

Plate With a Hole Optimization - Optimization

Author: Sebastien Lachance-Barrett, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up
 2. Initial Solution
 3. Input & Output Parameters
 4. Design of Experiments
 5. Response Surface
 6. Optimization
 7. Verification & Validation
- Exercises
Comments

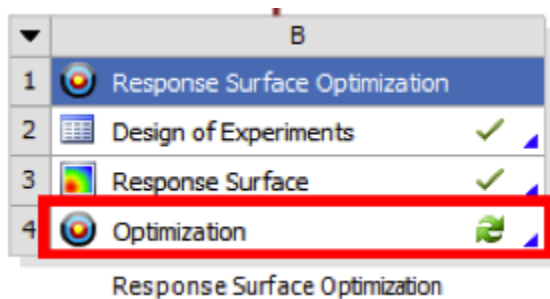


This tutorial was created with an older version of ANSYS (14.5), where the mesh generator and the refinement process was not as strong as it is now. This will result in a different solution than the one shown. (In ANSYS 16.1, the optimization results in a radius of ~1.25in-1.27in and in ANSYS 2019 R2 the optimization results in a radius of ~1.11-1.13 in)

Optimization

Set-Up of Optimization

Begin this step, by double clicking on **Optimization**.



At this point, ANSYS must be told that the objective function(volume) is to be minimized while staying below the 32.5 ksi Von Mises stress threshold. First, select "Objectives and Constraints" in the outline window. Then, in the "Table of Schematic B4: Optimization" window, select the parameter to be **P2-Surface Body Volume** and change the objective type to **Minimize**. Next, add in a second parameter which will be **P3-Equivalent (von Mises) Stress Maximum**, change the constraint type to **Values <= Upper Bound** and enter 32500 for the **Upper Bound**. Your table should now look like the one below.

Table of Schematic B4: Optimization						
	A	B	C		E	F
1	Name	Parameter	Objective		Constraint	
2			Type	Target	Type	Lower Bound
3	Minimize P2	P2 - Surface Body Volume	Minimize		No Constraint	
4	P3 <= 32500 psi	P3 - Equivalent (von-Mises) Stress Maximum	No Objective		Values <= Upper Bound	32500
*		Select a Parameter				

Now, execute the optimization by clicking on **Update** and click on Optimization from the outline window to view the results. The optimization should yield similar results to the following table.

Table of Schematic B4: Optimization				
	A	B	C	D
1	[-] Optimization Study			
2	Minimize P2	Goal, Minimize P2 (Default importance)		
3	P3 <= 32500 psi	Strict Constraint, P3 values less than or equals to 32500 psi (Default importance)		
4	[+] Optimization Method			
8	[-] Candidate Points			
9		Candidate Point 1	Candidate Point 2	Candidate Point 3
10	P1 - DS_R	1.5408	1.4853	1.4298
11	P2 - Surface Body Volume (in^3)	✗ 9.8136	✗✗ 9.8267	✗✗ 9.8395
12	P3 - Equivalent (von-Mises) Stress Maximum (psi)	★ ★ ★ 32500	★ ★ ★ 32420	★ ★ ★ 32342

The optimization tool found three candidate points that matched our given constraints and objectives. This computation was pretty fast because the optimization tool used the response surface model (plots) that previously generated. It did not actually solve our model by doing a matrix inversion. Remember that the response surface model is only an approximation of the relationship between the parameters and so our results might not be very accurate. Thankfully, we can solve our model using these candidate points to “verify” that they really do satisfy our constraints.

In the Properties of Schematic B4: Optimization window, insert a check to **Verify Candidate Points** and click on **Update** once again. Notice how much longer it takes to solve our model.

Properties of Schematic B4: Optimization		
	A	B
1	Property	Value
2	Design Points	
3	Preserve Design Points After DX Run	<input type="checkbox"/>
4	Failed Design Points Management	
5	Number of Retries	0
6	Optimization	
7	Optimization Method	Screening
8	Number of Samples	1000
9	Maximum Number of Candidates	3
10	Verify Candidate Points	<input checked="" type="checkbox"/>
11	Optimization Status	
12	Number of Evaluations	1002
13	Number of Failures	0
14	Size of Generated Sample Set	1002
15	Number of Candidates	3

The optimization should yield similar results to the following table. Surprise! Some candidate points do not satisfy the maximum Von Mises stress constraint (now marked with a red cross). This is why it is important to always verify the candidate points.

