ANSYS DOE and Design Optimization on Vehicle Brake Design

PROJECT REPORT

YONESHWAR BABU

ASU ID:1220454365

ARIZONA STATE UNIVERSITY

MAE 598: Design Optimization (Fall2021)

November 17, 2021

Under the guidance of

Yi Ren

Assistant Professor of aerospace and mechanical engineering

School for Engineering of Matter, Transport and Energy

Arizona State University

ABSTRACT

The primary objective of this project **"ANSYS DOE and Design Optimization on Vehicle Brake Design"** determine the optimum dimensions for the brake disc for a four-wheeler vehicle using ANSYS. These dimensions are for the disc inner radius, outer radius and thickness. Static Structural, modal and Transient 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 ANSYS. The system is optimized using MOGA by integrating all the load cases.

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

The three subsystems are as follows:

Structural Analysis: The brake disc has to 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.

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.

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.

Mathematical Model:

In this optimization study, the brake disc inner radius (P1), outer radius (P2) and thickness (P3) are the design variables. Firstly, the static structural, modal and Transient 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 regression analysis on these DOE points to obtain the volume, stress, frequency and temperature quadratic functions. The optimization problem is designed as follows:

Minimize : f1 : V =

Geometrical constraints for all subsystems:

$$g1 : -P1 \leq -66$$

$$g2 : P1 \leq 90$$

$$g3 : -P2 \leq -124$$

$$g4 : P2 \leq 150$$

$$g5 : -P3 \leq -5$$

$$g6 : P3 \leq 27$$

Design constraints:

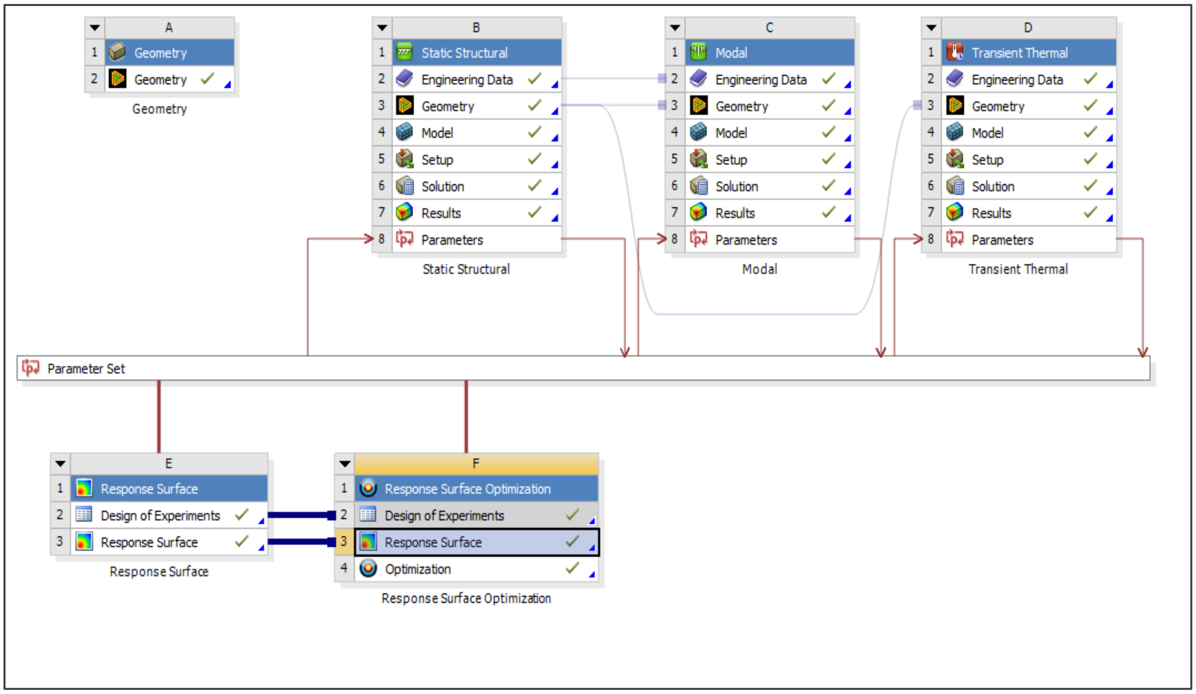
$$g7 : S \leq 14\text{MPa}$$

$$g8 : -F \leq -1200 \text{ Hz}$$

$$g9 : T \leq 400 \text{ C}$$

MODEL ANALYSIS:

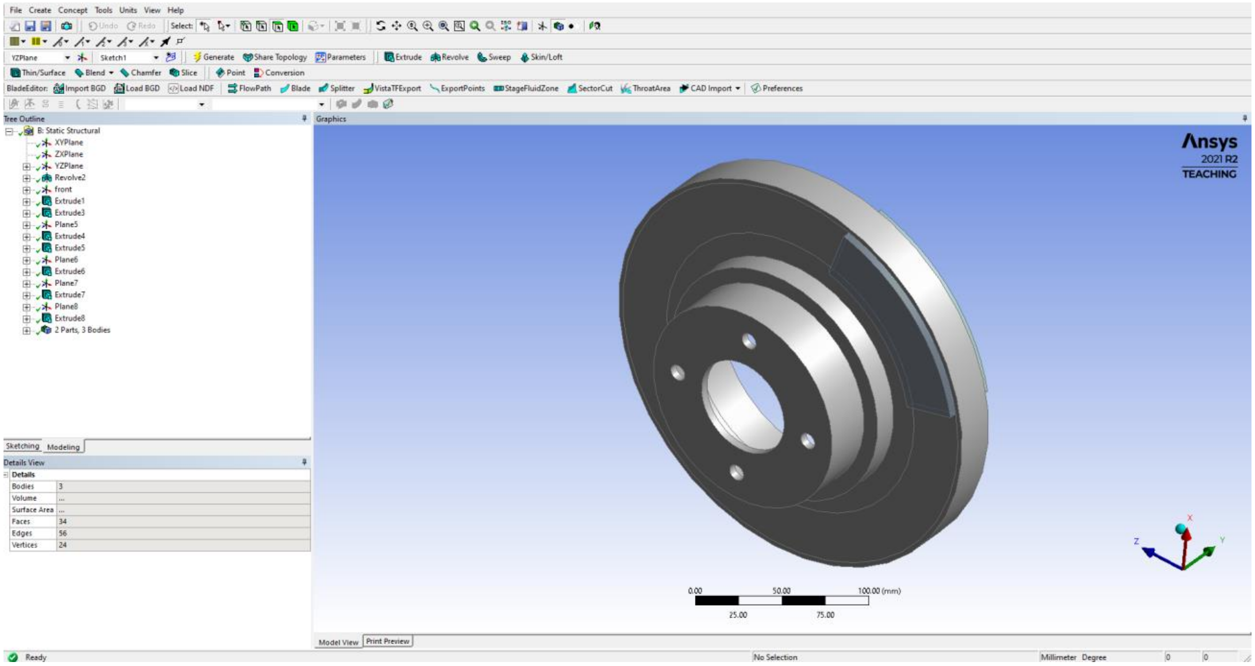
The flowchart for optimization in ANSYS is shown in the Figure. This figure pertains to the flowchart for structural analysis. Similar procedure is followed for modal and thermal analyses.



Ansys Optimization flowchart

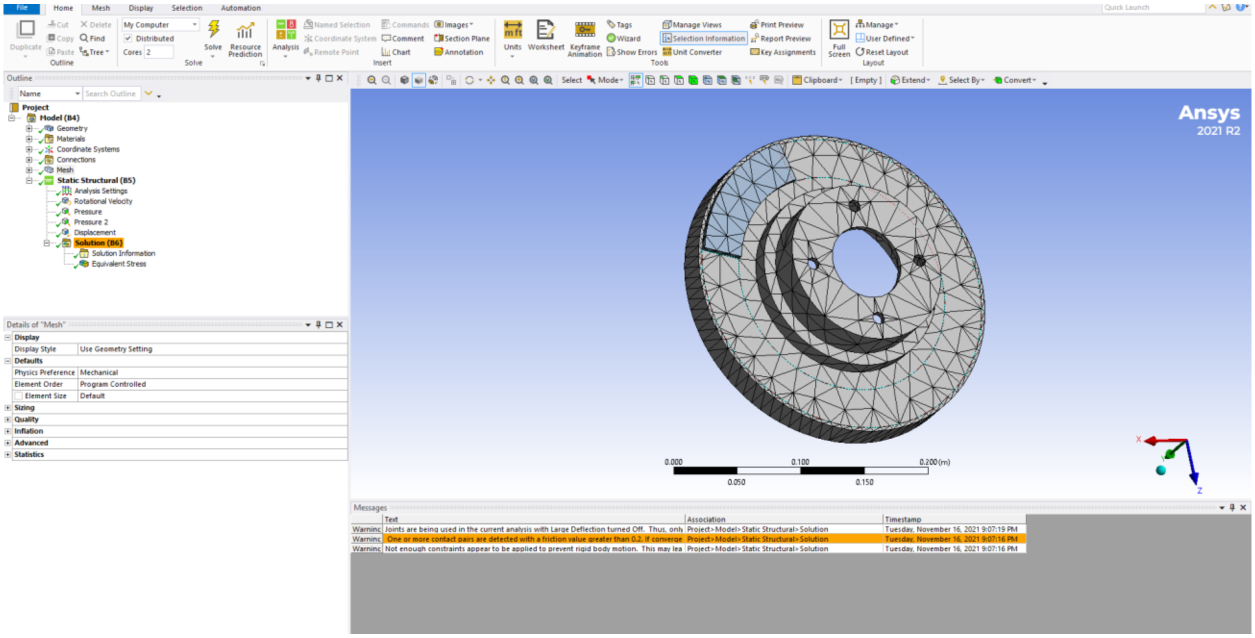
FE Model

The brake disc geometry is prepared in ANSYS Design Modeler as shown in Figure. 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.



Disc Brake CAD Model

The CAD geometry is meshed using 3 mm sized tetrahedral quadratic elements as shown in the Figure.



Disc Brake Mesh Model

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

Property	**Value**	**Unit**
Density	7200	kg/m3
Young's modulus	110	GPa
Poisson's ratio	0.28	
Thermal conductivity	52	W/m C
Specific heat	447	J/kgoC

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.

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;te = 5s; u = 90kmph; m = 1500kg; ν = 0.22; Ap = 3552mm2

Calculating the disc angular velocity (ω),
ω = u/Ravg = 250rad/s (14)
Evaluating the total braking force on the disc
(Fb), Fb = (KE ∗ 0.5 ∗ 0.7)/d = 16.406kN (15)

Formulating the pressure exerted by one brake pad on the disc
(P), P = Fb/(2 ∗ Ap ∗ ν) = 10.495MP a

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.

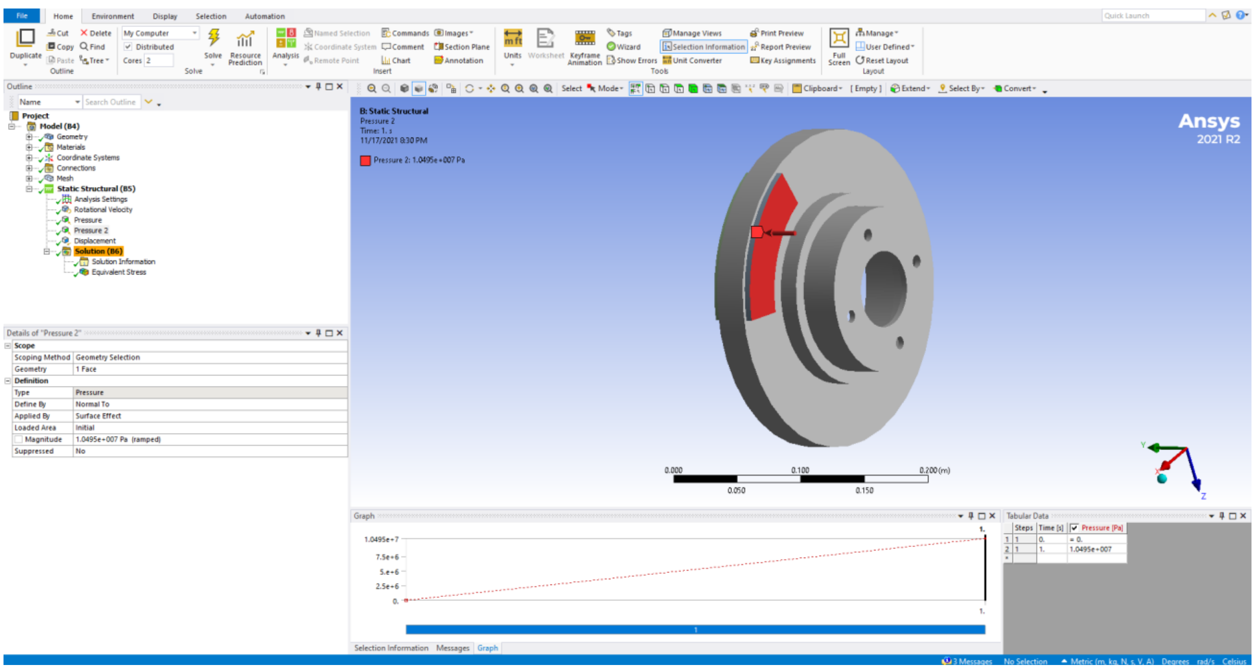
• The brake caliper pads are constrained in X and Z directions. They are kept free to move in the Y direction.

