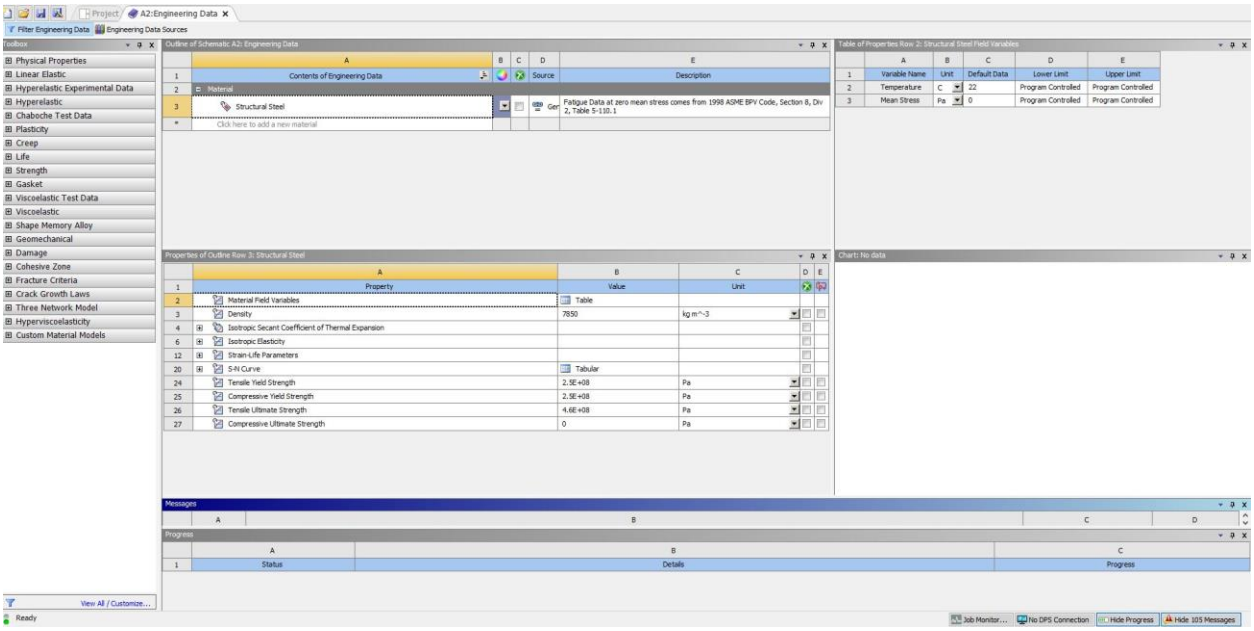
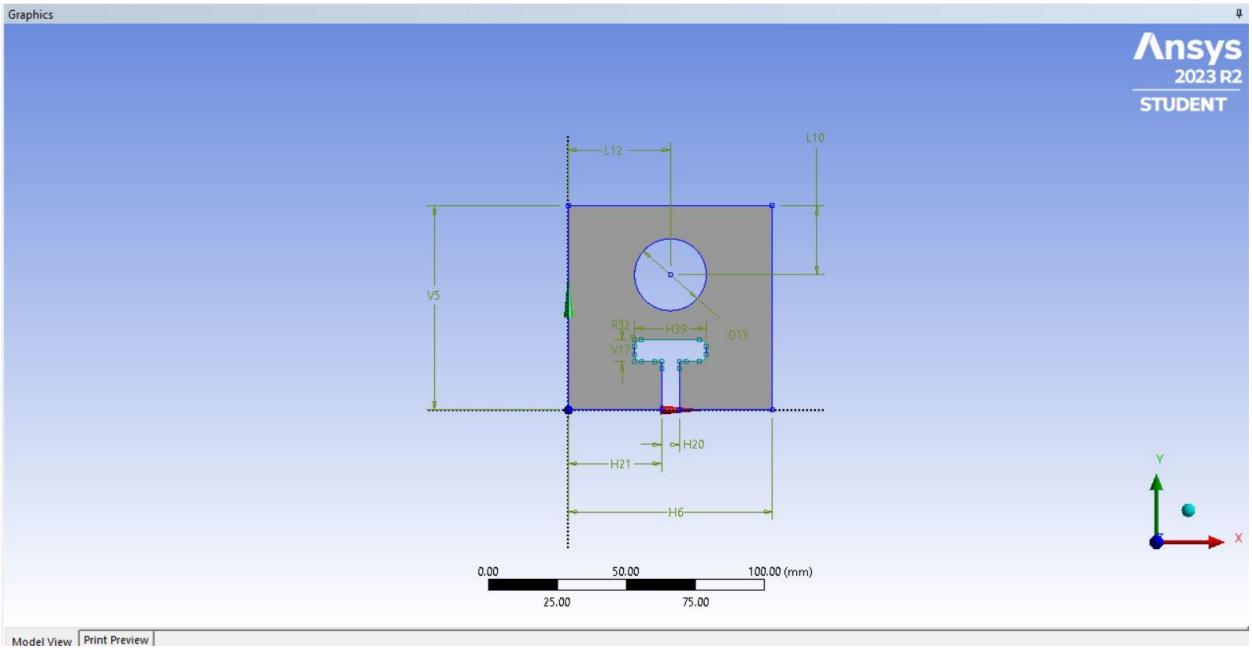
Ansys Design