Note: In version 2019 R2, we found that all three of the candidate points failed at this step, so don't be concerned if that is what you find.

Table of Schematic B4: Optimization							
	A	B	C	D	E	F	G
1	Optimization Study						
2	Minimize P2	Goal, Minimize P2 (Default importance)					
3	P3 <= 32500 psi	Strict Constraint, P3 values less than or equals to 32500 psi (Default importance)					
4	Optimization Method						
8	Candidate Points						
9		Candidate Point 1	Candidate Point 1 (verified)	Candidate Point 2	Candidate Point 2 (verified)	Candidate Point 3	Candidate Point 3 (verified)
10	P1 - DS_R	1.5408		1.4853		1.4298	
11	P2 - Surface Body Volume (in^3)	✗ 9.8136	✗ 9.8136	✗✗ 9.8267	✗✗ 9.8267	✗✗ 9.8395	✗✗ 9.8395
12	P3 - Equivalent (von-Mises) Stress Maximum (psi)	☆☆ 32500	✗ 32509	☆☆ 32420	☆☆ 32495	☆☆ 32342	☆☆ 32345

By selecting **candidate points** under the results section of the Outline of Schematic B4: Optimization window, you can also see how the results of each candidate points differ from the results of a specified reference candidate point. Additionally, you can even add new candidate points.

Output parameter values calculated from simulations (design point updates) are displayed in black text, while output parameter values calculated from a response surface are displayed in blue.

The number of gold stars or red crosses displayed next to each goal-driven parameter indicate how well the parameter meets the stated goal, from three red crosses (the worst) to three gold stars (the best).

Tip: Remember how we specified the radius to range from 1 to 2.5 inches to create the Response Surface? Well we now know that the optimized radius should be around 1.45 inches so no need to have that big of a range anymore. For a second round of optimization (not done in this tutorial), it would be a good idea to go back in Design of Experiments and change the lower and upper bounds to be, say 1.4 and 1.5 inches respectively. A smaller range will give you a more accurate response surface which will help you optimize the radius further.

Obtaining Deformation and Stress Results for Selected Design Point

We will select candidate point 2 as the design point. It is a good idea to review the deformation and stress plots at the chosen design point. To do this, let's set the radius from **Candidate Point 2** as the radius of the hole in **Design Modeler**. Select **(Right Click) Candidate Point 2 > Insert as Design Point**.

4		Candidate Point 1 (verified)		✗ 9.8136	-0.13 %
5		Candidate Point 2	1.4853	✗✗ 9.8267	0.00 %
6		Candidate Point 2 (verified)			
7		Candidate Point 3			
8		Candidate Point 3 (verified)			
*		New Custom Candidate Point			

Copy

Explore Response Surface at Point

Insert as Design Point

Insert as Refinement Point

Insert as Verification Point

Insert as Custom Candidate Point

Export Data

Next, click **Return to Project** and double click on **Parameter Set**. Selecting **Insert as Design Point** created the design point DP1. Now in the "Table of Design Points" **(Right Click) Current > Duplicate design point**.

Table of Design Points				
	A	B	C	
1	Name	P1 - DS_R	P2 - Surface Body Volume	P3 - Equ
2	Units		in^3	
3	Current	2		
4	DP 1	1.4853		
*				

Copy

Paste

Set Update Order by Row

Show Update Order

Optimize Update Order

Duplicate Design Point

Export Design Point as Project

Update Selected Design Points

Copy Design Points to

You have just duplicated the parameters from the original geometry into the new design point DP2. Now **(Right Click) DP1 > Copy Inputs to Current** and click on **Update All Design Points** in the toolbar.