• 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.

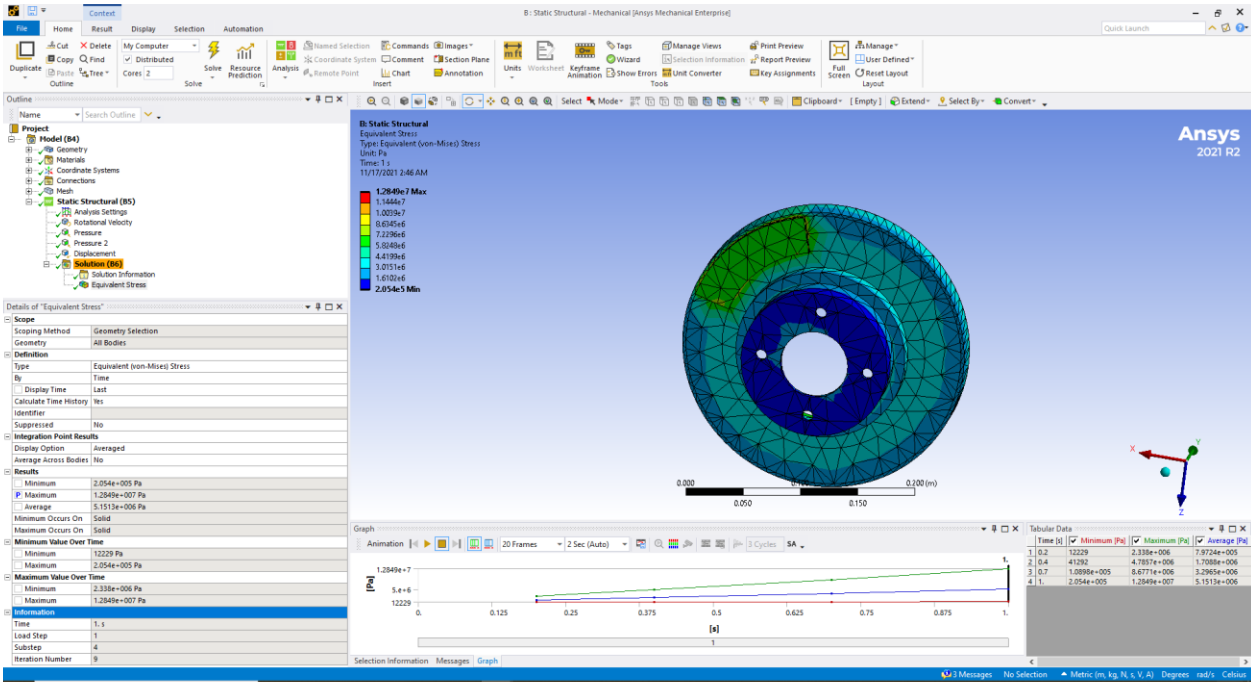
• Actuating pressure of 10.496 MPa is applied on the brake pads

.



Static structural boundary conditions

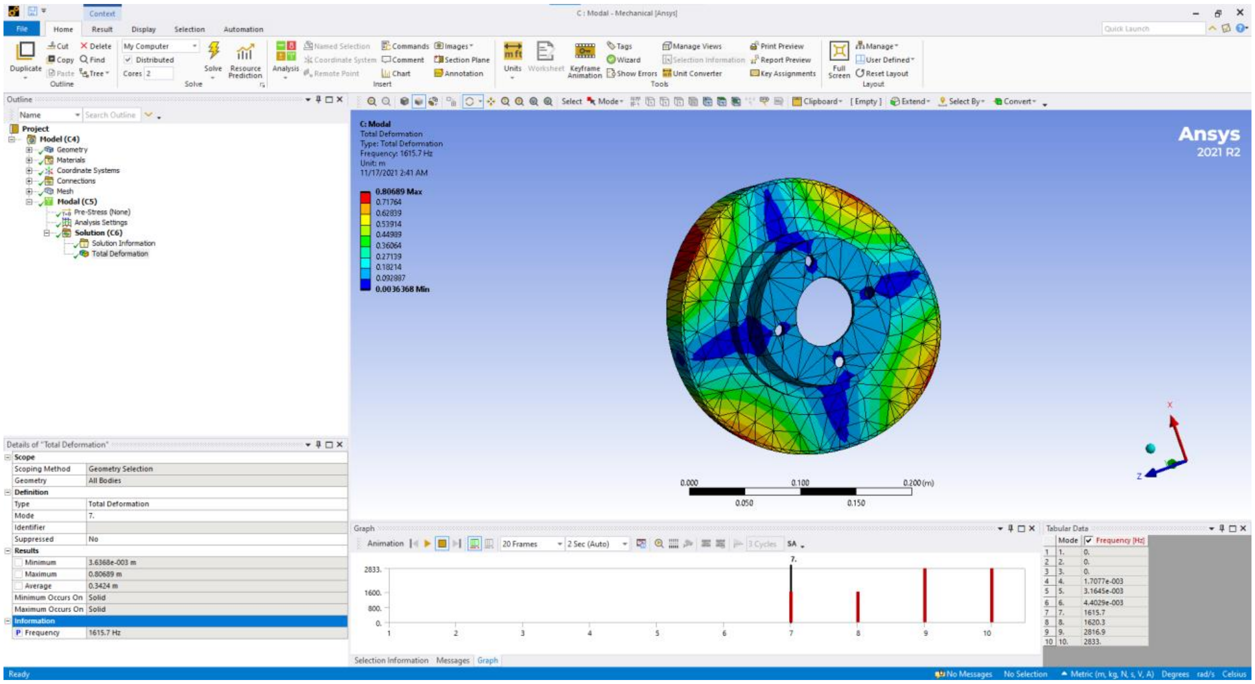
The stress plot obtained is as shown in Figure 6. As observed, the maximum stress of 12.849 MPa is obtained for initial conditions on the design variables.



Stress plot for initial conditions on the design variables

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.



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, te = 5s; Asp = 0.021m2 ; Tamb = 35 C

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

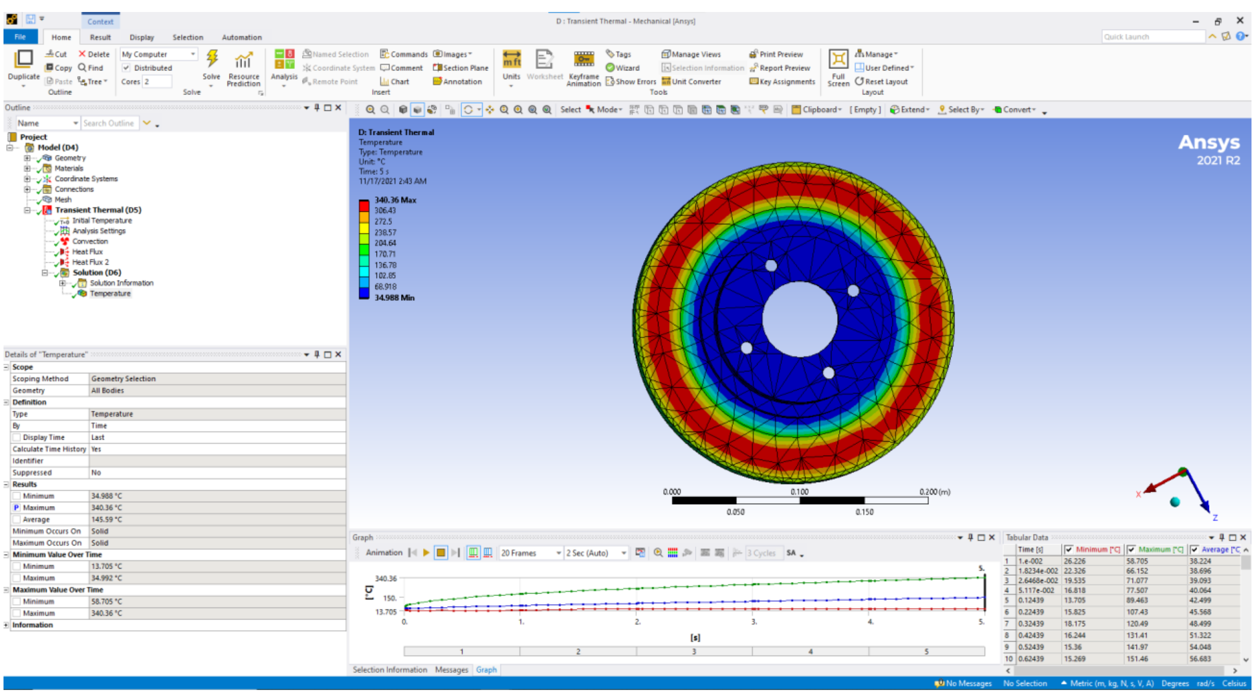
$$q = (KE * 0.5 * 0.7)/(te * Asp) = 1.5395e6W/m2$$

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

• Heat flux of 1.5395e6W/m2 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 5W/m2K which is the default value for standard air.

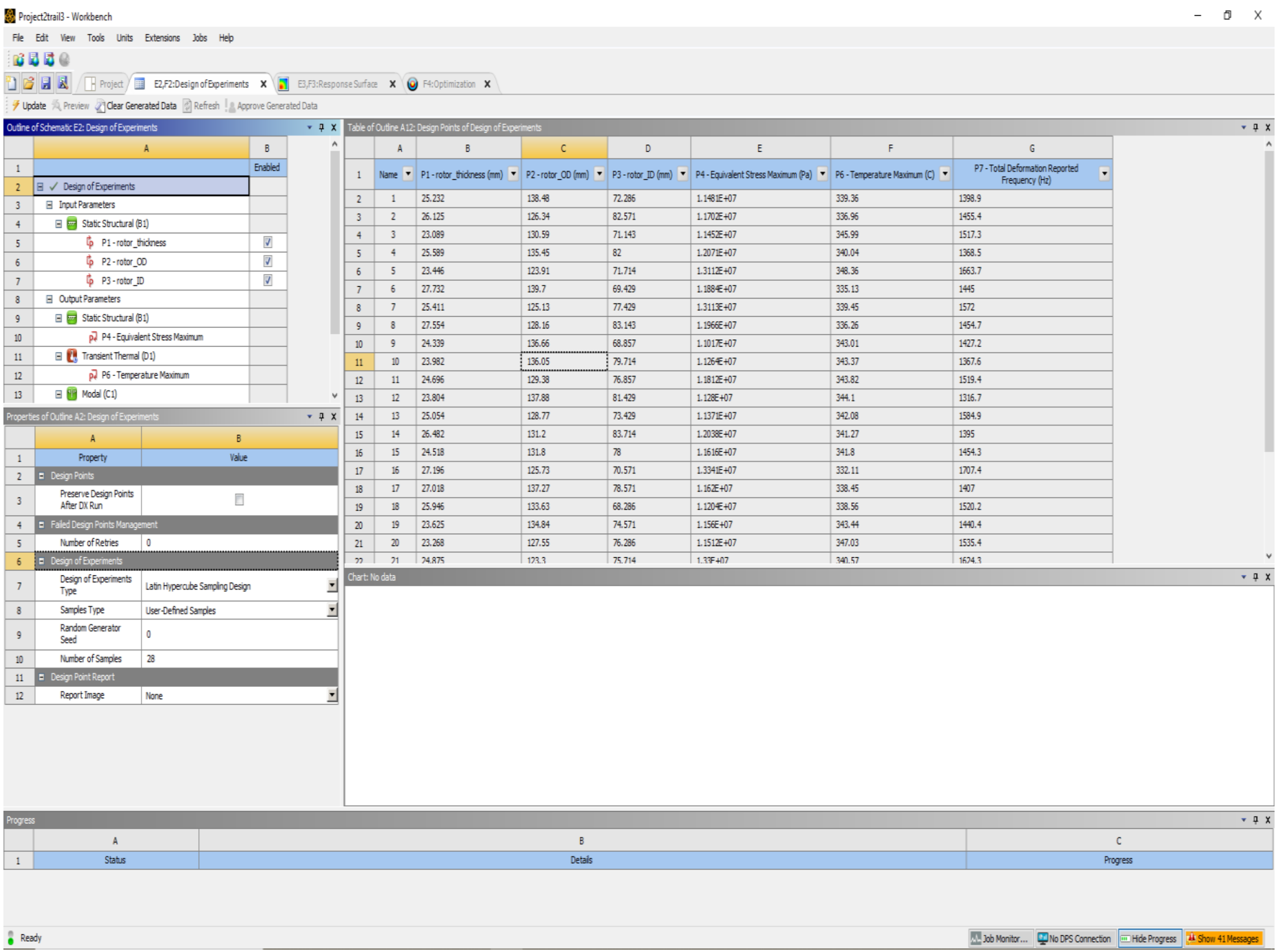
• Initial temperature is kept at 35oC. 14 The analysis is performed for 5 s which is the braking time. The boundary conditions for this analysis are shown