I created a structural analysis project in Ansys and designed a plate which will be constrained on one side and will have a load acting on one of the surfaces in the downward direction. The load will be acting on a slot in my part.

1. For the engineering data I choose the default structural steel properties.

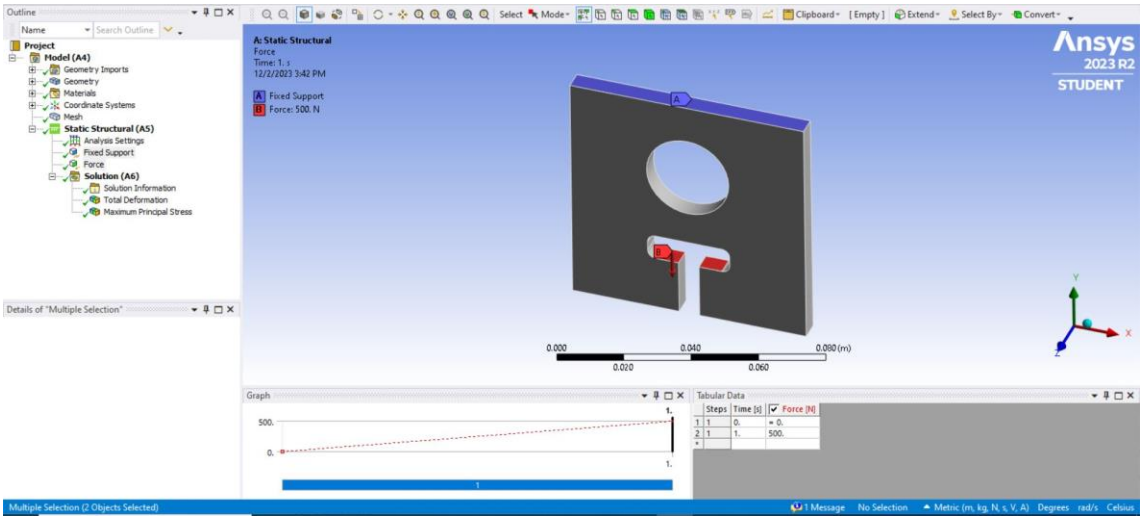


2. I then modeled my part in design modeler and create a rectangular part with fastener slot in between along which load will be acting.



