on all the surfaces with the air film coefficient of 5 W/m^2K which is the default value for standard air.

- Initial temperature is kept at 35^oC.

The analysis is performed for 5 s which is the braking time. The boundary conditions for this analysis are shown in Figure 8

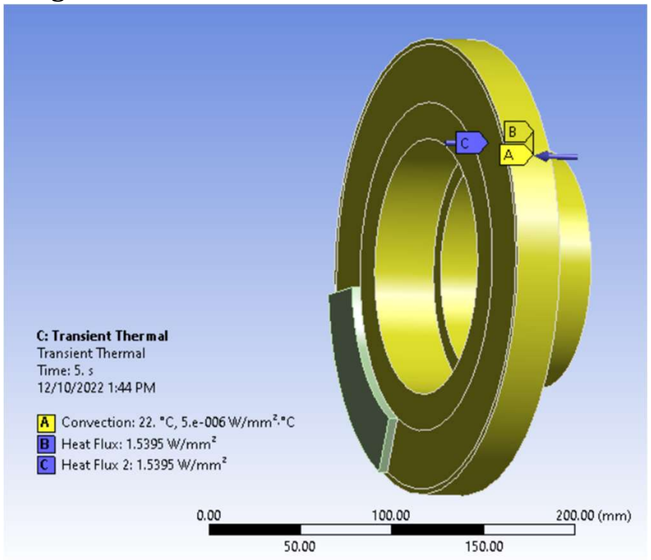

Figure 8: Transient thermal boundary conditions

The temperature plot obtained is as shown in Figure 9. As observed, the maximum temperature of 321^oC is obtained for initial conditions on the design variables.

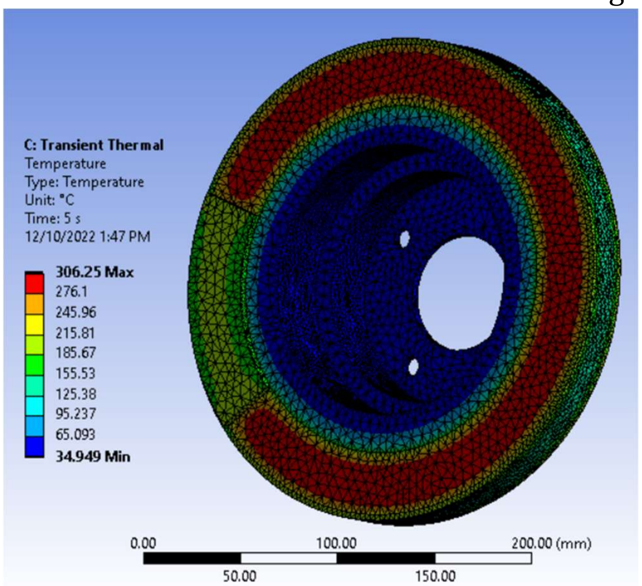

Figure 9: Temperature plot for initial conditions on the design variables

4.5 Design of Experiments

After the initial analysis of each subsystem, the relationship between the design variables and output response is determined. All the input variables are quantitative and continuous in nature. To obtain the accurate response surface, minimum number of design points from the given sample space are required. Latin Hypercube Sampling (LHS) technique with user defined sample points is used to create the response surface. The main advantage of LHS is that all the sample points are varying in nature. Unlike other sampling methods like Central Composite Design (CCD) and the full factorial methods, the number of simulations required for LHS remains constant even with an increasing number of parameters. After the design of experiments process, we obtain various combinations of input parameters and the corresponding response values. All the 10 DOE points are shown as follows-

Table of Schematic D2: Design of Experiments (Latin Hypercube Sampling Design : User-Defined Samples : Random Generator Seed = 0 : Number of Samples = 10)

