Hopp til innhold

- EDRMedeso portal
- Blogg
- Nyhetsbrev
- Ledige stillinger

Søk

- Forsiden
- Produkter
- Kurs og seminarer
- Konsulenttjenester
- Referanser
- Om EDRMedeso
- Support
- Kontakt oss

Forsiden / Blogg / ANSYS-bloggen / ANSYS Tutorial: Design Optimization in DX

+47 67 57 21 00

Kontaktskjema

ANSYS Tutorial: Design Optimization in DX

ANSYS Workbench Tutorial: Want to know more about you model? Run a What if scenario? How sensitive are you results for changes in the input variables? Maybe optimize your model in terms of weight? Then DesignXplorer is the tool for you.

This article will first give you a short summation of what DesignXplorer is and then show how to set up a simple optimization.

What is DesignXplorer?

Design Explorer is based on a method called Design of Experiments (DOE). This together with various optimization methods helps the program to develop an optimized structure based on selected input and output parameters. Input parameters can either come from DesignModeler or from various CAD systems. These parameters can be in terms of thickness, length, etc. They can also come from Mechanical in terms of forces, material properties, etc. The output parameters are calculated in Mechanical and can for example be in terms of total mass, stress or frequency response. After setting up an analysis with a number of input parameters and out parameters there are the steps that can be run within DesignXplorer:

- Design of Experiments
- Response surface
- Optimization
- Six Sigma Analysis

Design of Experiments

Design of experiments is the foundation that everything within DesignXplorer is built on. What this technique is about is to determine how many and for what input values the analysis shall be run. There are various techniques for this but the same goal for all is to get as good response surface as possible with as few input combinations as possible. So basically this step defines is how many analysis that will be run. Each combination that ANSYS solves for is referred to as a Design Points.

Response Surface

When the Design of Experiments is run the next step is to create a response surface based on these results. A response surface will be created for each output parameter. The response surface is basically created via curve fit through the Design Points. From this response surface you can then investigate output results for input variable combinations that hasn´t been solved for.

Goal Driven Optimization

To help you to select the combination of input variables that satisfies your goals best, you can run a Goal Driven Optimization. Here you have the possibility to give all you parameters different objective functions on which you also can give different importance. ANSYS will then give you a number of candidates that satisfies your goals in the best way.

Six Sigma Analysis

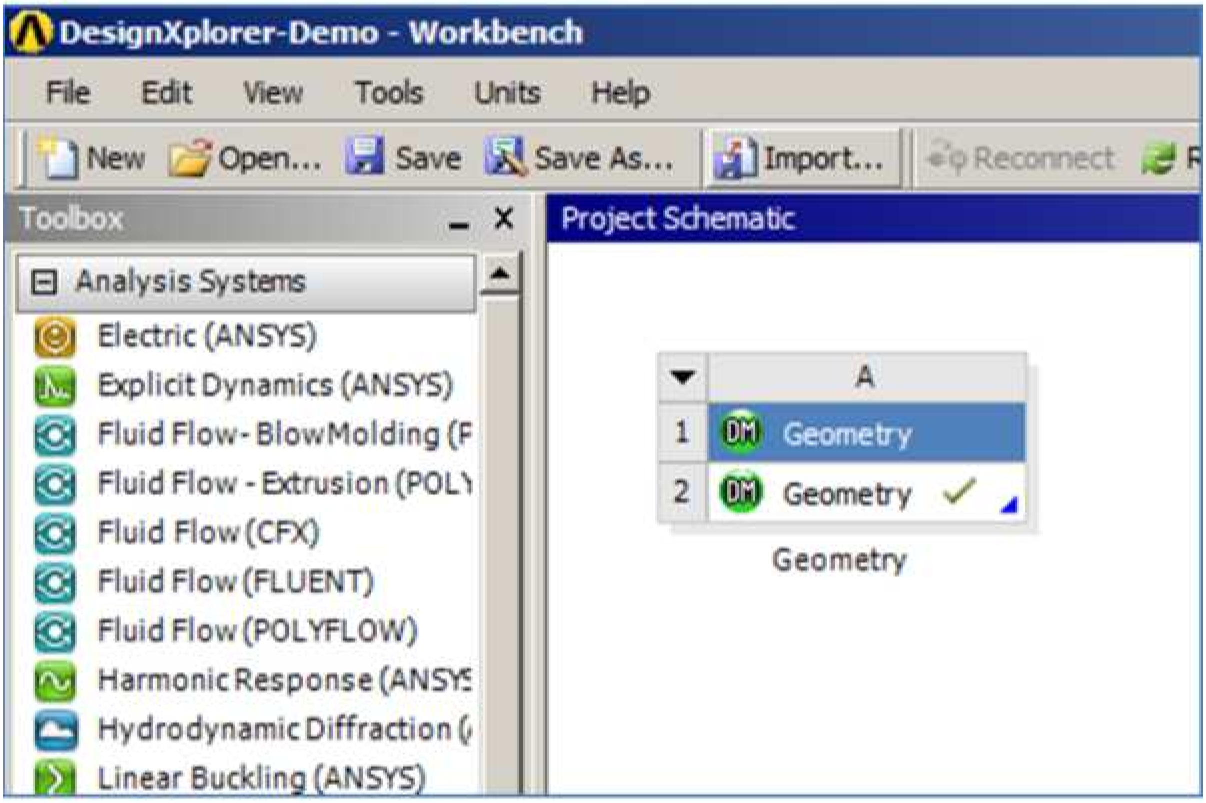
The Six Sigma Analysis is a method that includes both a Design of Experiment and a Response Analysis. What resides is that you also have the possibility to set the input parameters as uncertainties parameters. This means that you can see how uncertainties in the input variables will affect the results for the output parameters.