Temperature plot for initial conditions on the design variables

Design of Experiments:

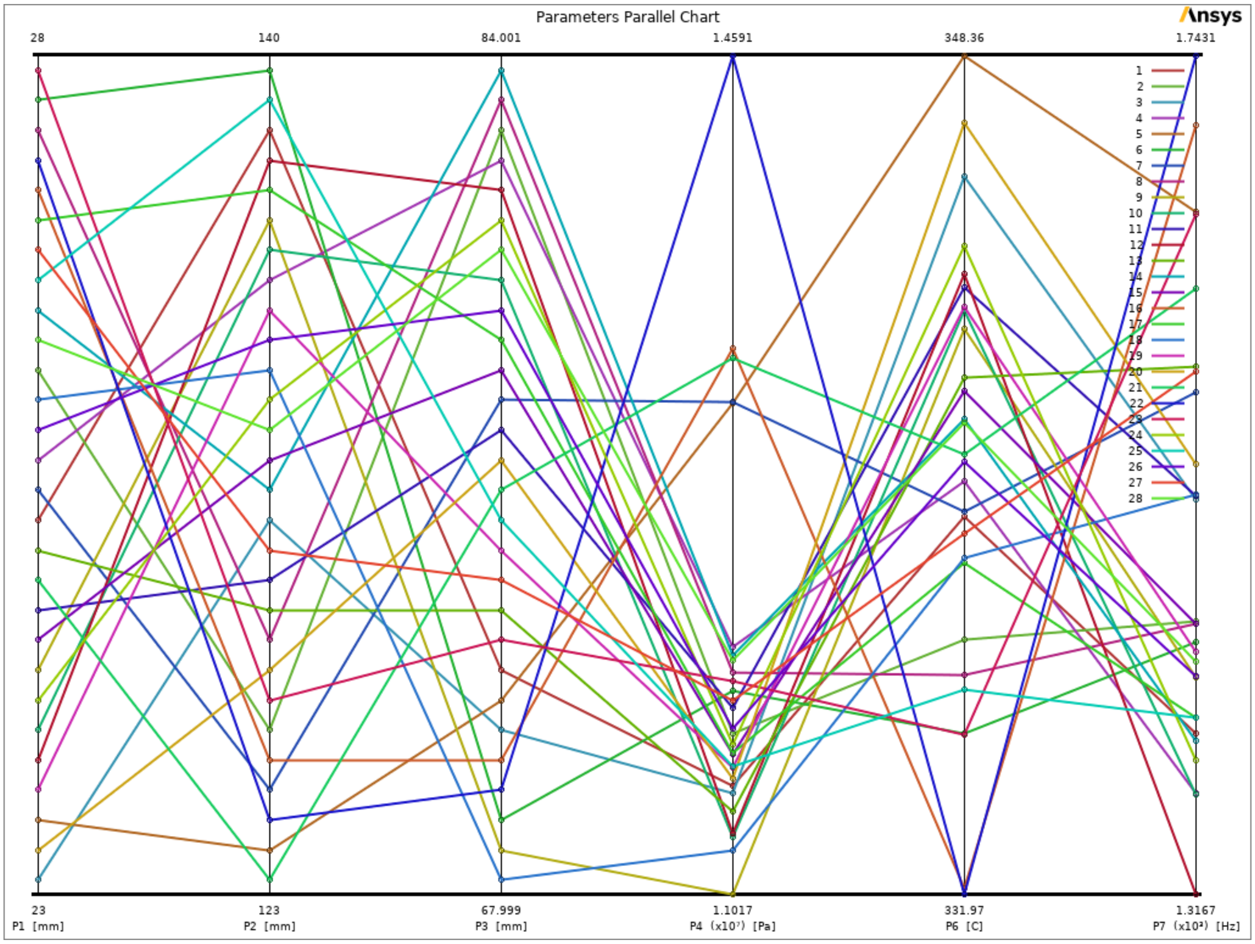
After the initial analysis of each subsystem, the relationship between the design variables and output response is determined. All of the input variables are quantitative and continuous 15 in nature. To obtain the accurate response surface, minimum number of design points from the given sample space 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 28 DOE points are shown in Appendix A.



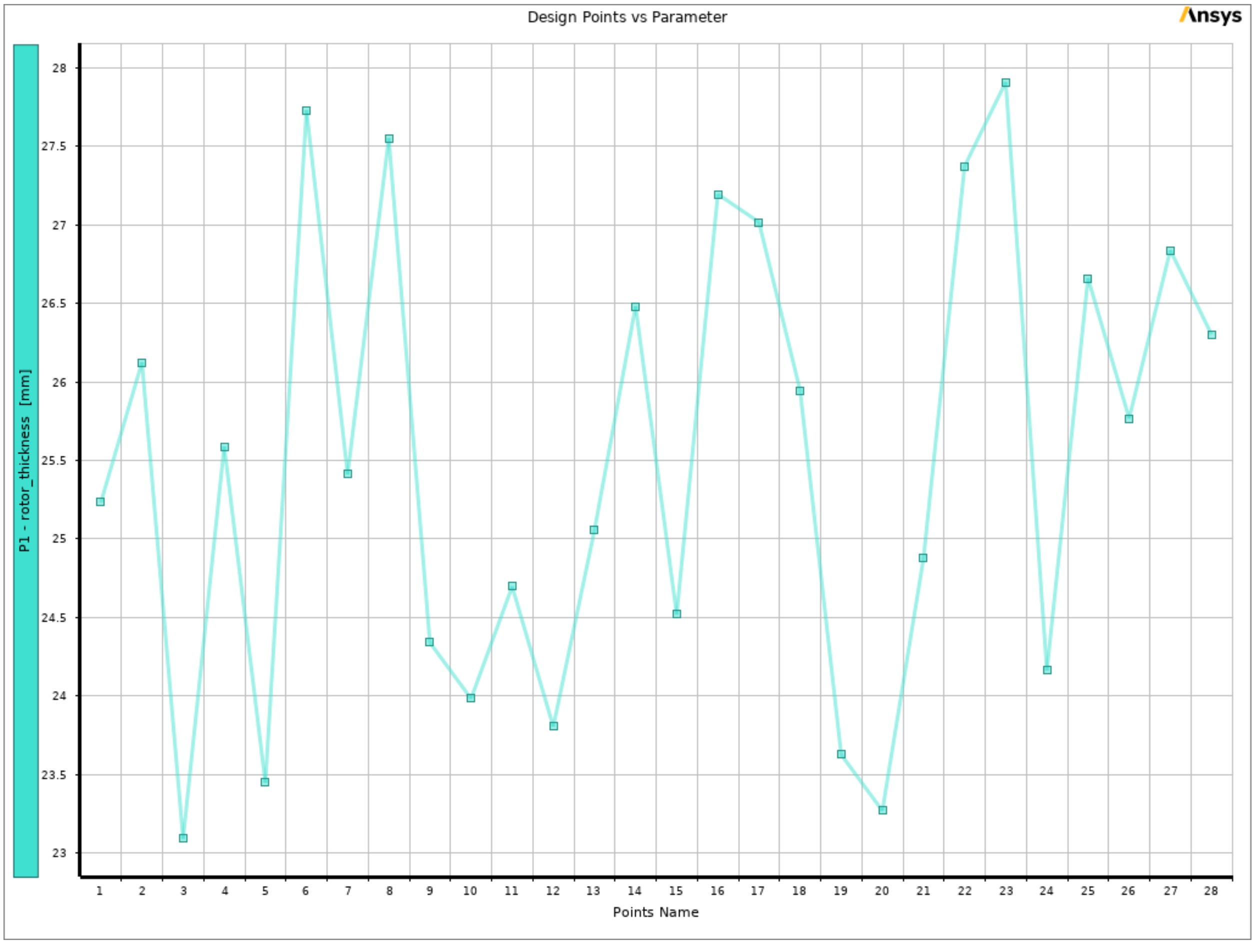
	A	B	C	D	E	F	G
1	Name	P1 - rotor_thickness (mm)	P2 - rotor_OD (mm)	P3 - rotor_ID (mm)	P4 - Equivalent Stress Maximum (Pa)	P6 - Temperature Maximum (C)	P7 - Total Deformation Reported Frequency (Hz)
2	1	25.232	138.48	72.286	1.1481E+07	339.36	1398.9
3	2	26.125	126.34	82.571	1.1702E+07	336.96	1455.4
4	3	23.089	130.59	71.143	1.1452E+07	345.99	1517.3
5	4	25.589	135.45	82	1.2071E+07	340.04	1368.5
6	5	23.446	123.91	71.714	1.3112E+07	348.36	1663.7
7	6	27.732	139.7	69.429	1.1884E+07	335.13	1445
8	7	25.411	125.13	77.429	1.3113E+07	339.45	1572
9	8	27.554	128.16	83.143	1.1966E+07	336.26	1454.7
10	9	24.339	136.66	68.857	1.1017E+07	343.01	1427.2
11	10	23.982	136.05	79.714	1.1264E+07	343.37	1367.6
12	11	24.696	129.38	76.857	1.1812E+07	343.82	1519.4
13	12	23.804	137.88	81.429	1.128E+07	344.1	1316.7
14	13	25.054	128.77	73.429	1.1371E+07	342.08	1584.9
15	14	26.482	131.2	83.714	1.2038E+07	341.27	1395
16	15	24.518	131.8	78	1.1616E+07	341.8	1454.3
17	16	27.196	125.73	70.571	1.3341E+07	332.11	1707.4
18	17	27.018	137.27	78.571	1.162E+07	338.45	1407
19	18	25.946	133.63	68.286	1.1204E+07	338.56	1520.2
20	19	23.625	134.84	74.571	1.156E+07	343.44	1440.4
21	20	23.268	127.55	76.286	1.1512E+07	347.03	1535.4
22	21	24.875	123.3	75.714	1.33E+07	340.57	1624.3