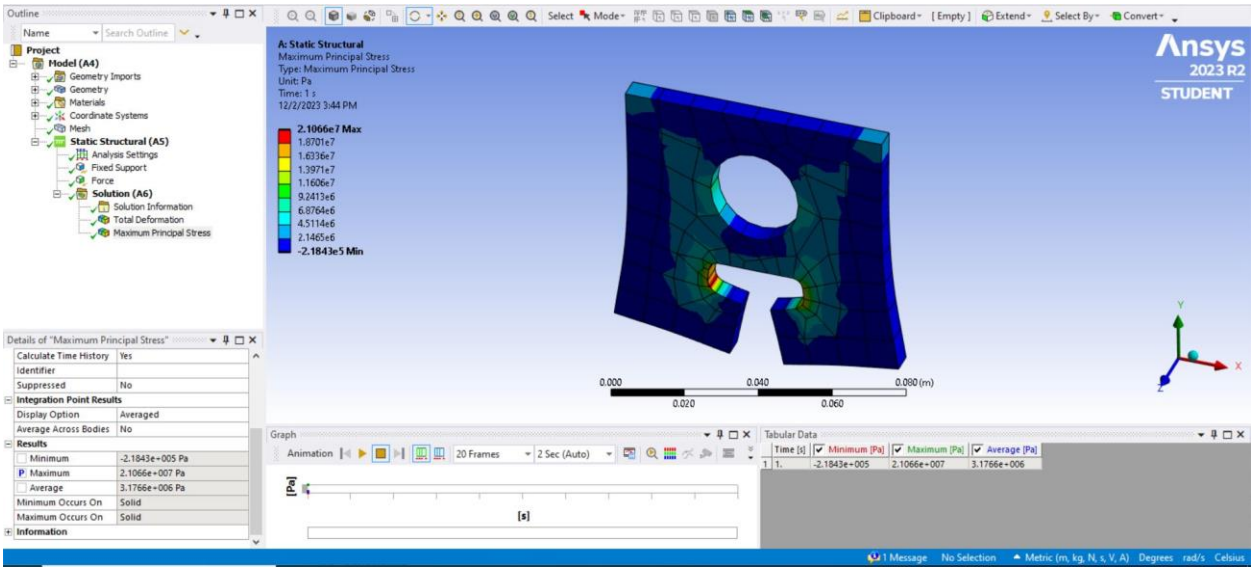
In this design, the radius of the corner, the height and width of the slot were set as modifiable parameters. These will get tuned by the DOE to get the best possible combination.

3. After creating the CAD model, I needed to constrain my geometry and also mesh the part and run my results.



From the image it can be seen that I add a load of 500 N in the downward direction of my slot and I constrain the top surface of my part as a fixed support.

I choose the maximum principal stress as my output parameter. Which can be seen below.