How to run an Optimization in DesignXplorer.

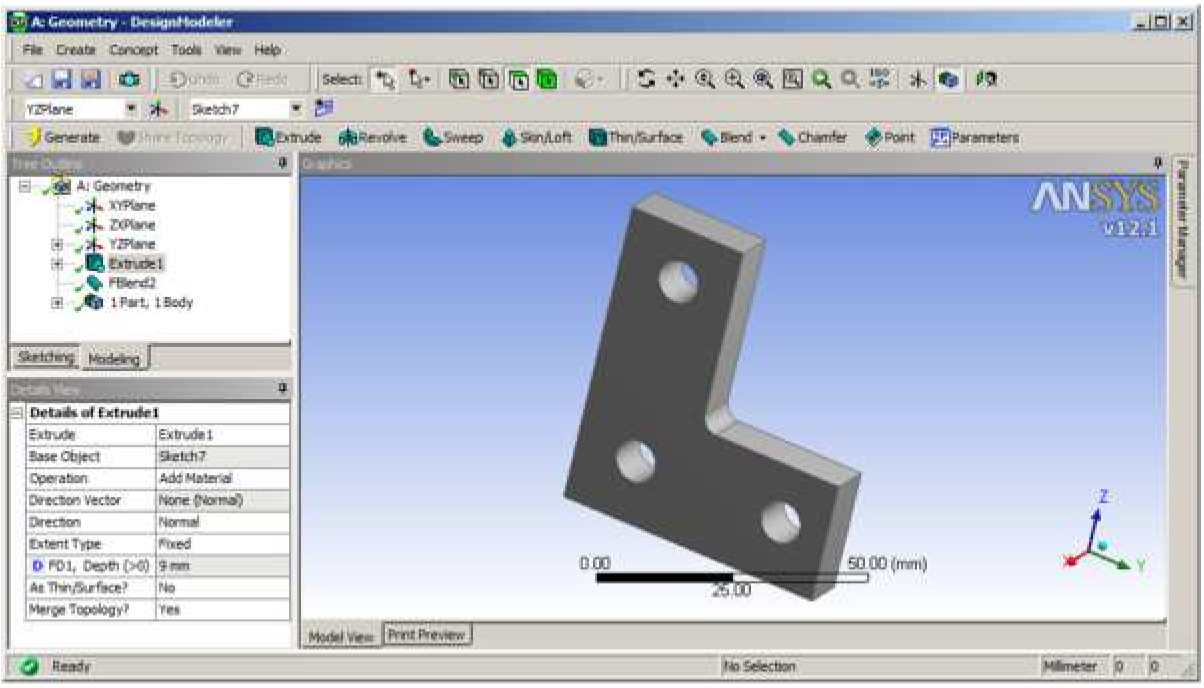
In the tutorial below it will be shown how to set up an optimization analysis in DesignXplorer. We will use two input variables, a thickness and a radius, and two output variables, total mass and Principal Stress. The goal is to find the best combination between thickness and radius to minimize the mass of the structure and to ensure that the stress will be below a defined value.

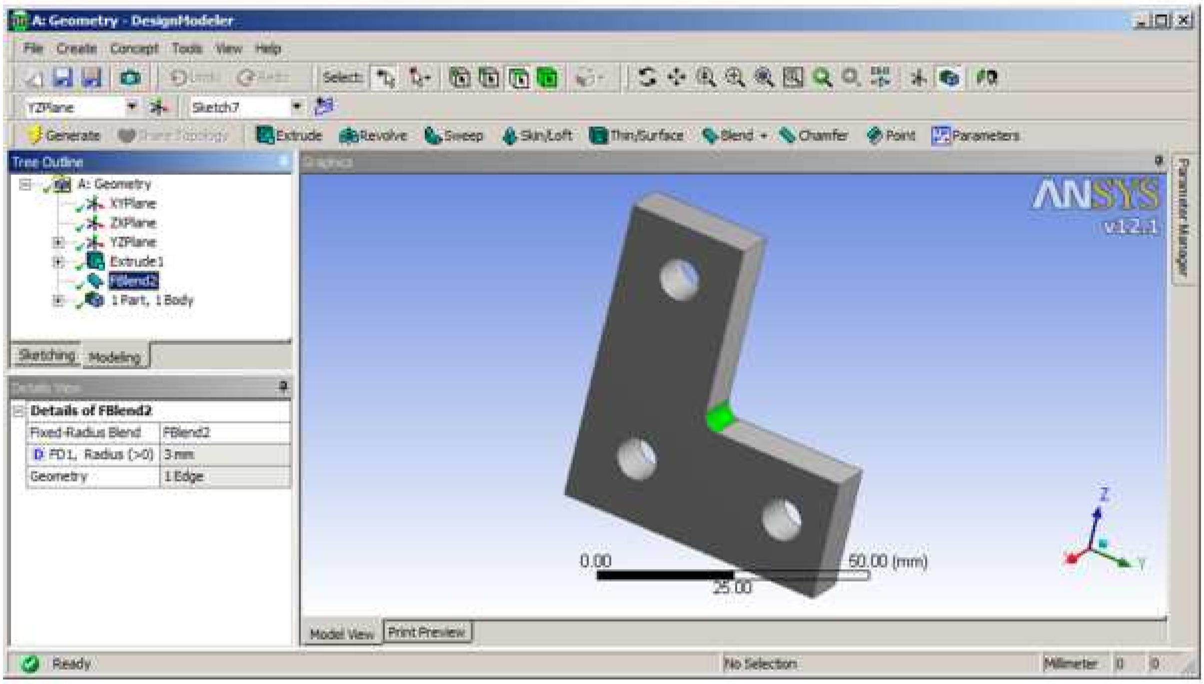
Step 1: Parametric model

The first step is to create a parametric model. In this case ANSYS DesignModeler has been used, but there are a number of different CAD systems that also can be used.