Response Surface

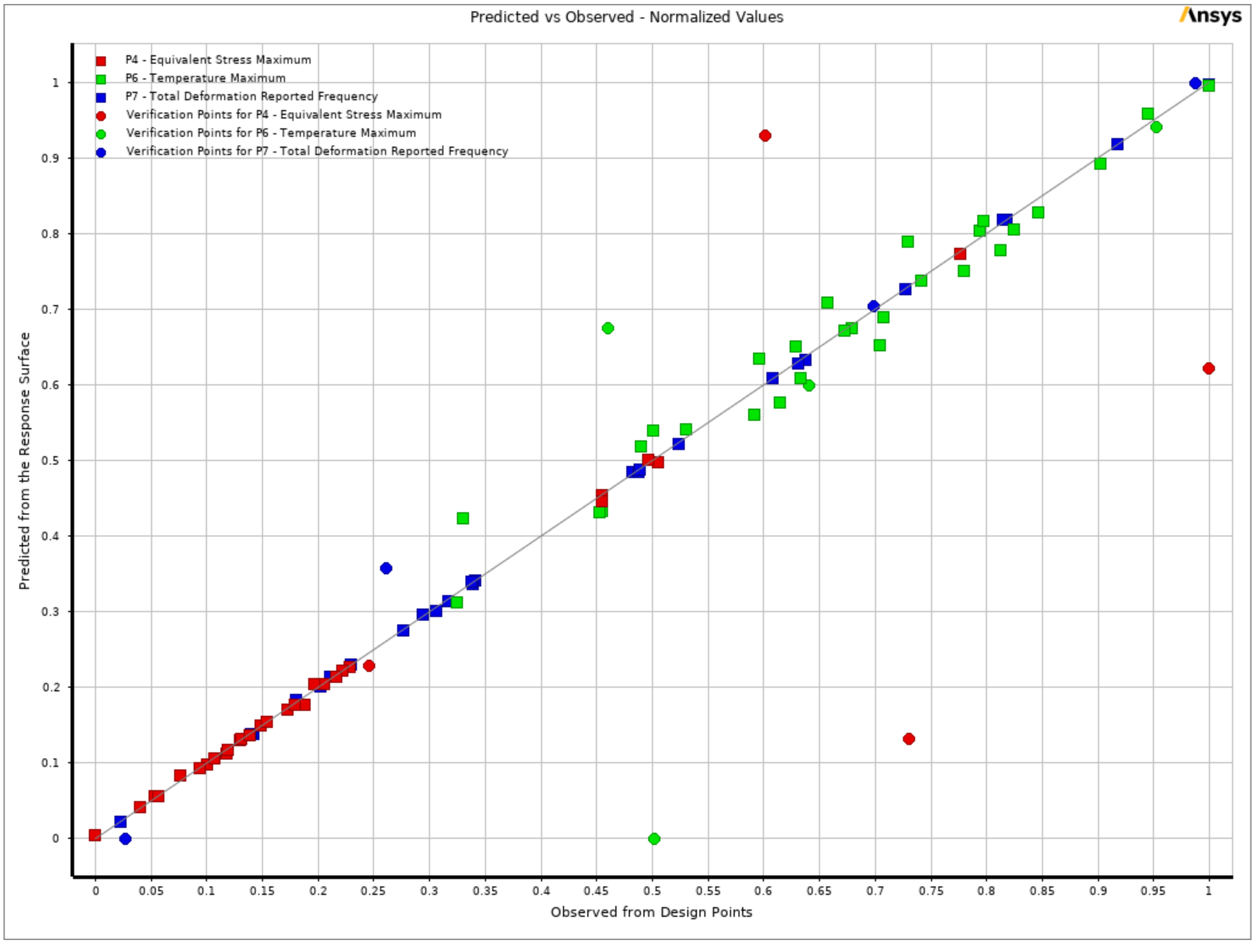
After DOE, a response surface is generated for all the input and output values using the least squares methodology. The data points are fitted with a standard 2nd order model. The points generated on the response surface are then used to perform the optimization. The goodness of fit plots for all the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



Parameters parallel

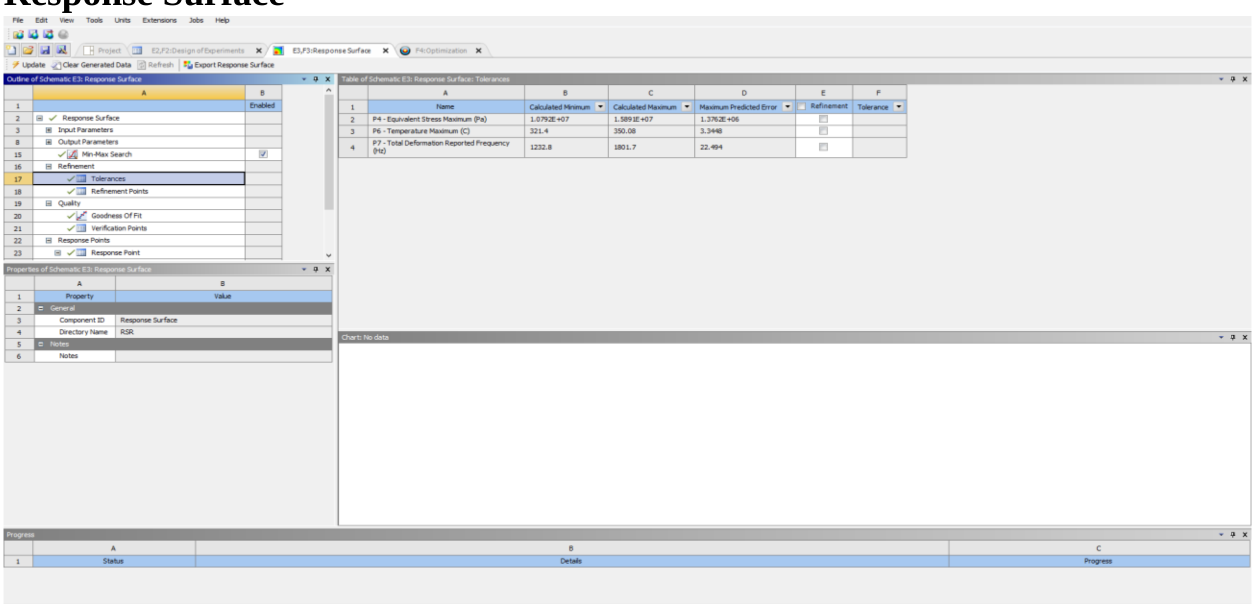


Points vs parameters

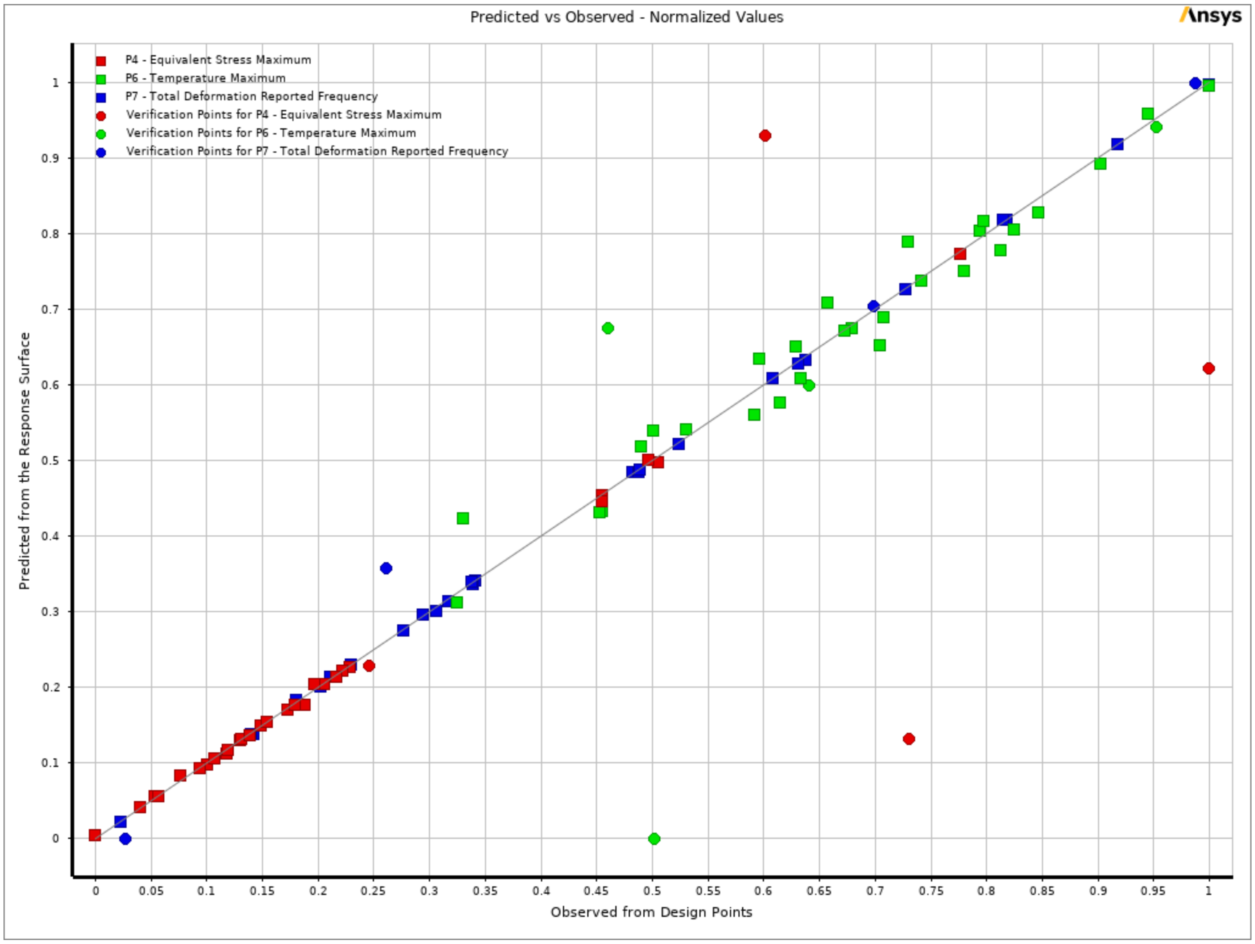


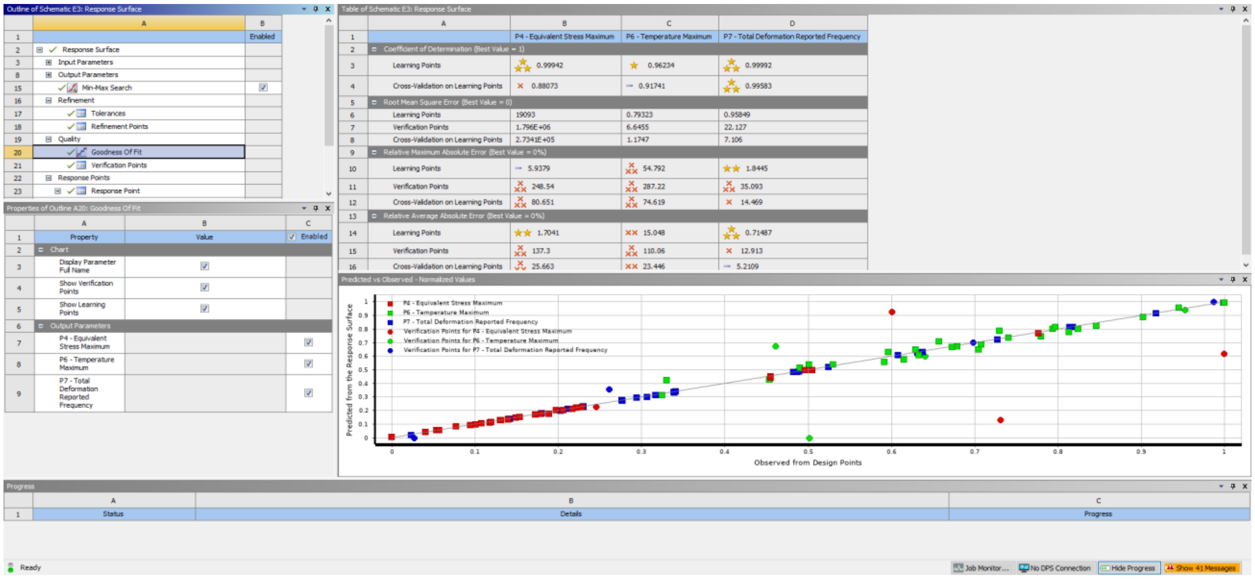
Goodness of fit plot for structural analysis