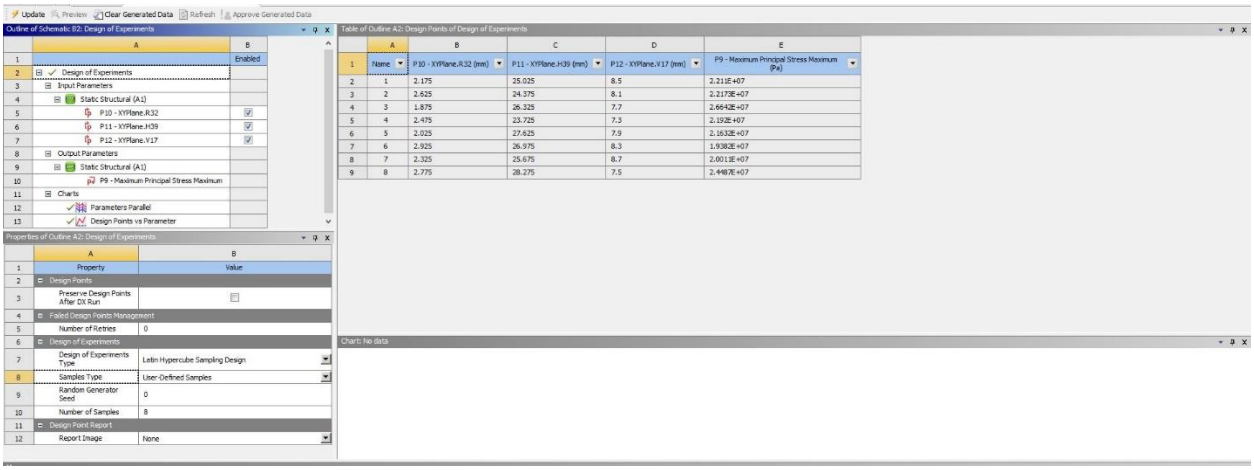


4. Following are the list of the parameters chosen

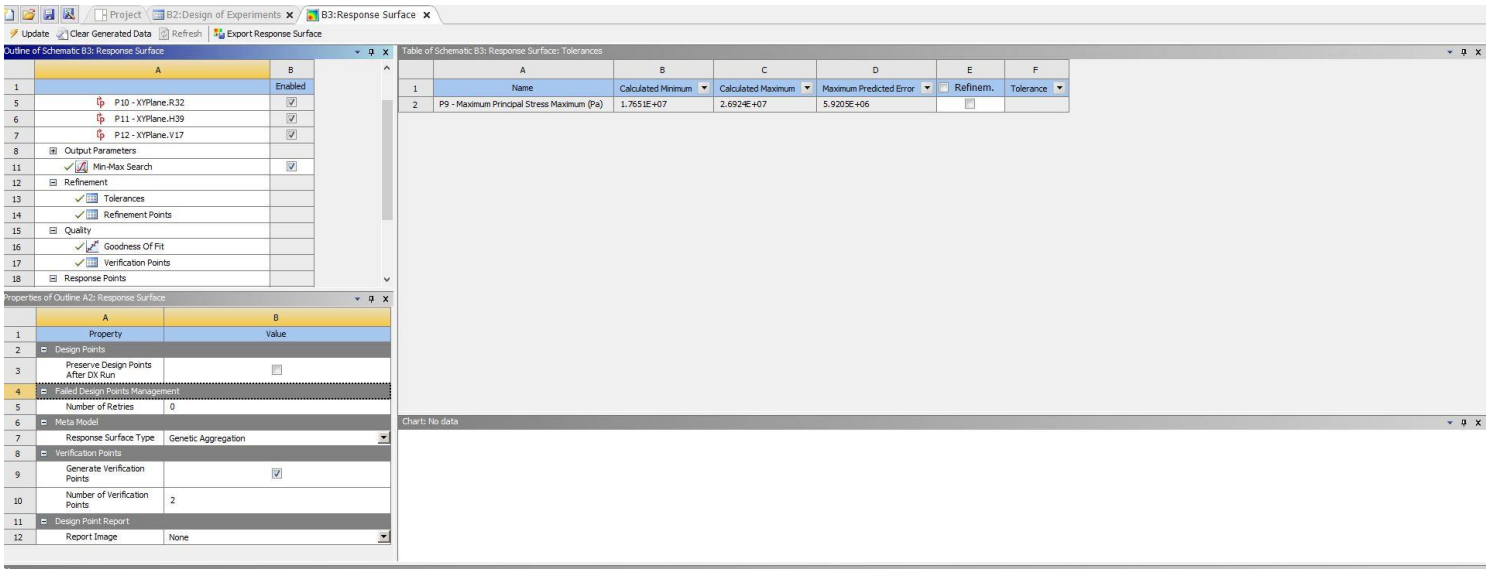
Outline of Schematic A8: Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	Static Structural (A1)			
4	P10	XYPlane.R32	2.5	mm
5	P11	XYPlane.H39	26	mm
6	P12	XYPlane.V17	8	mm
*	New input parameter	New name	New expression	
8	Output Parameters			
9	Static Structural (A1)			
10	P9	Maximum Principal Stress Maximum	2.1066E+07	Pa
*	New output parameter		New expression	
12	Charts			

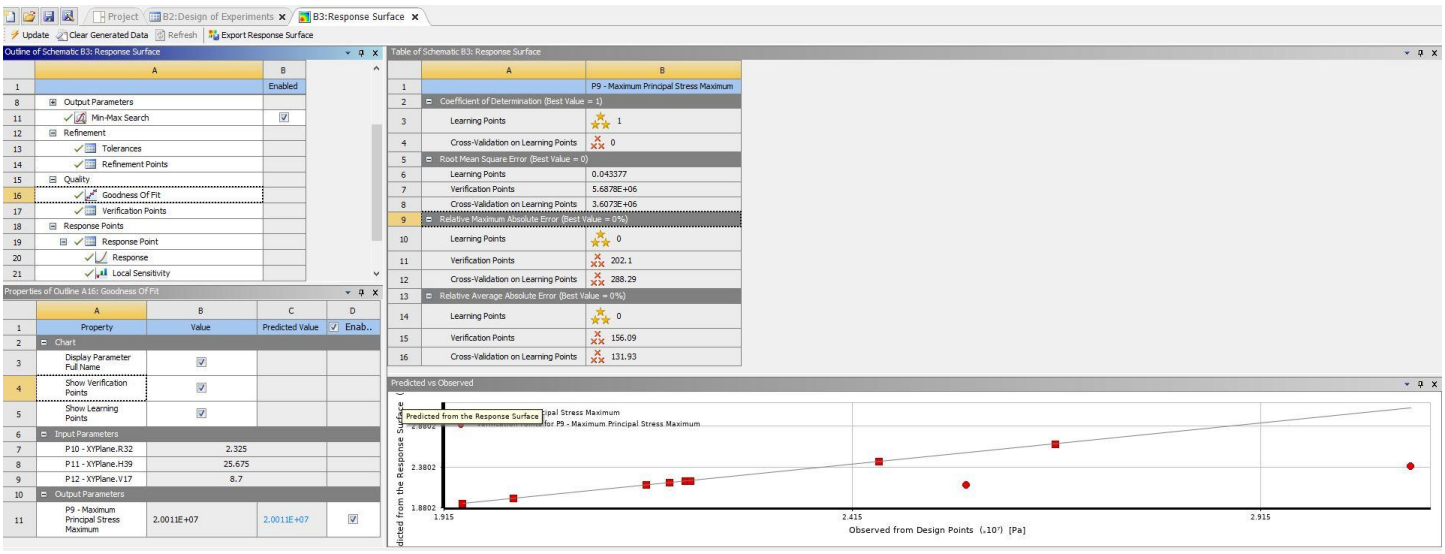
5. We now run the DOE analysis: We use Latin Hyper cube sampling design and select the user defined number of samples where I choose 8 samples.