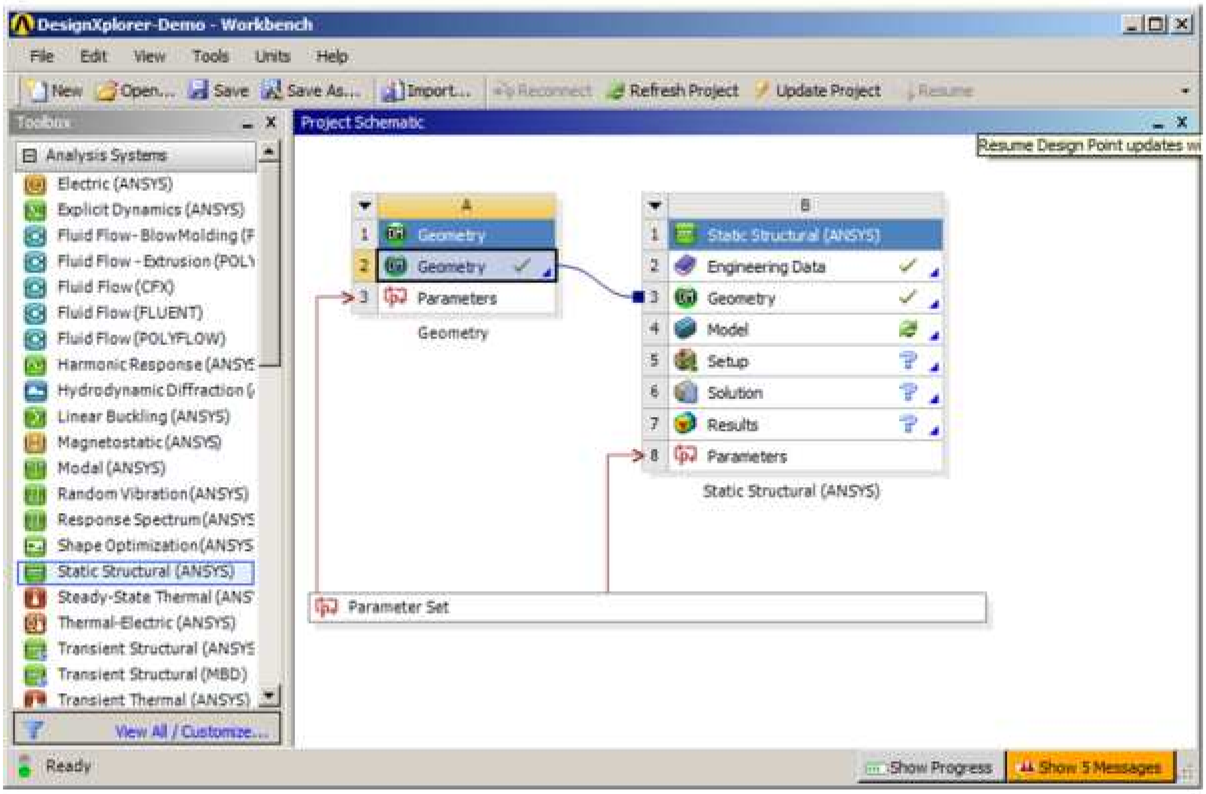
Within DesignModeler the parameters are simply defined by checking the box between the desired parameter. In this case we have two parameters. A thickness and a radius.





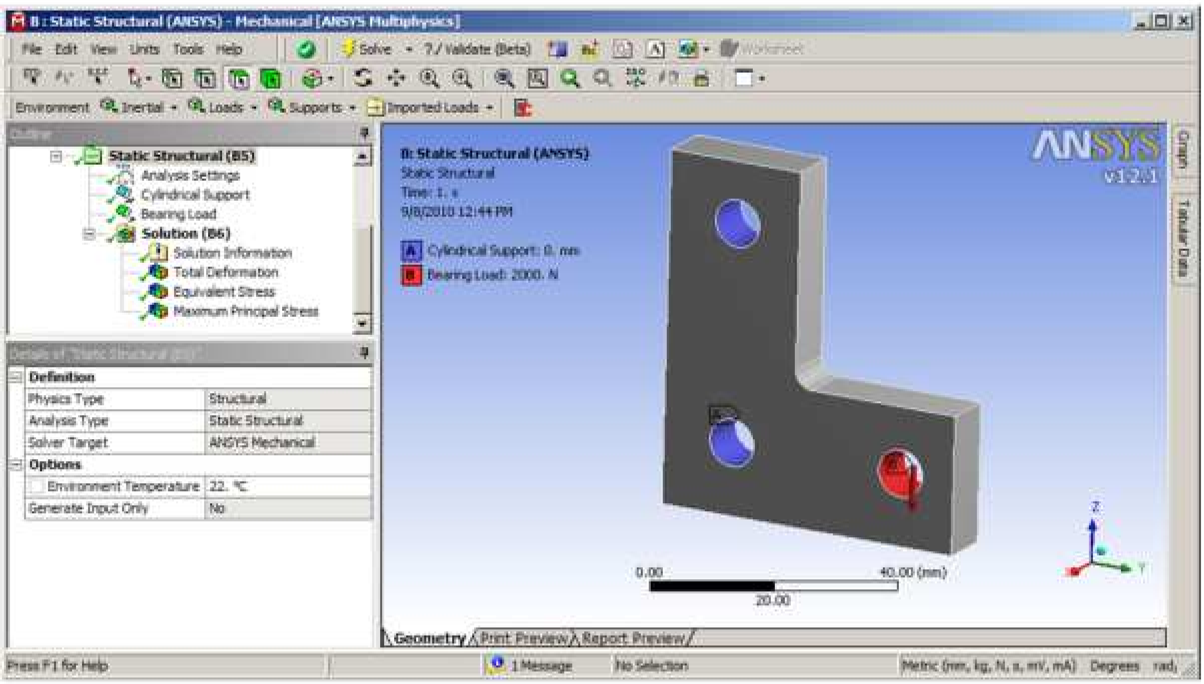
Step 2: Set up the analysis to be run.