Response Surface

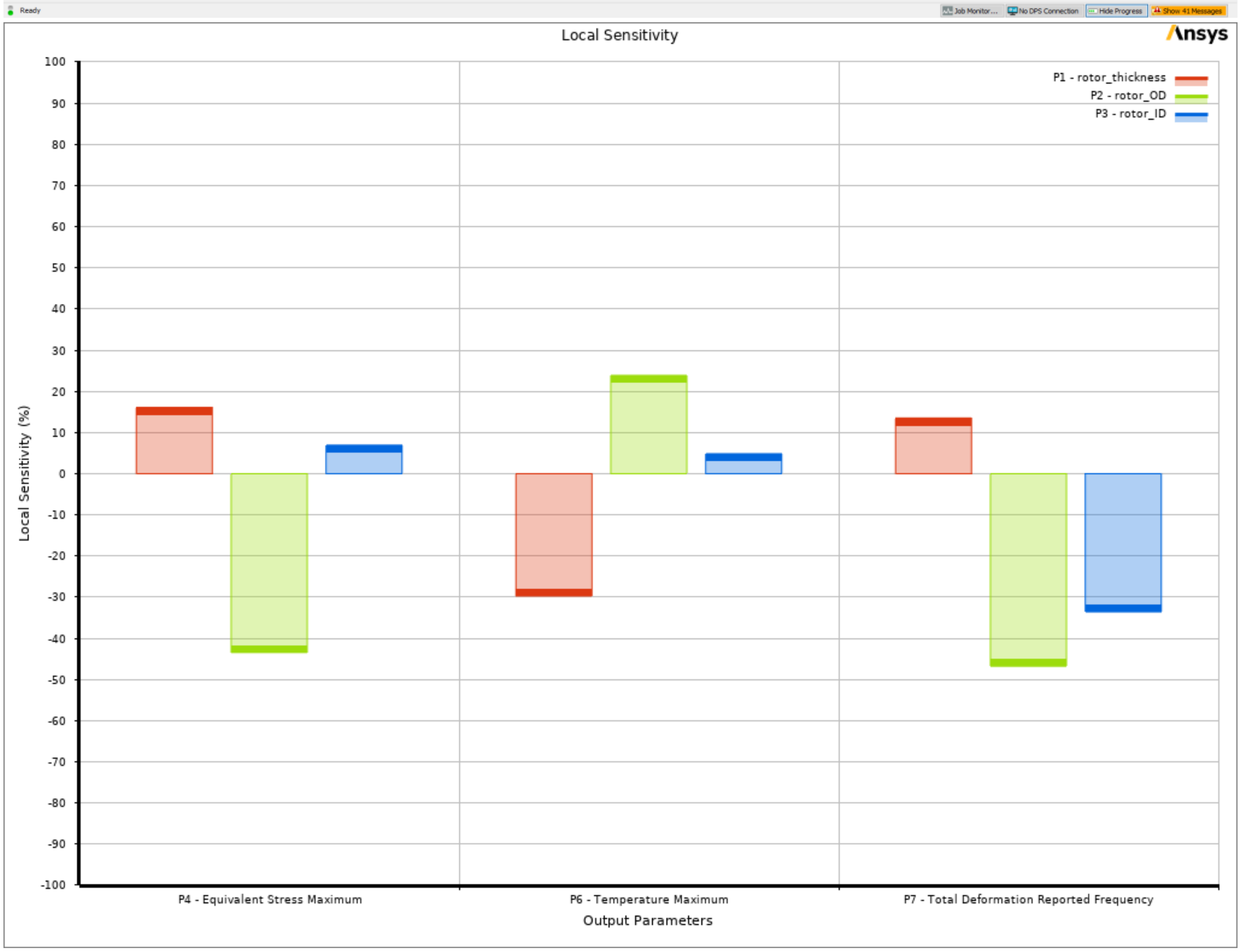


e

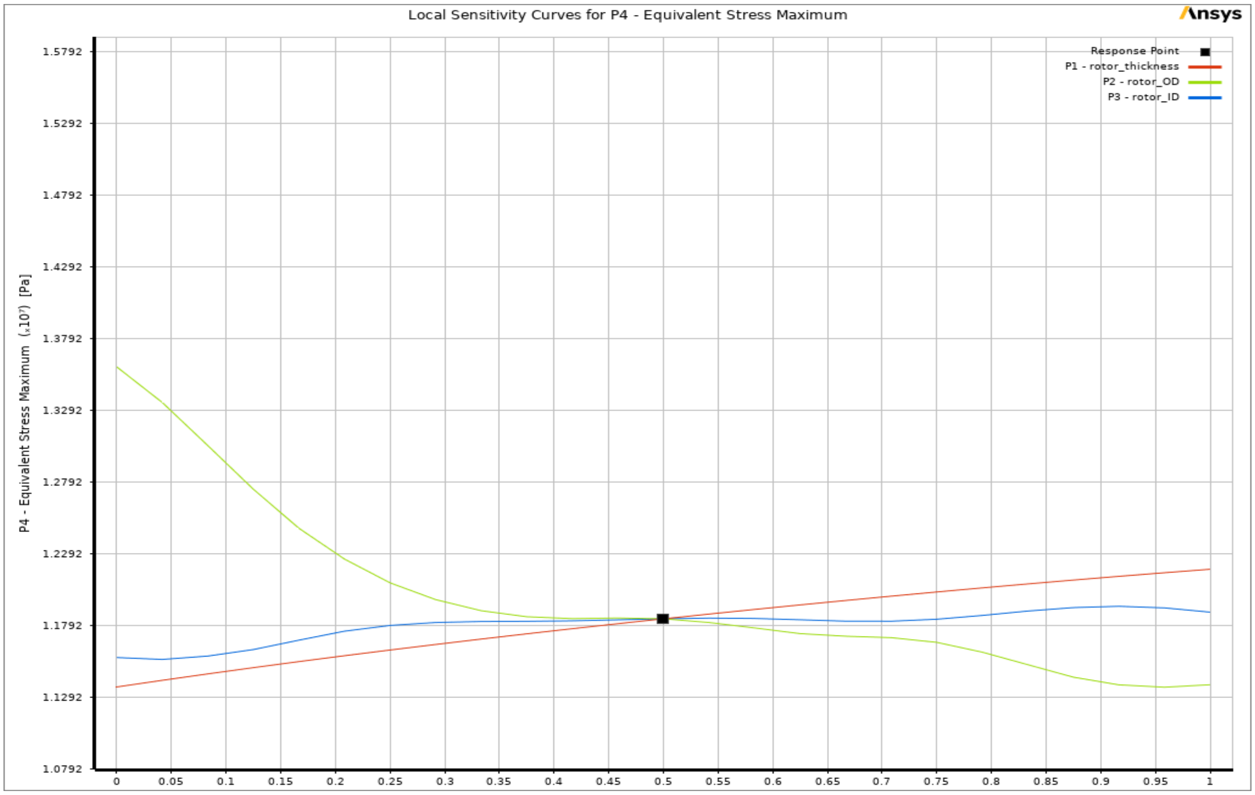

Predicted vs Observed - Normalized Values
P4 - Equivalent Stress Maximum
P6 - Temperature Maximum
P7 - Total Deformation Reported Frequency
Verification Points for P4 - Equivalent Stress Maximum
Verification Points for P6 - Temperature Maximum
Verification Points for P7 - Total Deformation Reported Frequency
Predicted from the Response Surface
Observed from Design Points

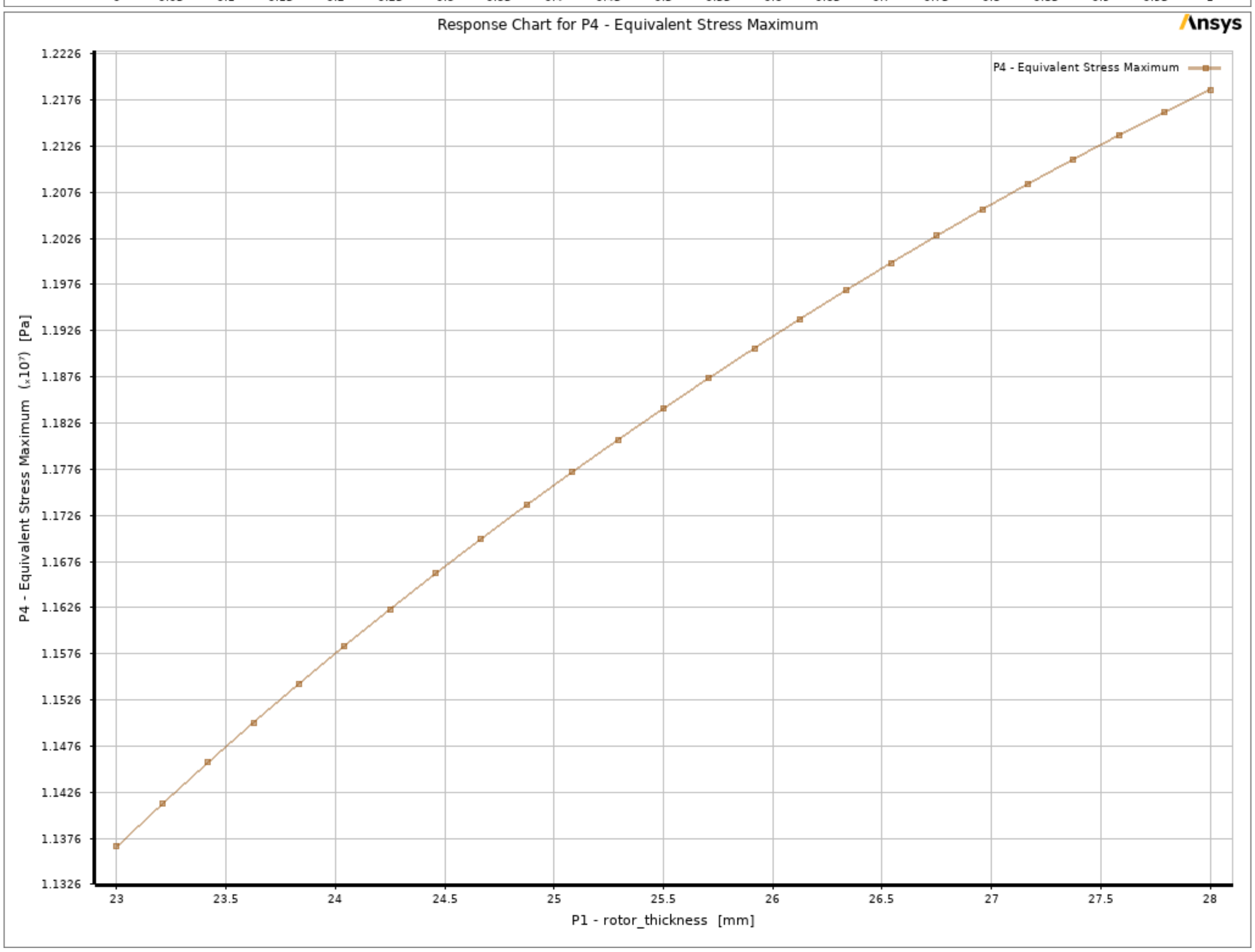


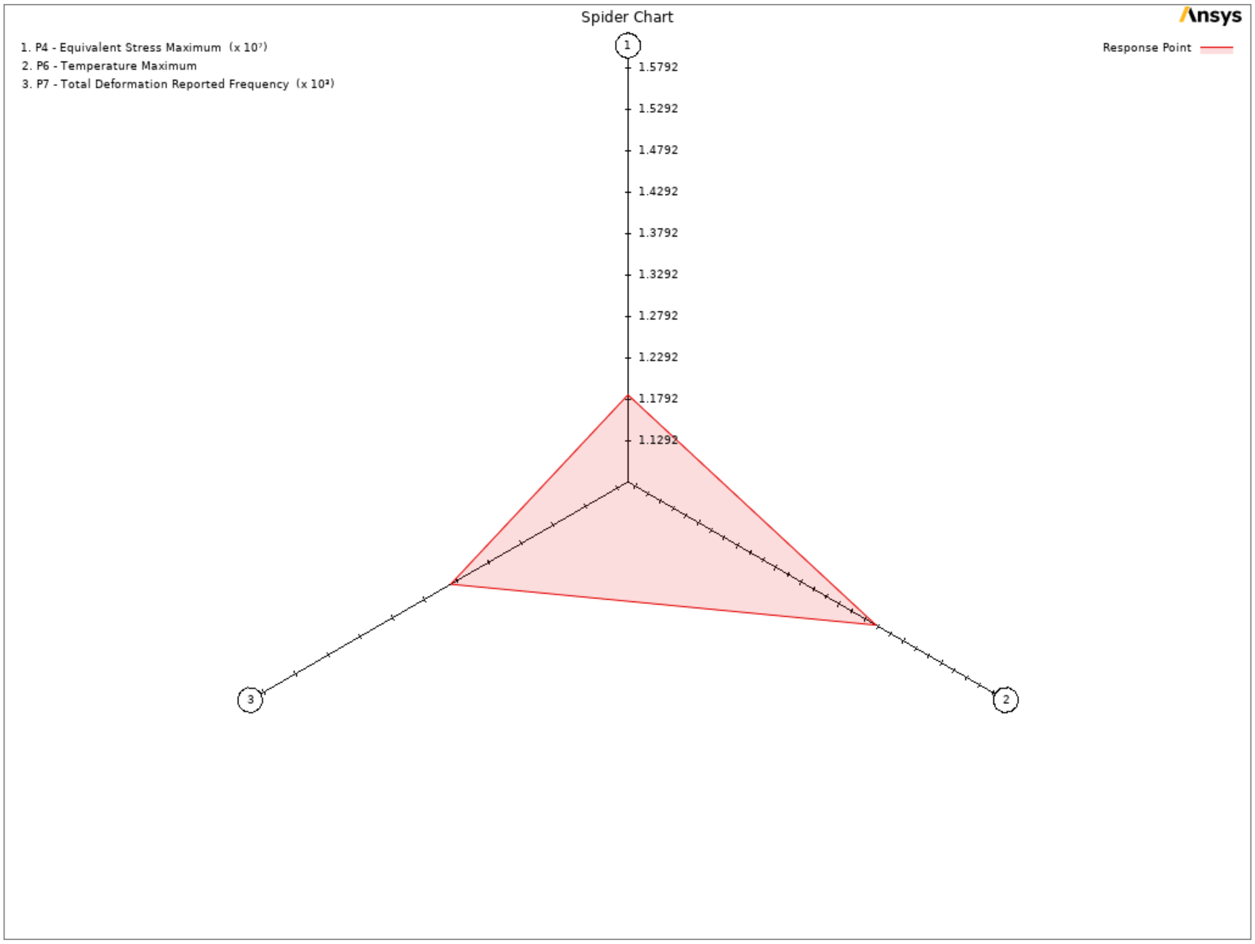
Outline of Schematic E3: Response Surface
Table of Schematic E3: Response Surface
Properties of Outline A20: Goodness Of Fit
Predicted vs Observed - Normalized Values
Observed from Design Points
Predicted from the Response Surface
Progress
Ready