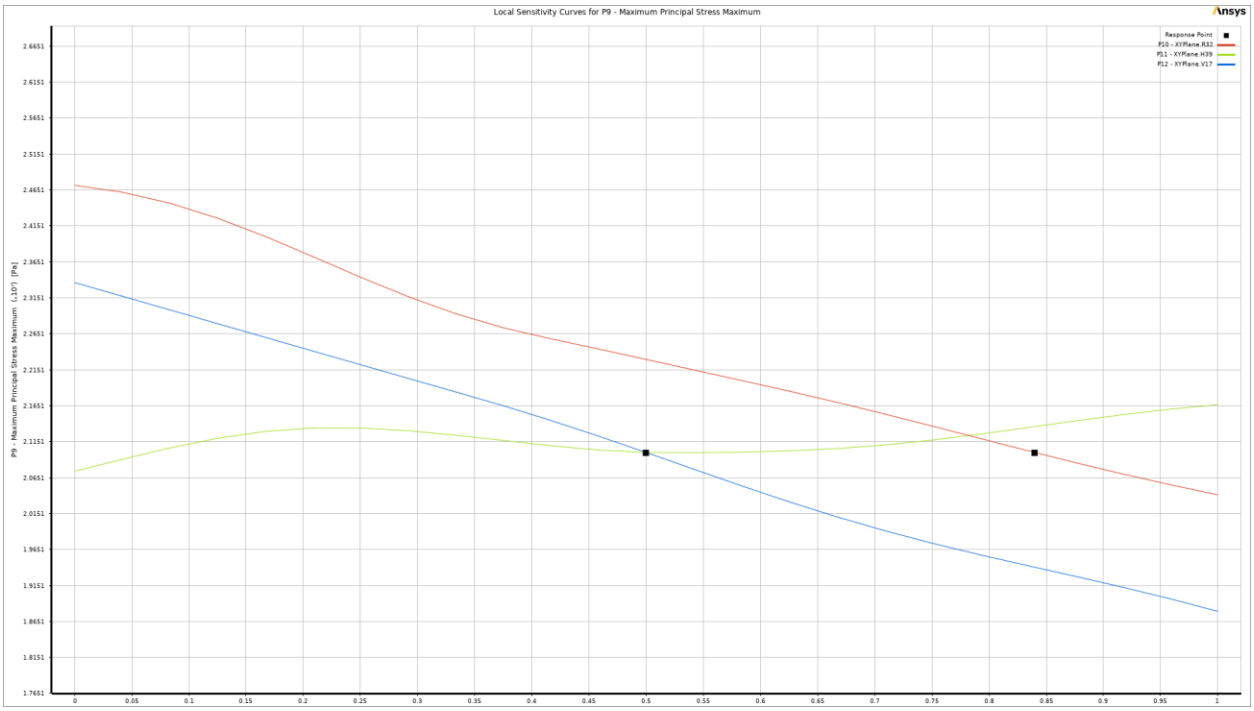
6. We next run the response surface analysis and we select the genetic aggregation from the response surface type. We also generate 2 verification points