The next step is to transfer the parametric model from DesignModeler to Mechanical and set up the analysis. In this case we will run a static analysis which is simply dragged and dropped onto the defined geometry. Note that there now is a Parameter Set associated with the geometry and the Static Structural project.

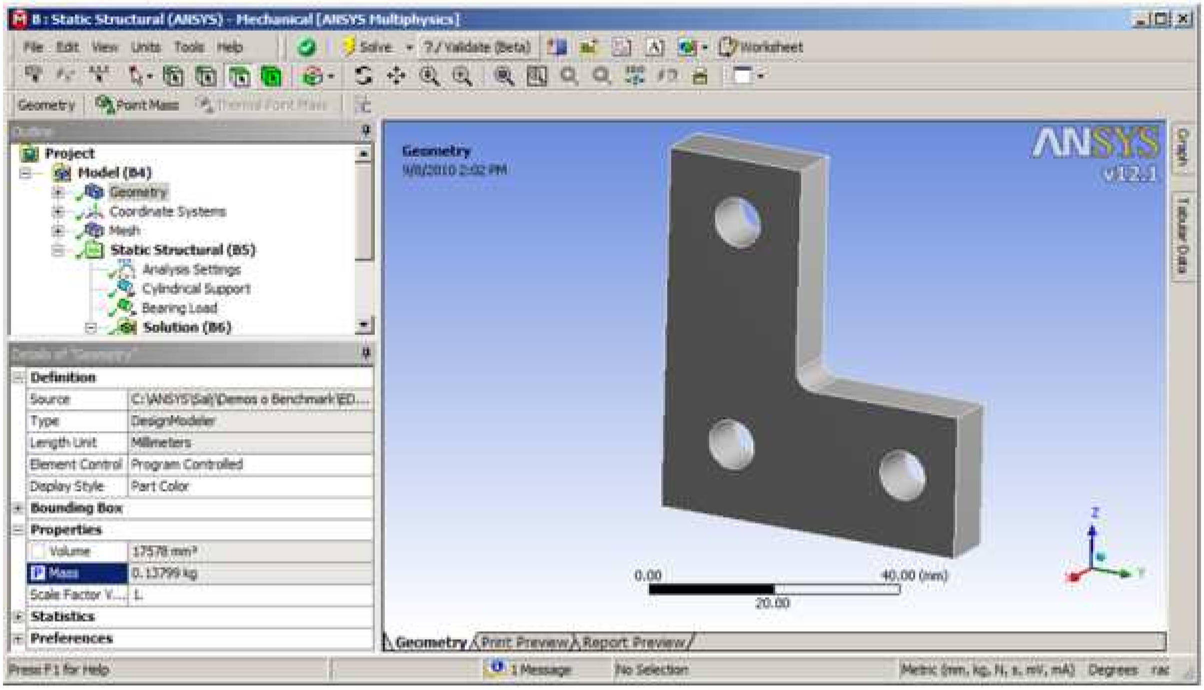


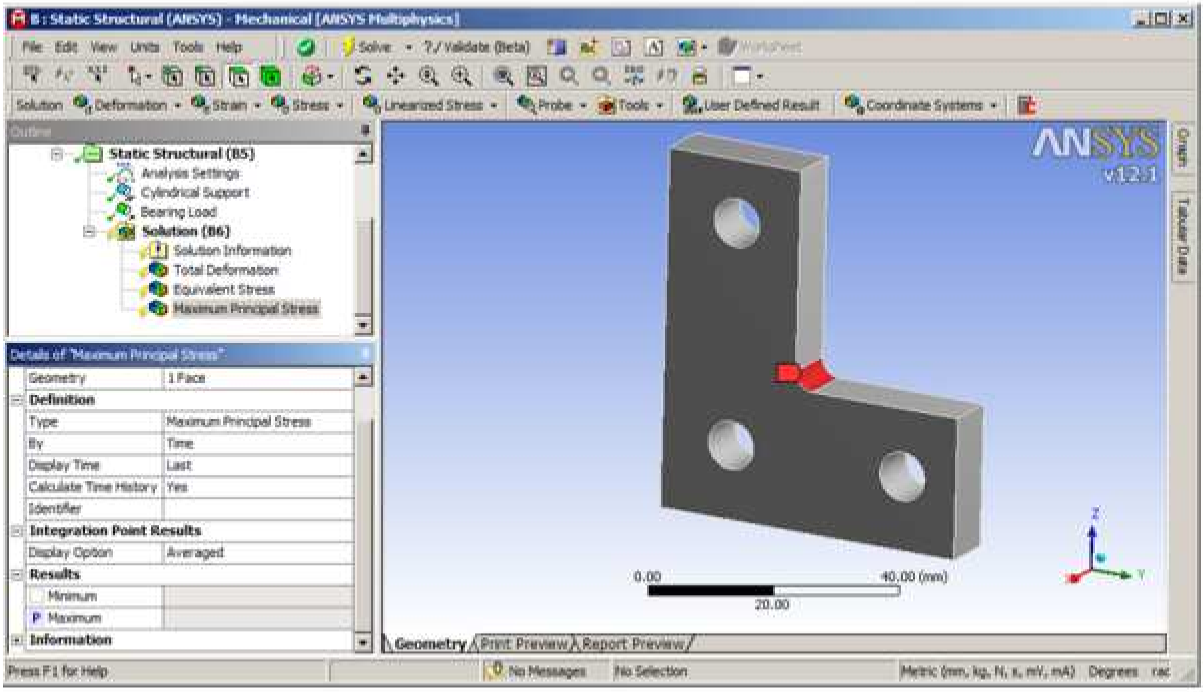
Now you will have to define your materials, loads, boundary conditions, mesh settings etc. as usual in Mechanical. What resides from an ordinary analysis is that you also will have to define you output parameters. In this case we will use total mass and Maximum Principal Stress as output parameters.