Local Sensitivity
Ansys
P1 - rotor_thickness
P2 - rotor_OD
P3 - rotor_ID
Local Sensitivity (%)
P4 - Equivalent Stress Maximum
P6 - Temperature Maximum
P7 - Total Deformation Reported Frequency
Output Parameters


Local Sensitivity Curves for P4 - Equivalent Stress Maximum
Ansys
Response Point
P1 - rotor_thickness
P2 - rotor_OD
P3 - rotor_ID
P4 - Equivalent Stress Maximum (x10⁷) [Pa]
1.5792
1.5292
1.4792
1.4292
1.3792
1.3292
1.2792
1.2292
1.1792
1.1292
1.0792
0
0.05
0.1
0.15
0.2
0.25
0.3
0.35
0.4
0.45
0.5
0.55
0.6
0.65
0.7
0.75
0.8
0.85
0.9
0.95
1


Response Chart for P4 - Equivalent Stress Maximum
Ansys
P4 - Equivalent Stress Maximum
P4 - Equivalent Stress Maximum (x10⁷) [Pa]
1.2226
1.2176
1.2126
1.2076
1.2026
1.1976
1.1926
1.1876
1.1826
1.1776
1.1726
1.1676
1.1626
1.1576
1.1526
1.1476
1.1426
1.1376
1.1326
23
23.5
24
24.5
25
25.5
26
26.5
27
27.5
28
P1 - rotor_thickness [mm]



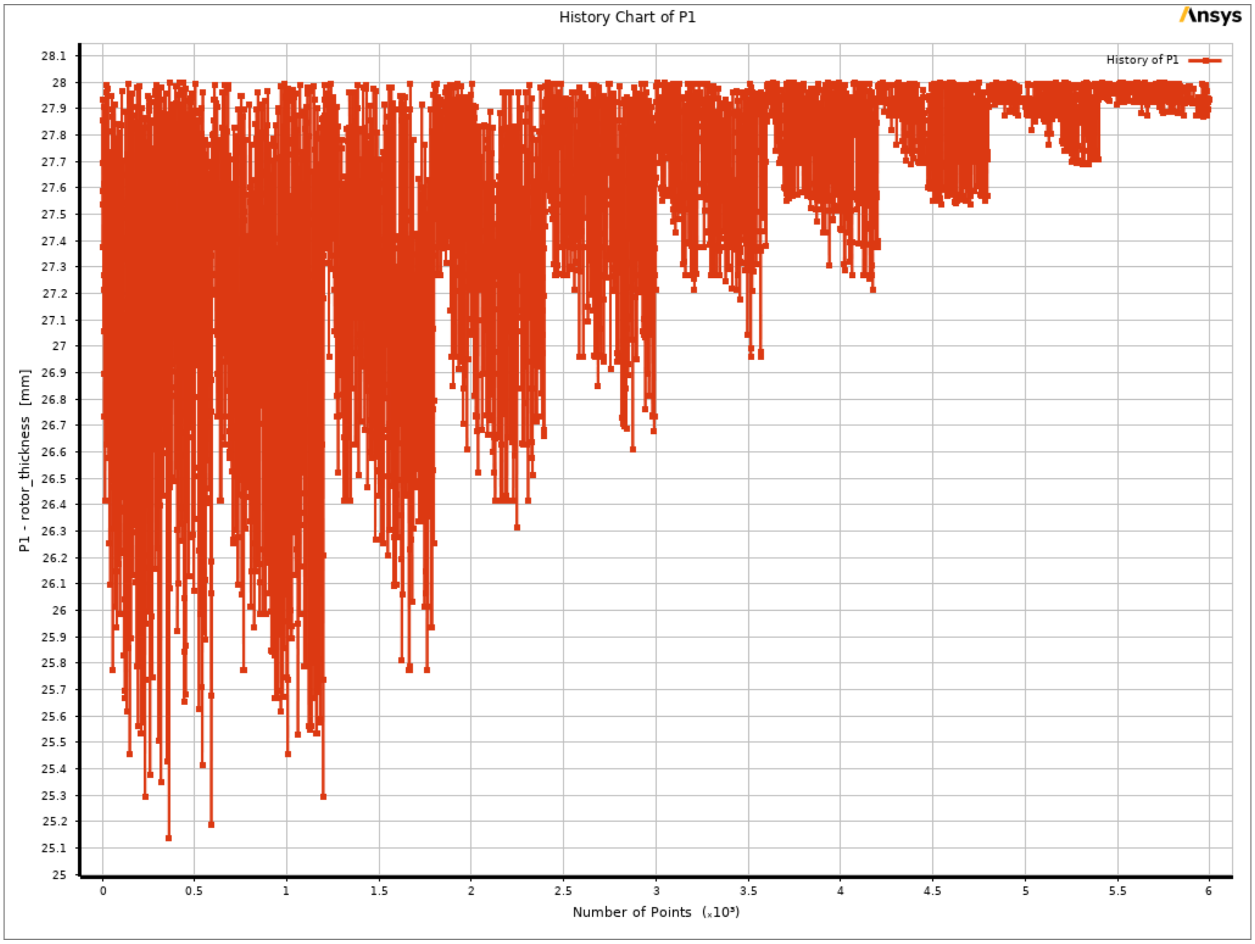
Optimization Study

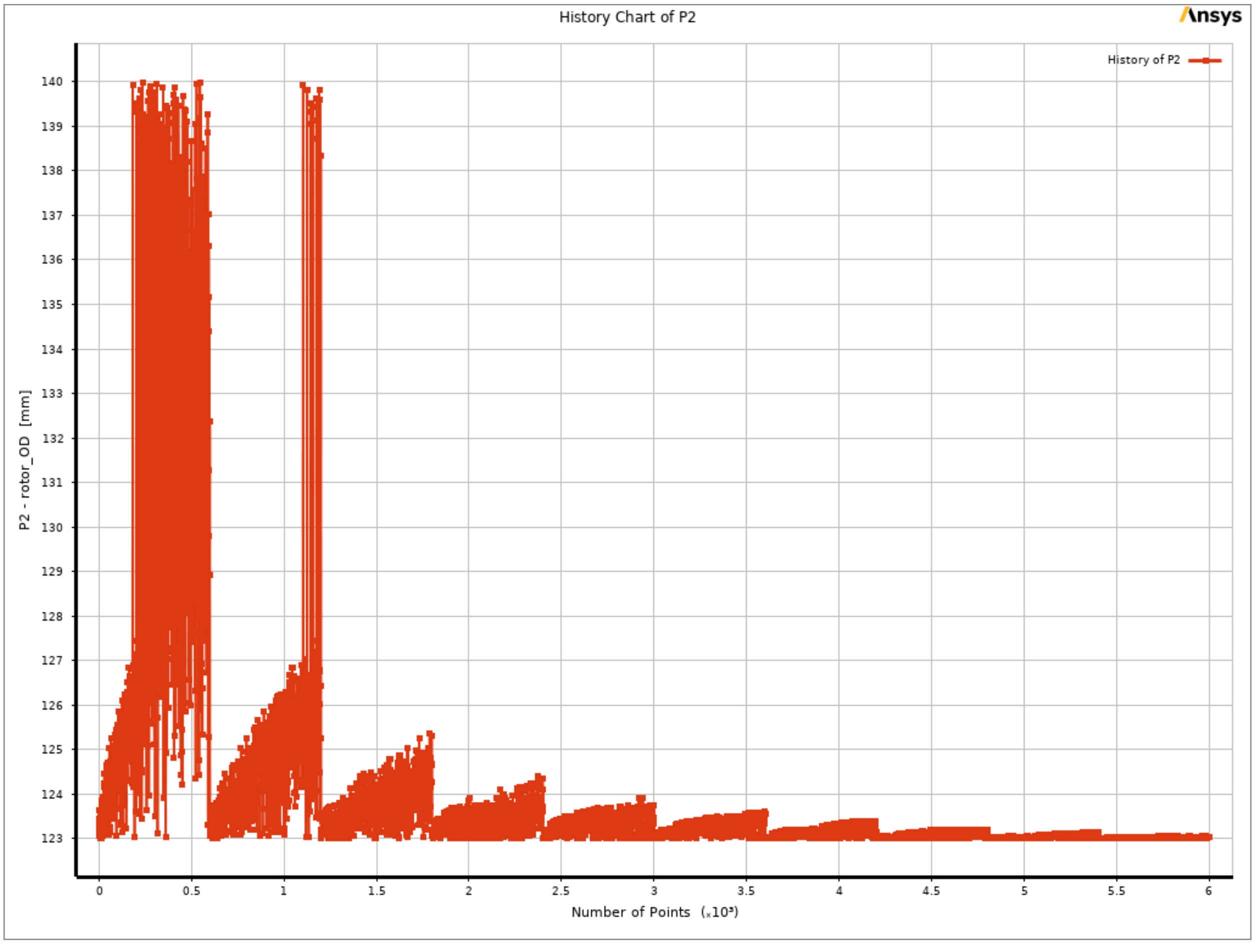
Structural Analysis

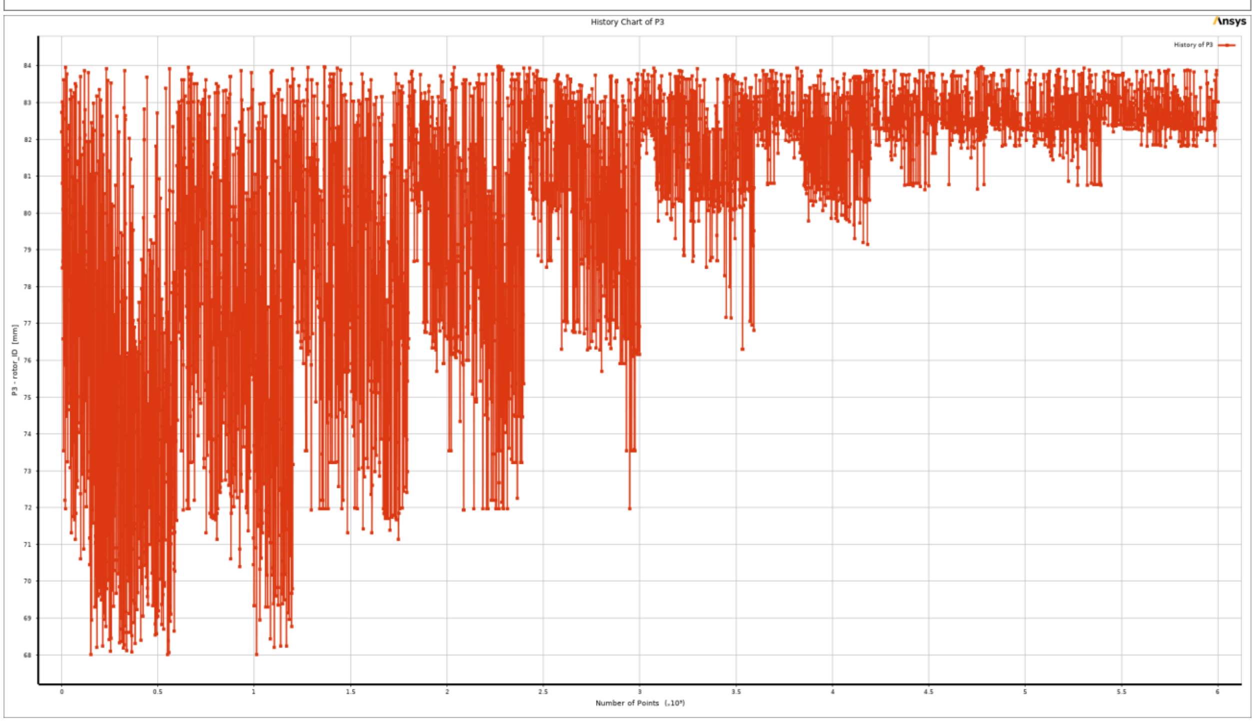
Optimization For the subsystem 1 optimization problem, the upper bound on the stress constraint from Equation 11 is parametrized and the corresponding optimal volume is evaluated using the Interior-Point Method (IPM) in MATLAB. A Pareto curve of optimal volume against the respective stress upper bound is plotted using the IPM and MOGA algorithms as shown in Figure 13. All the MATLAB codes are attached in Appendix B. As observed from the figure, the curves obtained using both the algorithms are in good agreement with each other. Using the Pareto curve, one can easily obtain the optimal volume value for a given stress upper bound constraint.