	A	B	C	D	E	F	G	H	I
1	Name	P1 - rotor_thickness (mm)	P2 - rotor_OD (mm)	P3 - rotor_ID (mm)	P6 - Equivalent Stress Maximum (MPa)	P7 - Equivalent Stress Maximum Maximum Value Over Time (MPa)	P8 - Total Deformation Maximum (mm)	P9 - Temperature Maximum (C)	P10 - Solid Volume (mm^3)
2	1	24.75	131.75	70.75	12.803	12.803	23.567	303.69	1.1573E+06
3	2	26.25	125.75	84.55	16.272	16.272	23.834	302.34	9.6023E+05
4	3	22.75	137.75	82.25	26.08	26.08	23.399	310.13	1.1098E+06
5	4	25.75	134.75	68.45	13.899	13.899	22.476	301.09	1.2792E+06
6	5	23.25	133.25	73.05	22.429	22.429	24.11	308.35	1.1117E+06
7	6	27.25	136.25	75.35	62.206	62.206	21.48	298.11	1.3157E+06
8	7	25.25	139.25	79.95	17.956	17.956	21.863	302.26	1.2599E+06
9	8	26.75	130.25	77.65	14.918	14.918	22.745	299.28	1.1395E+06
10	9	24.25	128.75	86.85	40.186	40.186	24.192	305.87	9.4264E+05
11	10	23.75	127.25	66.15	25.274	25.274	25.043	307.85	1.0636E+06

4.6 Response Surface

After DOE, a response surface is generated for all the input and output values using the least squares methodology. The data points are fitted with a standard 2^{nd} order model. The points generated on the response surface are then used to perform the optimization. The goodness of fit plots for all the subsystems are shown below.