First define loads,



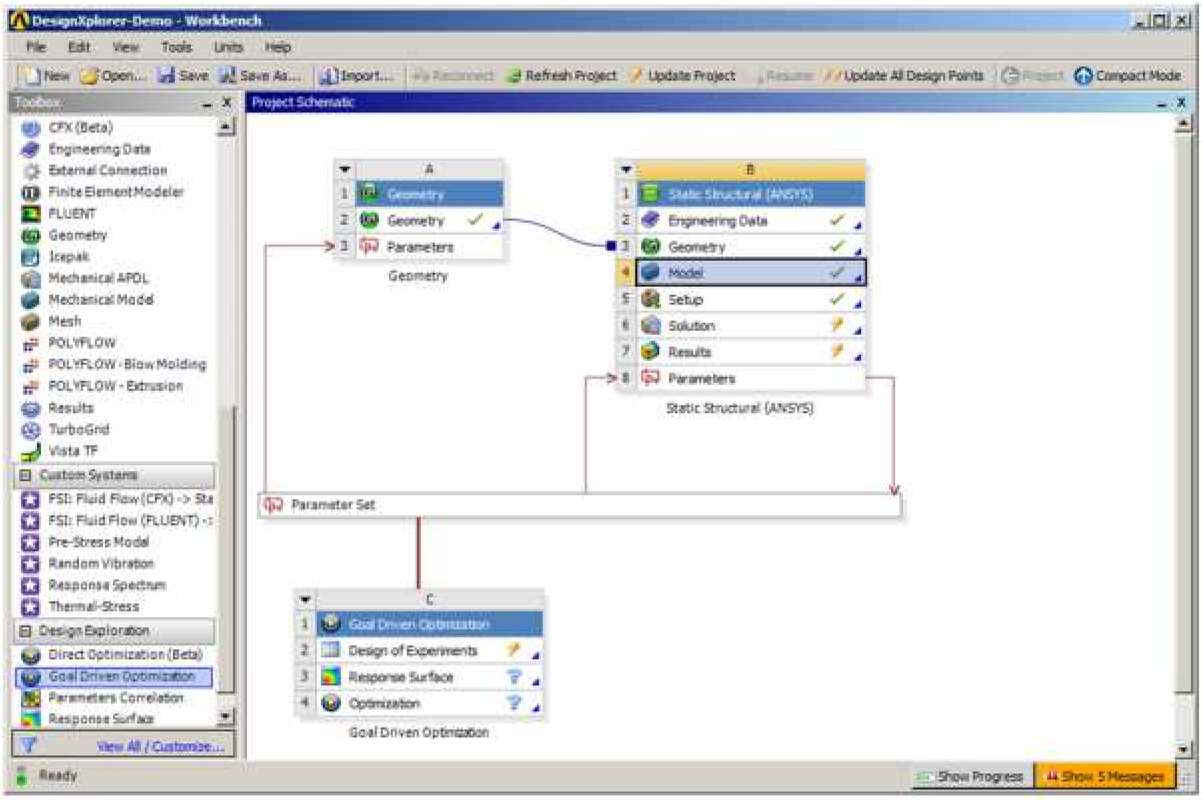
then define output parameters.