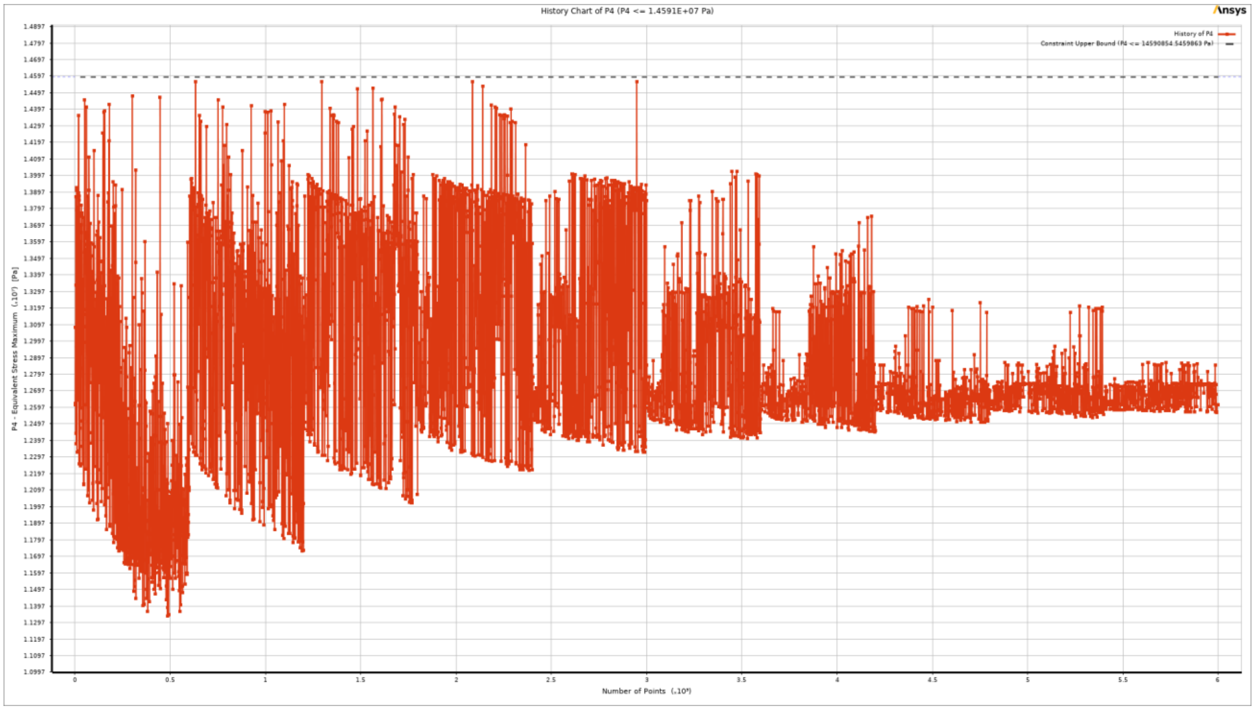
Table of Schematic F4: Optimization

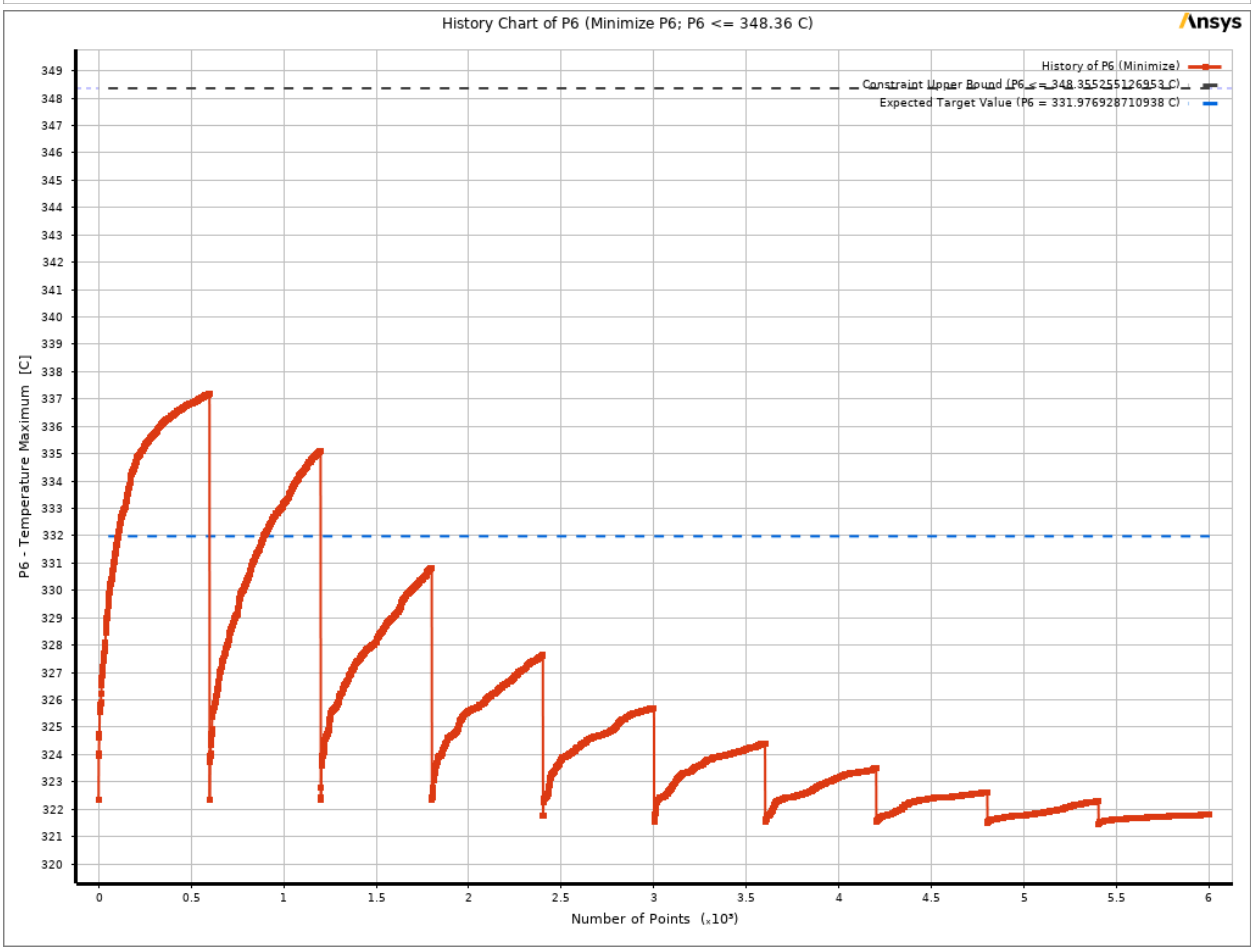
	A	B	C	D
1	⊟ Optimization Study			
2	Minimize P6; P6 <= 348.36 C	Goal, Minimize P6 (Default importance); Strict Constraint, P6 values less than or equals to 348.36 C (Default importance)		
3	P4 <= 1.4591E+07 Pa	Strict Constraint, P4 values less than or equals to 1.4591E+07 Pa (Default importance)		
4	P7 >= 1316.7 Hz	Strict Constraint, P7 values greater than or equals to 1316.7 Hz (Default importance)		
5	⊟ Optimization Method			
6	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.		
7	Configuration	Generate 3000 samples initially, 600 samples per iteration and find 3 candidates in a maximum of 20 iterations.		
8	Status	Converged after 7619 evaluations.		
9	⊟ Candidate Points			
10		Candidate Point 1	Candidate Point 2	Candidate Point 3
11	P1 - rotor_thickness (mm)	27.988	27.991	27.984
12	P2 - rotor_OD (mm)	123.01	123.01	123.01
13	P3 - rotor_ID (mm)	83.323	82.998	82.577
14	P4 - Equivalent Stress Maximum (Pa)	1.2609E+07	1.2638E+07	1.2695E+07
15	P6 - Temperature Maximum (C)	321.46	321.47	321.53
16	P7 - Total Deformation Reported Frequency (Hz)	1524.5	1530.2	1537.7