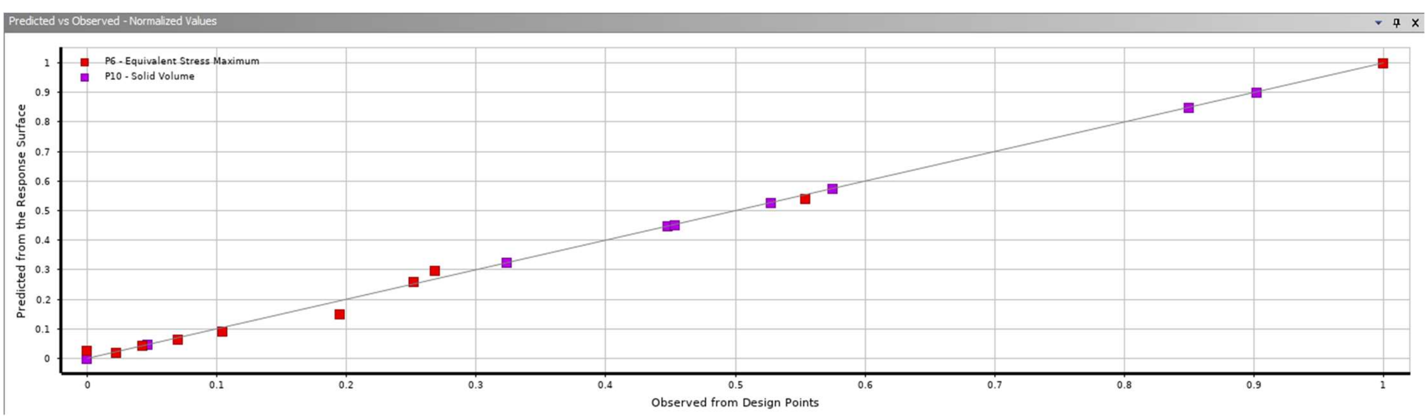

Figure 10: Goodness of fit plot for structural analysis

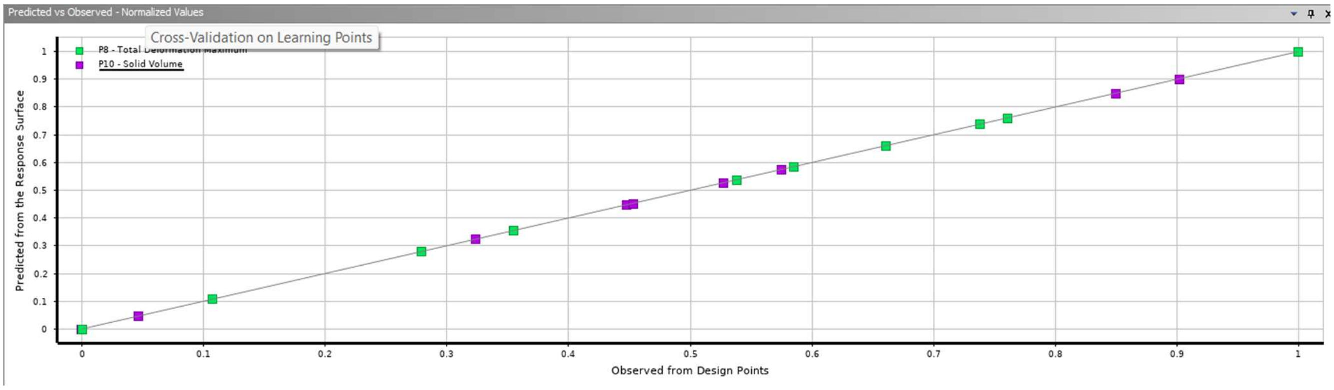

Figure 11: Goodness of fit plot for modal analysis

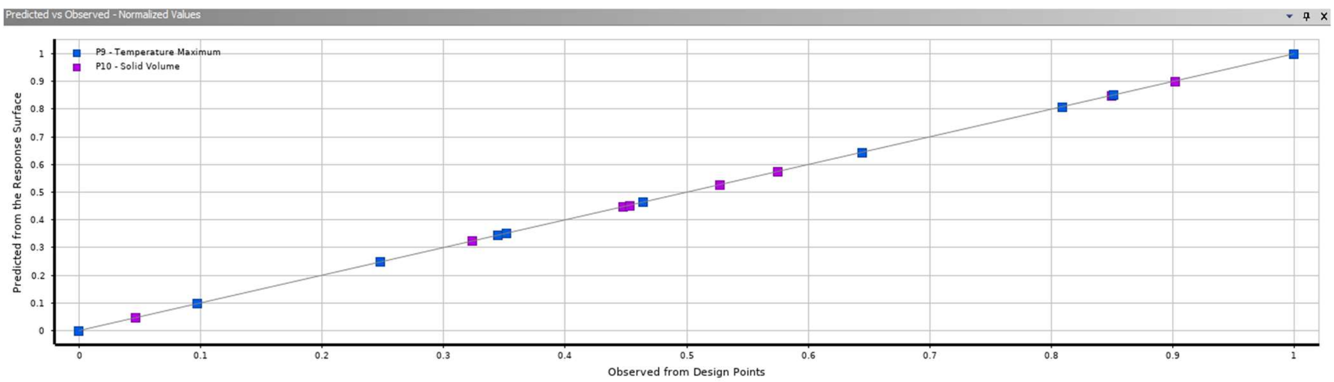

Figure 12: Goodness of fit plot for thermal analysis

5 Optimization Study

Study results from MOGA analysis method in Design Optimization.

Table of Schematic D4: Optimization , Candidate Points

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
1	Reference	Name	P1 - rotor_thickness (mm)		P2 - rotor_OD (mm)		P3 - rotor_ID (mm)		P6 - Equivalent Stress Maximum (MPa)		P7 - Equivalent Stress Maximum Maximum Value Over Time (MPa)	P8 - Total Deformation Maximum (mm)	P9 - Temperature Maximum (C)	P10 - Solid Volume (mm^3)	
2			Parameter Value	Variation from Reference	Parameter Value	Variation from Reference	Parameter Value	Variation from Reference	Parameter Value	Variation from Reference				Parameter Value	Variation from Reference
3		Candidate Point 1	22.754	0.34%	125.32	0.01%	80.356	0.52%	19.525	-1.54%	19.525	25.783	312.32	8.7821E+05	-0.07%
4		Candidate Point 1 (verified)		0.34%		0.01%		0.52%	15.121	-23.75%	15.121	25.6	313.22	8.9135E+05	1.42%
5		Candidate Point 2	22.772	0.42%	125.12	-0.15%	80.933	1.24%	19.823	-0.03%	19.823	25.808	312.44	8.7062E+05	-0.93%
6		Candidate Point 2 (verified)		0.42%		-0.15%		1.24%	14.003	-29.39%	14.003	25.607	313.42	8.8376E+05	0.56%
7		Candidate Point 3	22.677	0.00%	125.31	0.00%	79.942	0.00%	19.83	0.00%	19.83	25.819	312.58	8.7883E+05	0.00%
8		Candidate Point 3 (verified)		0.00%		0.00%		0.00%	12.998	-34.46%	12.998	25.655	313.59	8.9217E+05	1.52%
*		New Custom Candidate Point	25		132.5		76.5								

6 Discussion of Results

From previous section, design optimization study we found that candidate-2 is giving decent results in terms of maximum stress, maximum temperature and total maximum deformation. The dimensions for candidate-2 are –