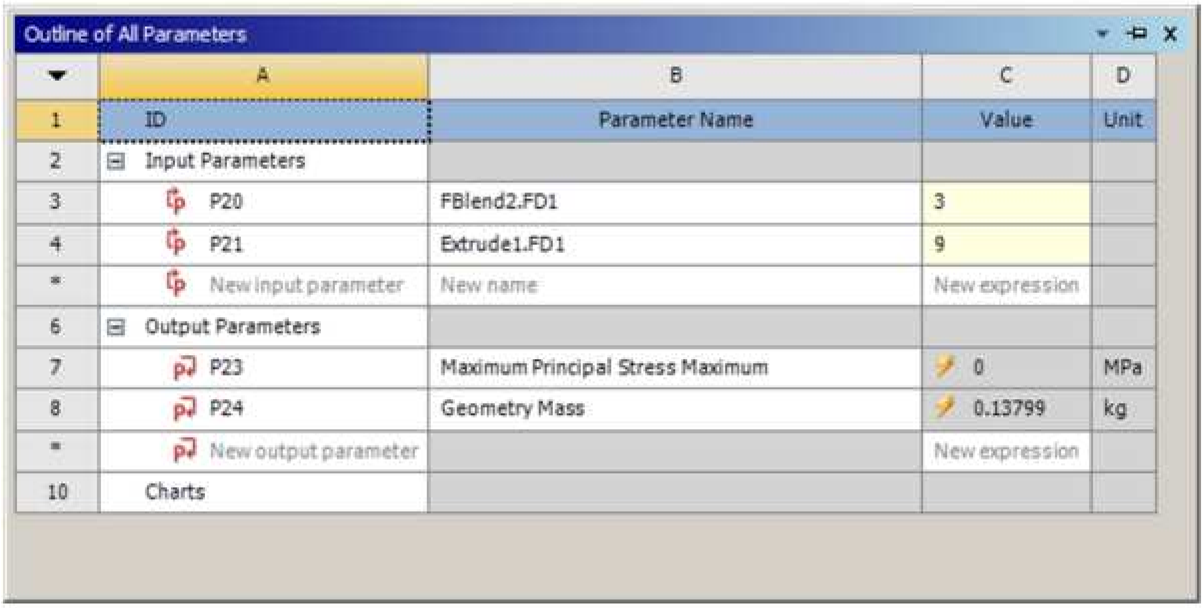


Step 3: Add a Goal Driven Optimization to you project.

The next step is to add a Goal Driven Optimization to the current project. To do this, simply drag it from the toolbox and drop it onto the current analysis.



One could also check the parameters set to see that all your parameters are properly defined.

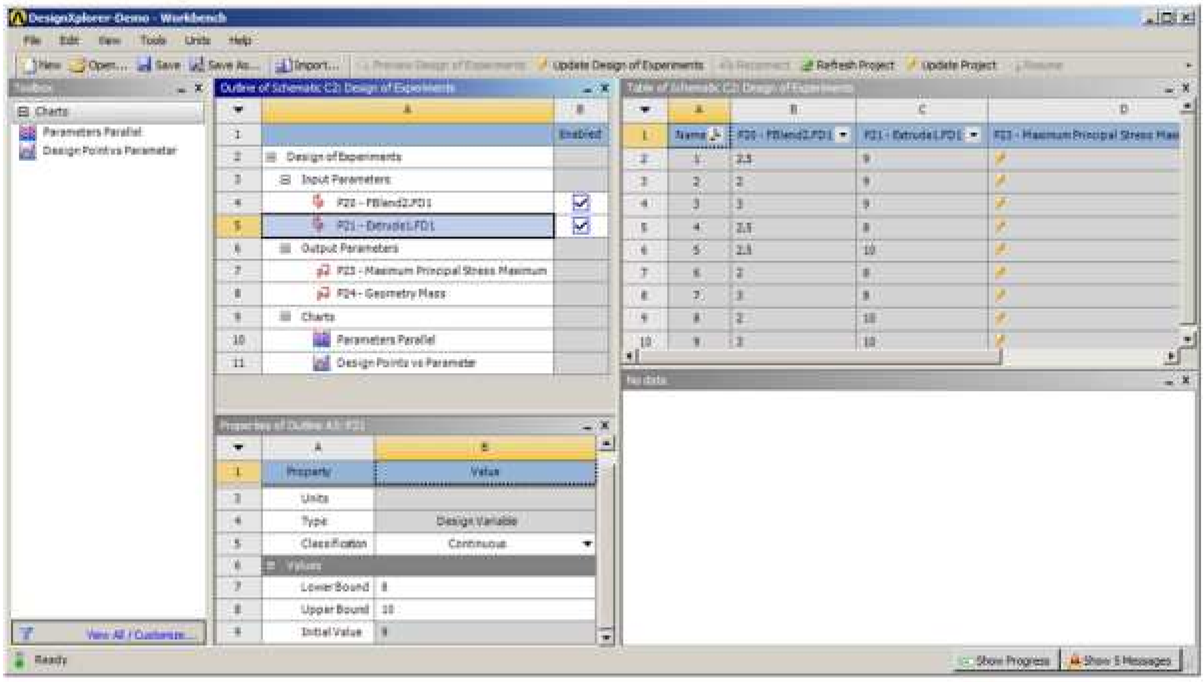


Outline of All Parameters

	A	B	C	D
1	ID	Parameter Name	Value	Unit
2	Input Parameters			
3	P20	FBlend2.FD1	3	
4	P21	Extrude1.FD1	9	
*	New input parameter	New name	New expression	
6	Output Parameters			
7	P23	Maximum Principal Stress Maximum	0	MPa
8	P24	Geometry Mass	0.13799	kg
*	New output parameter		New expression	
10	Charts			

Step 4: Define Design of Experiments.

To define Design of Experiments, double click Design of Experiments in the Project Schematic. The goal here is to set up an appropriate number of analysis to get as accurate response surface as possible. (And of course, the fewer parameter combinations the better when it comes to calculation time.) For each input parameter you will have to define boundaries in between which they are allowed to vary. In this case the radius are allowed to vary between 2 and 3 mm. The thickness is allowed to vary between 8 and 10 mm. The picture below shows how the boundaries are defined for the radius together with the selected design points to be run.



Step 5: Run the Design of Experiments.

This part of the analysis is usually the most time consuming part. When pressing the button Update Design of Experiments ANSYS will sequentially run an analysis for each suggested design point. In this case, where we have two input parameters, ANSYS suggested 10 design points to be run. If we have more input parameters we need more design points to be able to create an accurate response surface which will lead to even more analyses to be run. If that is the case, ANSYS luckily has a solution to speed up this process. If you have a computer or cluster with several CPU's available and you combine that with a parallel license for ANSYS you can run the analyses in parallel instead. This will drastically speed up the calculation time compared to doing a sequential run.