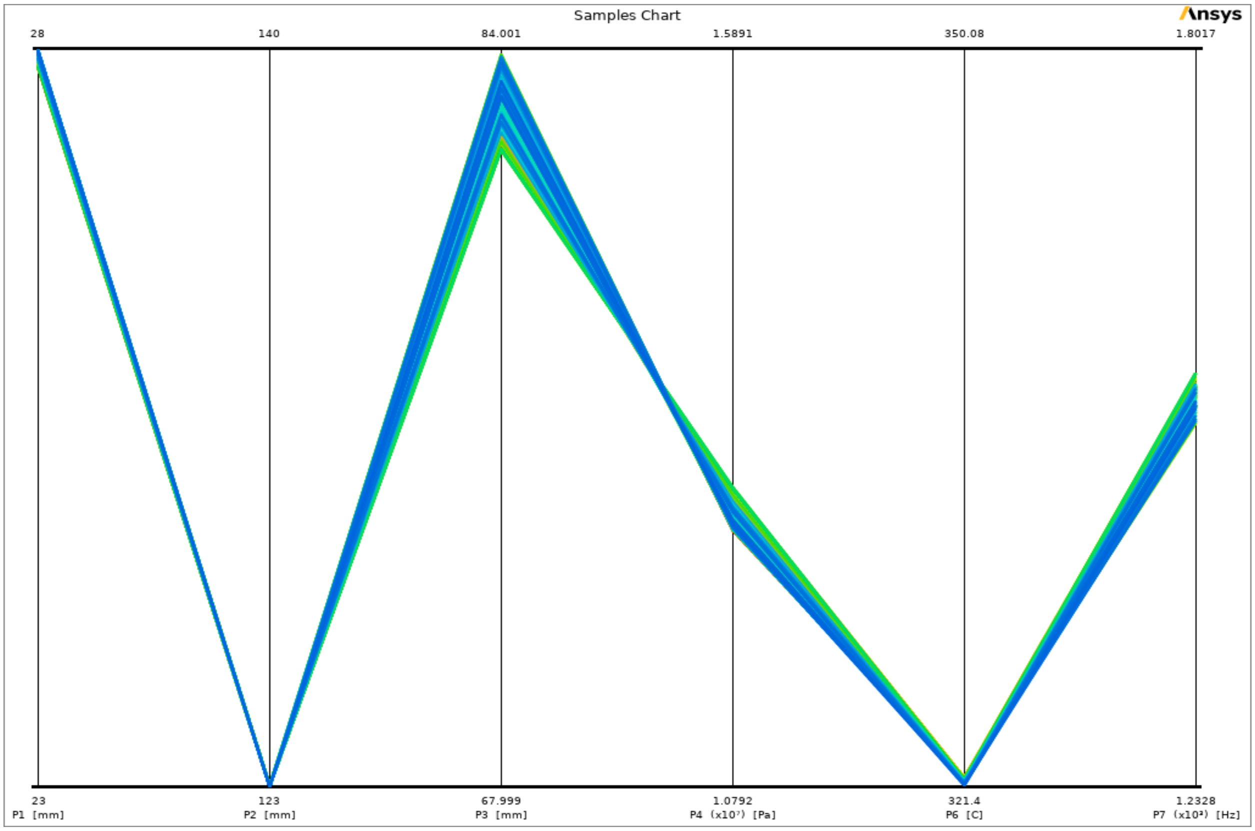


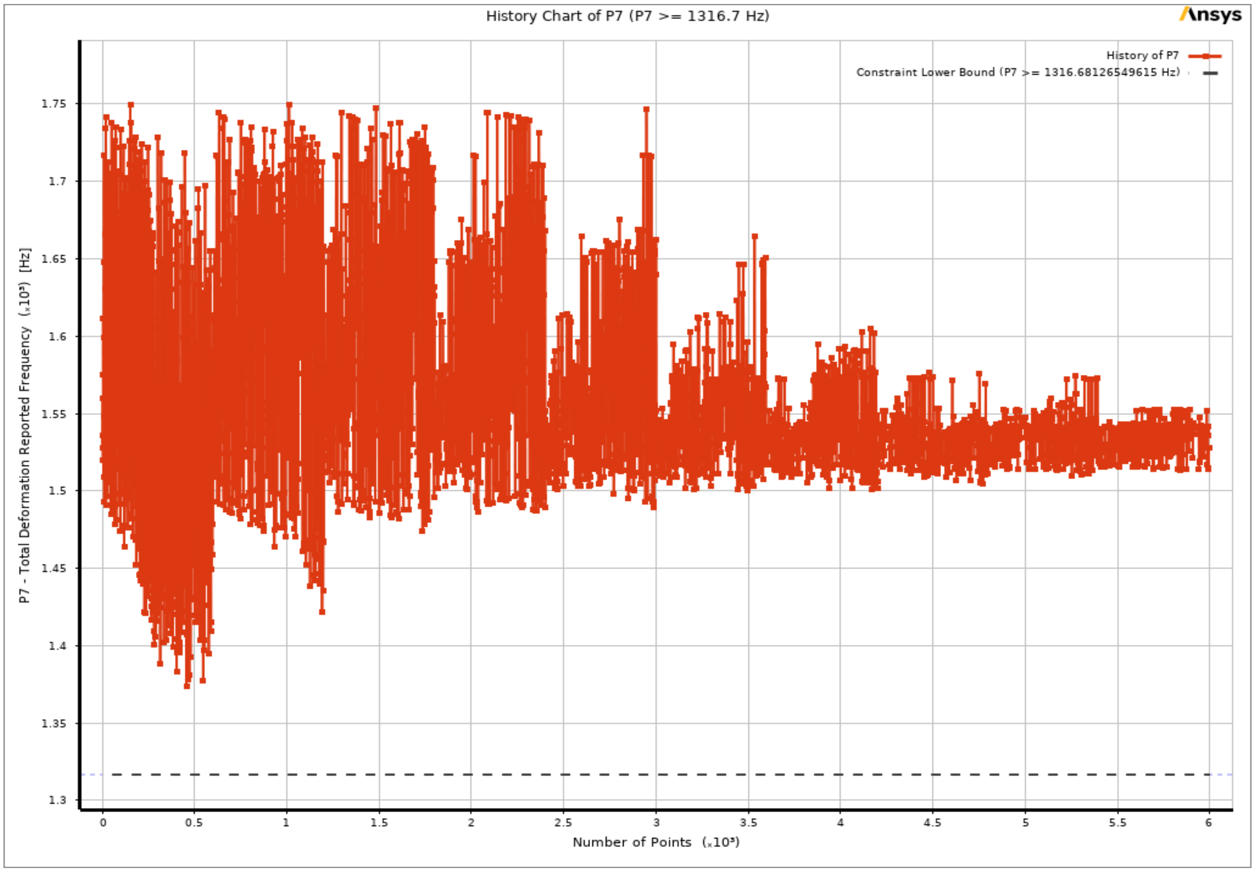

History Chart of P2
Ansys
History of P2
P2 - rotor_OD [mm]
140
139
138
137
136
135
134
133
132
131
130
129
128
127
126
125
124
123
0
0.5
1
1.5
2
2.5
3
3.5
4
4.5
5
5.5
6
Number of Points ($\times10^3$)

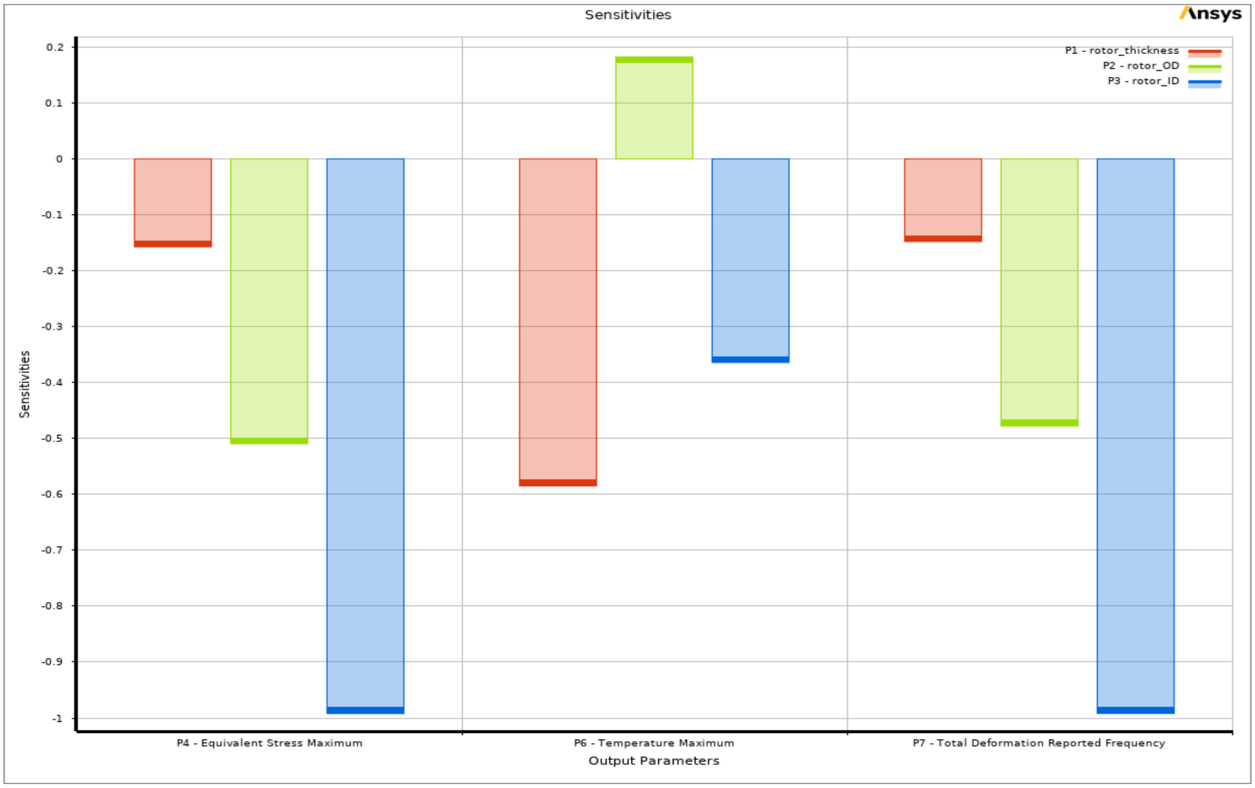

History Chart of P3
Ansys
History of P3
P3 - rotor_ID [mm]
Number of Points ($\times10^3$)

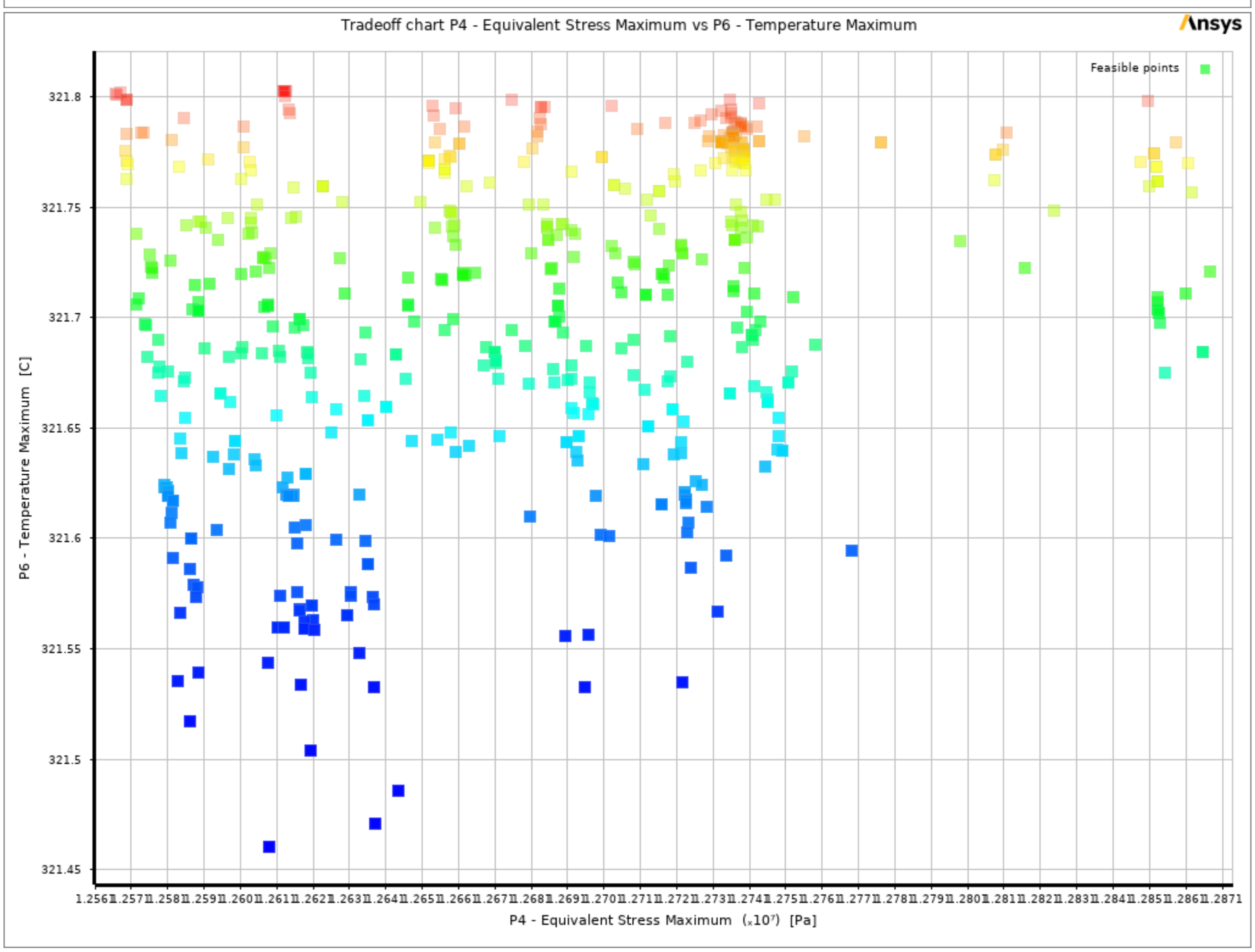

History Chart of P4 (P4 <= 1.4591E+07 Pa)
Ansys
History of P4
Constraint Upper Bound (P4 <= 14590854.5459863 Pa)
P4 - Equivalent Stress Maximum [Pa]
Number of Points


History Chart of P6 (Minimize P6; P6 <= 348.36 C)
Ansys
History of P6 (Minimize)
Constraint Upper Bound (P6 <= 348.355255126953 C)
Expected Target Value (P6 = 331.976928710938 C)
P6 - Temperature Maximum [C]
Number of Points (x10³)


Samples Chart
Ansys
28
140
84.001
1.5891
350.08
1.8017
23
P1 [mm]
123
P2 [mm]
67.999
P3 [mm]
1.0792
P4 (x10⁷) [Pa]
321.4
P6 [C]
1.2328
P7 (x10³) [Hz]


History Chart of P7 (P7 >= 1316.7 Hz)
Ansys
History of P7
Constraint Lower Bound (P7 >= 1316.68126549615 Hz)
P7 - Total Deformation Reported Frequency (x10³) [Hz]
1.75
1.7
1.65
1.6
1.55
1.5
1.45
1.4
1.35
1.3
0
0.5
1
1.5
2
2.5
3
3.5
4
4.5
5
5.5
6
Number of Points (x10³)


Sensitivities
Ansys
P1 - rotor_thickness
P2 - rotor_OD
P3 - rotor_ID
Sensitivities
0.2
0.1
0
-0.1
-0.2
-0.3
-0.4
-0.5
-0.6
-0.7
-0.8
-0.9
-1
P4 - Equivalent Stress Maximum
P6 - Temperature Maximum
P7 - Total Deformation Reported Frequency
Output Parameters


Tradeoff chart P4 - Equivalent Stress Maximum vs P6 - Temperature Maximum
Ansys
Feasible points
P6 - Temperature Maximum [C]
321.8
321.75
321.7
321.65
321.6
321.55
321.5
321.45
P4 - Equivalent Stress Maximum (x10⁷) [Pa]

System Integration Study

After performing the individual subsystem level optimization, an attempt is made to integrate all the subsystems. There are four objective functions in this problem. As it is not possible to plot 4D Pareto curves, three objective functions are considered at a time so that 3D Pareto curves can be easily plotted from them in MATLAB. This study helps us understand the relationship between three objective functions at a time.

Combined Modal & Thermal Optimization

The mathematical model for this system level optimization is as follows: M in : f1, −f3, f4 s.t. g1, g2, g3, g4, g5, g6, g8 and g9 Optimization is performed in MATLAB using the single objective IPM (fmincon) method. This algorithm being used for single objective optimization problem, the volume function is considered to be the primary objective and the frequency and temperature objective functions are considered as constraints. The lower and upper bounds on frequency and temperature respectively are varied and a Pareto curve for volume, frequency and temperature is obtained as shown in Figure 19. The optimal volume value for 1200 Hz lower bound on frequency and 400oC upper bound on temperature is obtained from this Pareto curve and is correlated with a MOGA value obtained from ANSYS.

References

Y. Ren, MAE 598 - Design Optimization Notes and ANSYS DOE tutorial, 2021.