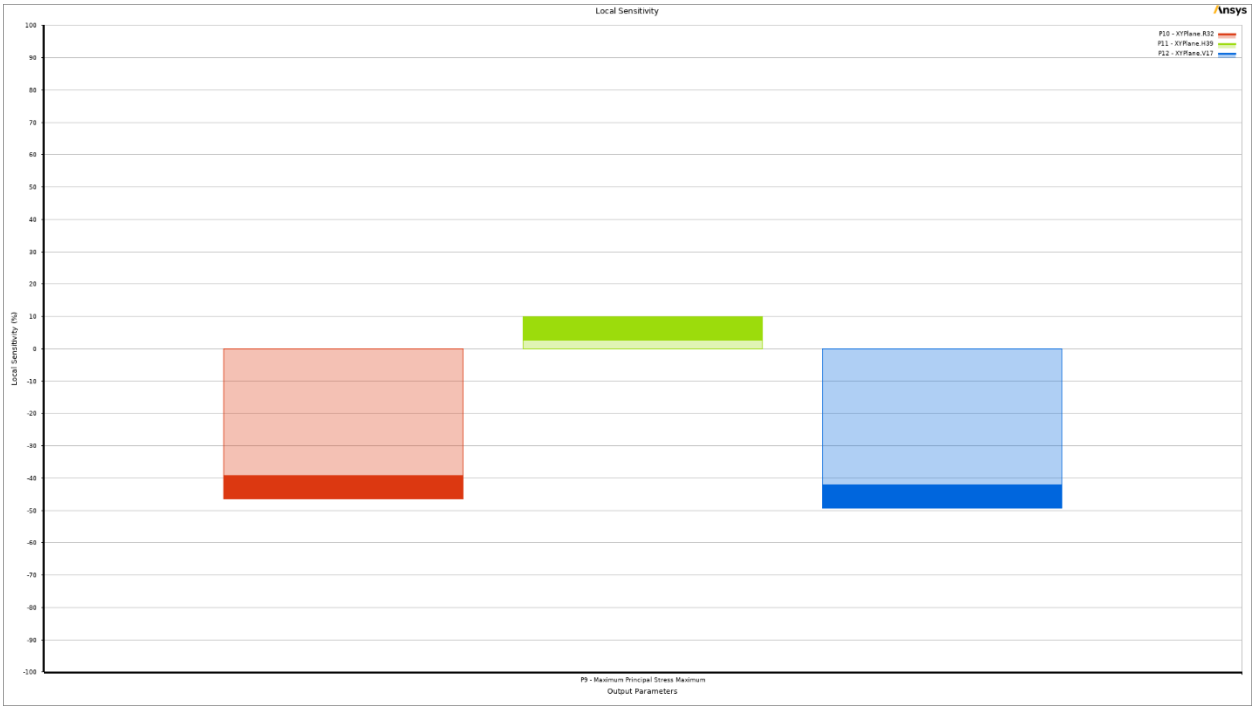


7. We check the verification points that are shown in the response surface analysis and we can see the verification points prediction are close to the model.



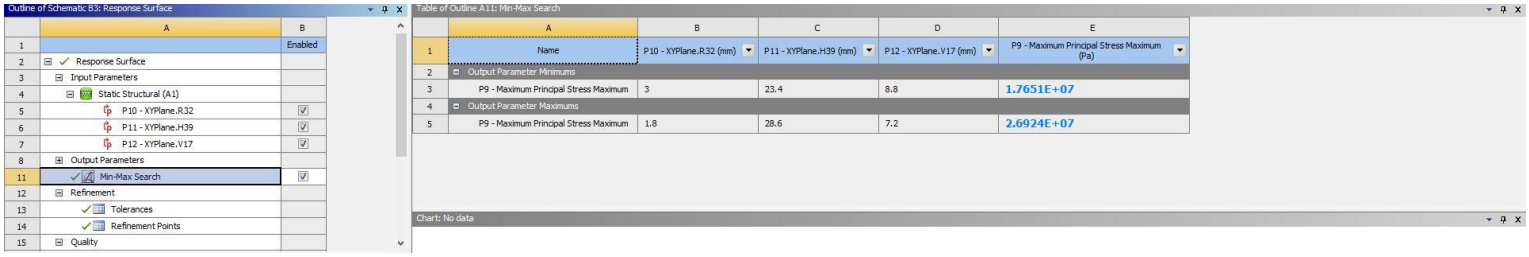
We next check the sensitivity analysis of the design parameters:





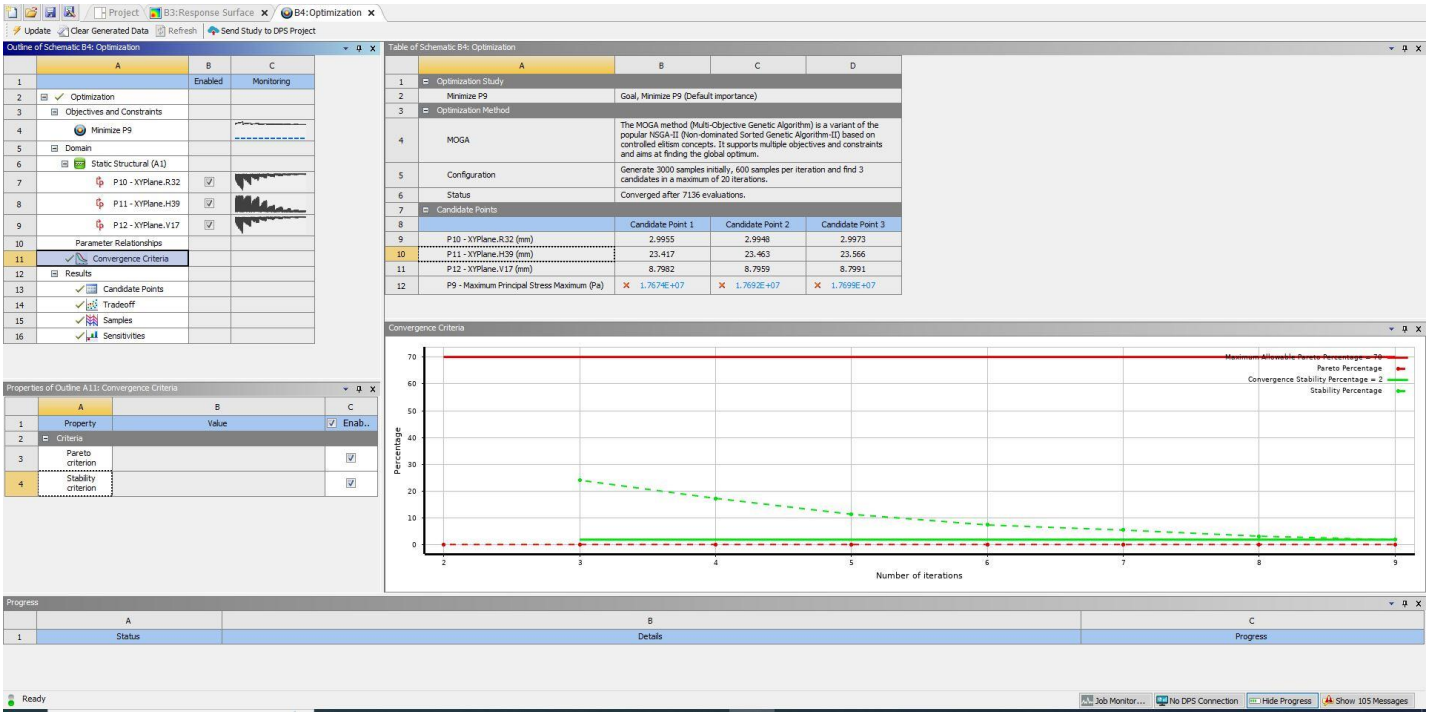
We can see that the radius parameter and the vertical height parameter play a larger role in affecting the maximum principal stress.

We can also see the maximum and minimum values that can be achieved by varying the parameters.