The figure below shows the results of the Design of Experiments after all design points have been run.

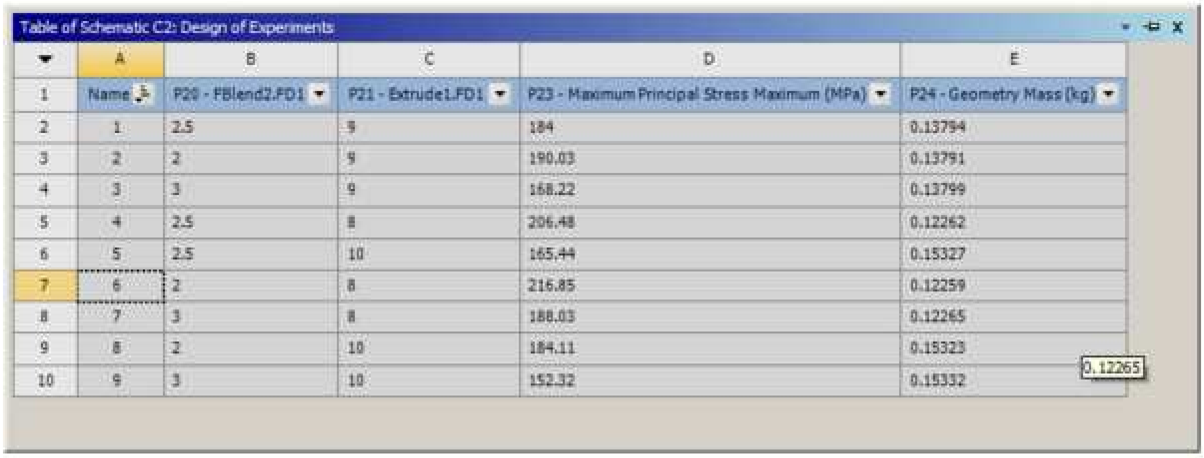
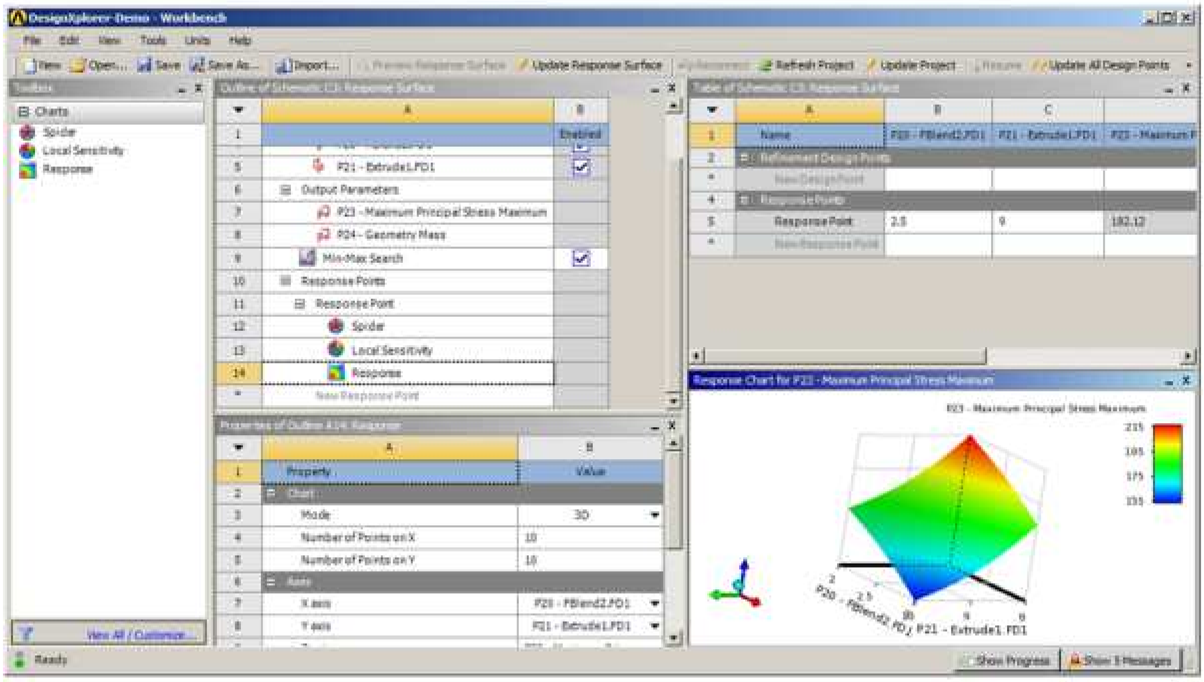


Table of Schematic C2: Design of Experiments

	A	B	C	D	E
1	Name	P20 - FBlend2.FD1	P21 - Extrude1.FD1	P23 - Maximum Principal Stress Maximum (MPa)	P24 - Geometry Mass (kg)
2	1	2.5	9	184	0.13794
3	2	2	9	190.03	0.13791
4	3	3	9	168.22	0.13799
5	4	2.5	8	206.48	0.12262
6	5	2.5	10	165.44	0.15327
7	6	2	8	216.85	0.12259
8	7	3	8	188.03	0.12265
9	8	2	10	184.11	0.15323
10	9	3	10	152.32	0.15332

Step 6: Calculate Response Surface.

When we have a solution for all design points we can calculate a response surface. As mentioned before this is a curve fit through the results for each design point so this is a quite fast operation.



Besides the capability to plot the response surface itself, there are here a lot of capabilities to plot the connection between input and output variables. Among other things we have the possibility to plot how sensitive output results are based on changes in the input variables.

Step 7: Run Optimization.