Table of Design Points			
	A	B	C
1	Name ▾	P1 - DS_R ▾	P2 - Surface Body Volum
2	Units		in^3
3	Current	2	9.6858
4	DP 1	1.4853	⚡
5	DP 2		
*			

Copy

Set Update Order by Row

Show Update Order

Optimize Update Order

Delete Design Point

Copy inputs to Current

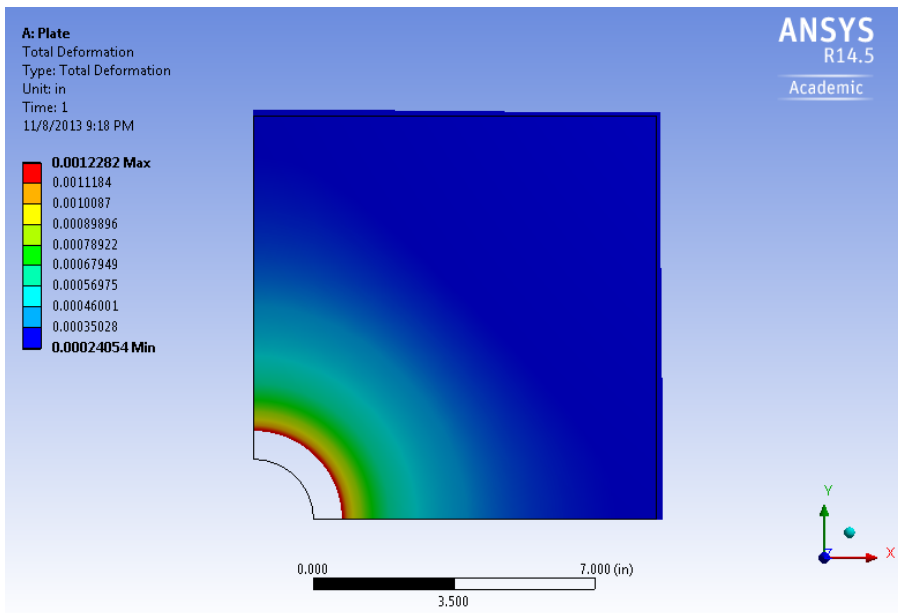
Duplicate Design Point

Update Selected Design Points

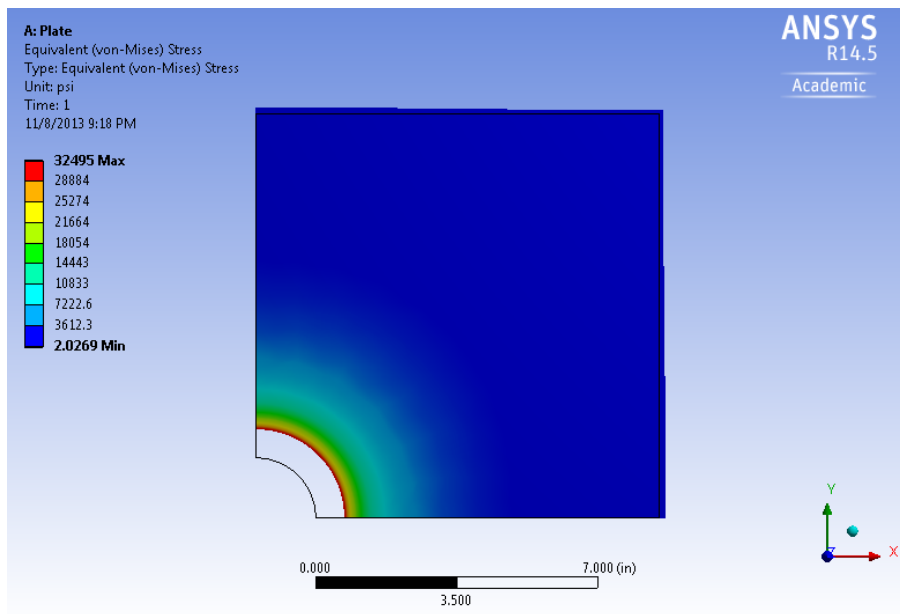
Copy Design Points to

The radius of **Candidate point 2** has been inserted as the radius in the **Design Modeler**. Let's now view the results of our model with our optimized radius of 1.4853. Click on **Return to Project** and double click **Results**. The graphs below display the total deformation and the equivalent Von Mises stress. You should realize that we did a fantastic job with this optimization problem! It does not get much better than this as the equivalent Von Mises stress lies just under our constraint. Well...not really. Let's take a look at the verification and validation step.

Total Deformation



Equivalent Von Mises Stress



[Go to Step 7: Verification & Validation](#)

[Go to all ANSYS Learning Modules](#)