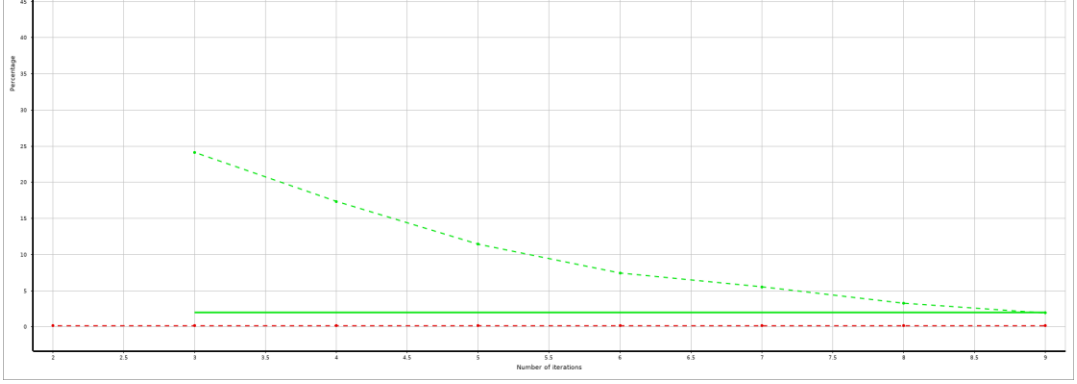


8. We finally conducted the optimization study: We choose the output parameter of maximum principal stress as the objective function that needs to be minimized.

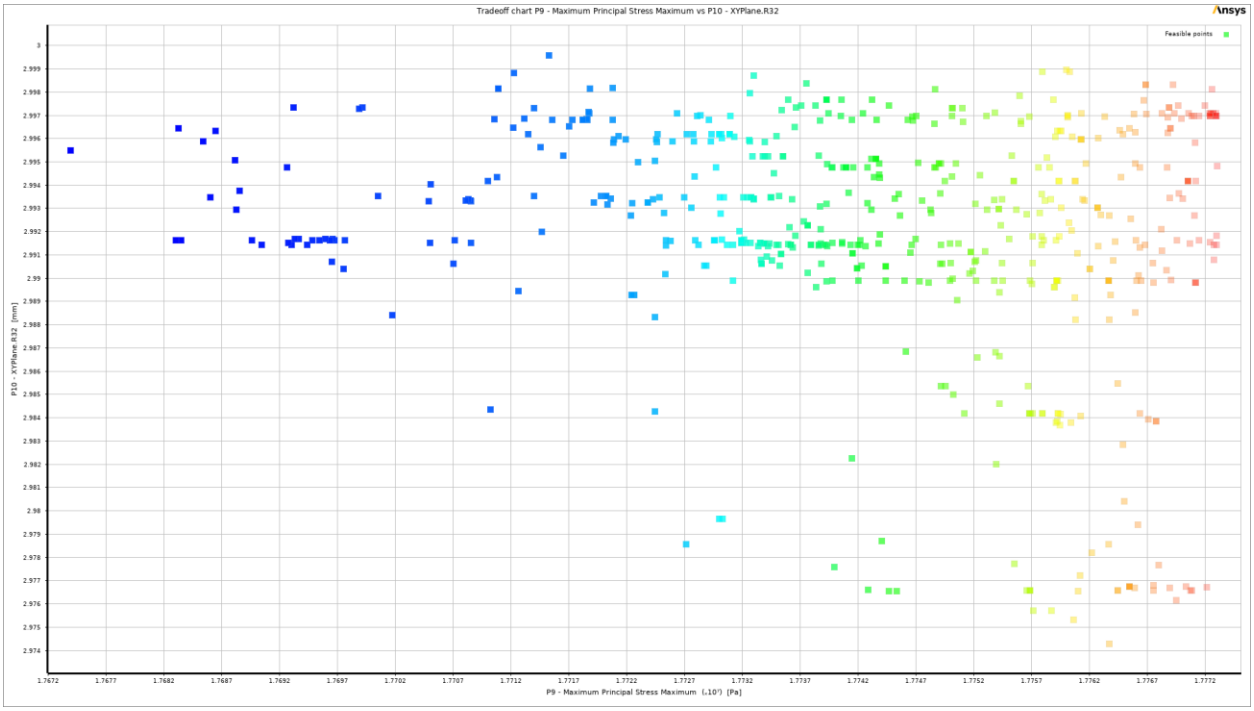
For the optimization method we use MOGA- which is expanded as the multi-objective genetic algorithm



We validate the optimal solution by looking at the convergence criteria, since we have convergence for this set of parameters



We also have the tradeoff analysis for the radius parameter that is shown.



Conclusion:

1. We see that for the parameters: Radius, Height and Length we have the following values that are optimum
    a. Radius: **3 mm**
    b. Height: **8.8 mm**
    c. Length: **23.4 mm**
2. Having these optimum values we can minimize the maximum principal stress to **1.765 x 10^7 Pa**