Now when we have a response surface we can use this to find an optimal design. When you opened Optimization you have the possibility to set different objectives for you parameters and also the possibility to weight those in regard to importance. In this case we have an objective for the Principal Stress and the Geometry Mass. For the stress we've set a criteria where we want to have a structure that don't experience any stresses above a target value of 180 MPa. This is set to high importance. For the mass we've set an objective to minimize the mass with importance low. The input variables are left with no objective since there we are only interested in that they are inside the boundaries that we defined from the start. Any value in between those boundaries are ok.

Table of Schematic C4: Optimization

	A	B	C	D	E
1		P20 - FBlend2.FD1	P21 - Extrude1.FD1	P23 - Maximum Principal Stress Maximum (MPa)	P24 - Geometry Mass (kg)
2	Optimization Study				
3	Objective	No Objective	No Objective	Values <= Target	Minimize
4	Target Value			180	
5	Importance	Default	Default	Higher	Lower

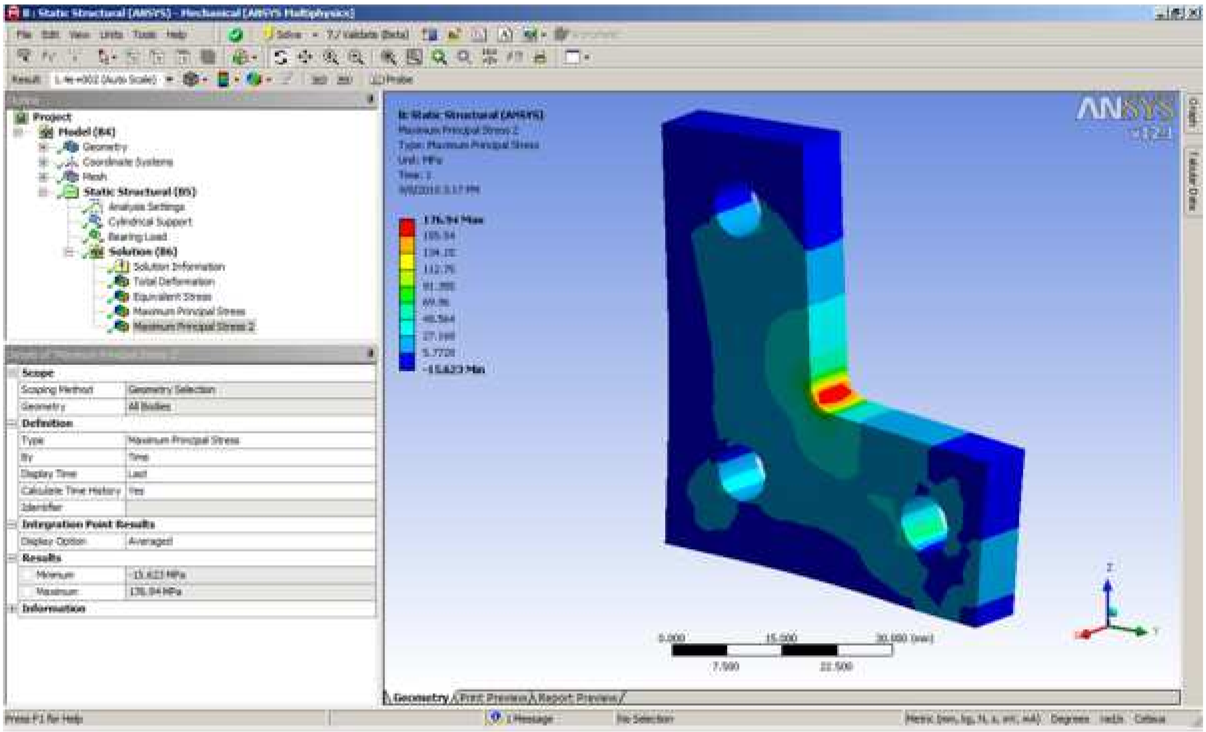
When we've defined objectives it is just to run the optimization and rate the different candidates that ANSYS gives us. In this case ANSYS gives us the following candidates to choose between.

Table of Schematic C4: Optimization

	A	B	C	D	E
1		P20 - FBlend2.FD1	P21 - Extrude1.FD1	P23 - Maximum Principal Stress Maximum (MPa)	P24 - Geometry Mass (kg)
2	Optimization Study				
3	Objective	No Objective	No Objective	Values <= Target	Minimize
4	Target Value			180	
5	Importance	Default	Default	Higher	Lower
6	Candidate Points				
7	Candidate A	2.925	8.4631	180.12	★★ 0.12967
8	Candidate B	2.905	8.7131	★ 175.2	★ 0.13351
9	Candidate C	2.785	8.9006	175.43	0.13641

Now we can select the candidate that suites us the best. Based on the rates (stars) given by ANSYS, is seams like Candidate B is the best solution for us. When selected the appropriate candidate it could always be a good practise to insert this as a design point and run the analysis again to make sure that there is no substanially difference between the real solution and that taken from the response surface.

The result from the final run can be seen in the figure below.



Do you want to learn more?

We have an ANSYS training course: Introduction to ANSYS DesignXplorer

Please contact ANSYS Technical Support at EDR on +47 6757 2120 for technical questions, or ask for a sales representative for questions about licensing.

...or simply send us your contact information and we will get back to you:

Kontaktskjema

You will also find more tutorials in our **ANSYS Blog!**

Relaterte produkter

- ANSYS Structural Mechanics

Se alle produkter

Leif Tronstads plass 4, 1337 Sandvika | Tlf: 67 57 21 00 | E-post: firmapost@edr.no