Rotor thickness = 22.772 mm
Rotor OR = 125.12 mm
Rotor IR = 80.933mm
New Volume = 8.7062e+05 mm^3

Initial Volume = 9.967e+05 mm^3

$$\text{Weight Reduction} = \frac{\text{Initial Volume} - \text{New Volume}}{\text{Initial Volume}} = \underline{12.65\ \%}$$

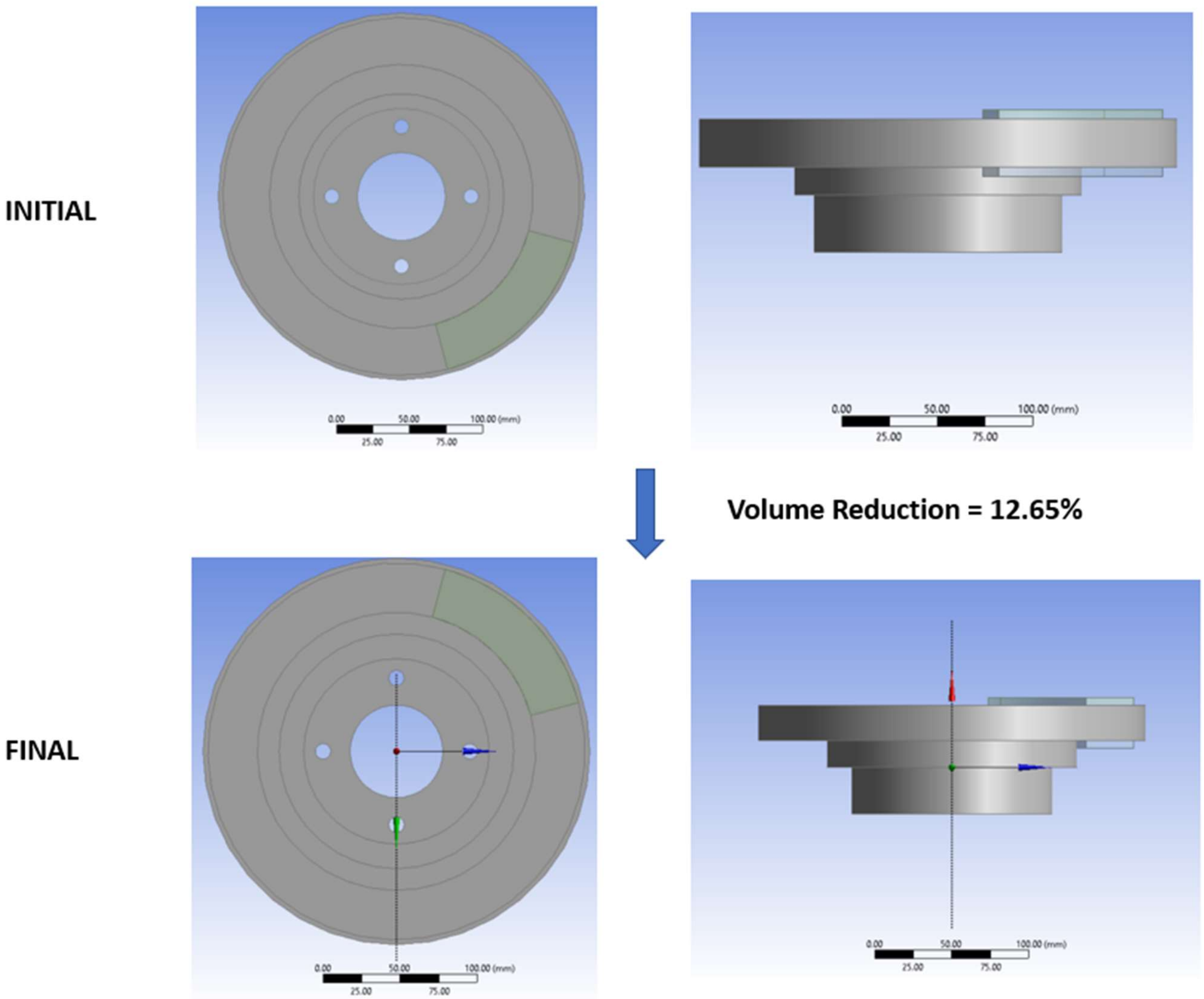

Figure 13: Initial and final brake disc geometries