A VACUUM VESSEL'S OPTIMIZATION WITH ANSA AND ISIGHT

Rong Tang*, Xing Hao
Beijing FEAonline Engineering Co.,Ltd, China

KEYWORDS –
Ansa, Task manager, MORPH,optimization.

ABSTRACT –
This papers discusses the design optimisation process that was followed for a vacuum vessel structure. The ANSA software was used for the generation of the mesh and the set-up of the optimization task. The ABAQUS software was used to solve the model, and the μETA one to understand and analyze the results and identify areas to reduce the structures weight. Through the ANSA MORPH function, a parameterized model was established, the optimize interface was set in ANSA &μETA and the optimization task on the structure wight loss in ISIGHT.

TECHNICAL PAPER -

1. INTRODUCTION

In avacuum vessel'sstructure design project, there was a need to make improvements to an existing vacuum vessel, and to meet the strength requirements of the structurewhile reducing the weight to a minimum level. We had proposed two ways to accomplish this optimization job:The Experimental design optimization (DOE) method and the Numerical optimization method.

Because DOE methodis just an approximate optimization one, and becausethe weight requirementsare very strict, the Numerical optimization method was selectedfor the vacuum vessel model's optimization task.

Toestablish the vacuum vessel's FEA model and set the optimization tasks the ANSA software was selected as the most suitable and effective solution. The ABAQUS software was usedfor solvingand μETA for post-processing.

The vacuum vessel is an axially symmetrical model.Part of the model and the cross-sectionare shown below:

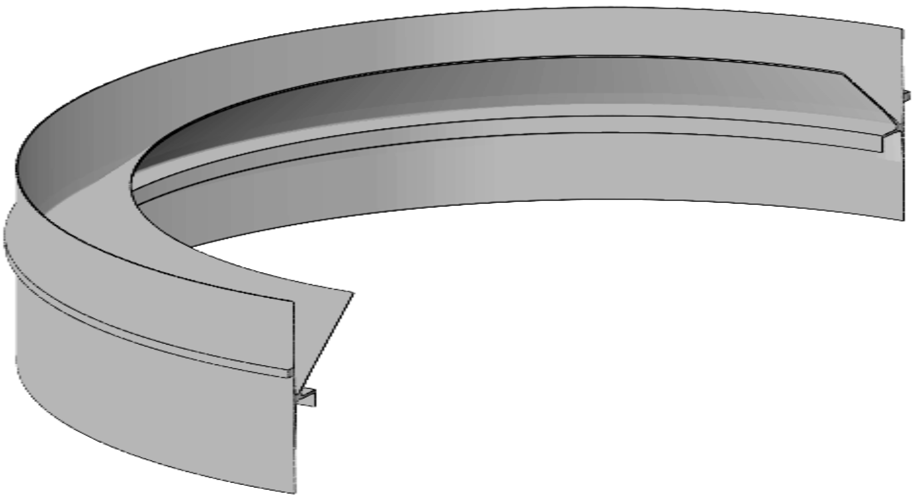

Figure 1 –part of the model

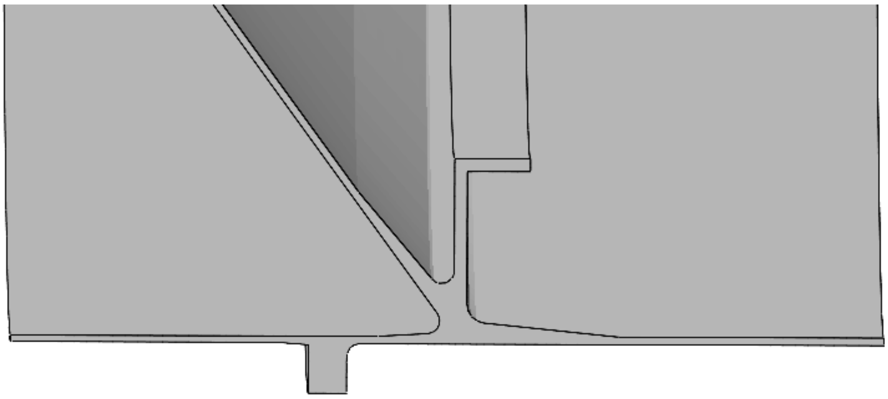

Figure 2 – cross-section

First a mesh model from geometry was createdfrom a two-dimensional CAD drawing in STP format. Line features were imported into the ANSA environment.Then in the TOPO Module,by using line features to create the cross-section plane, and by doing the necessary geometry clean up the model was ready to be meshed.

In the mesh Module, the grid parameters were adjusted, and the mesh type was specifiedas type CAX. After meshing the model, and in the ABAQUS deck module, the material propertiesand boundary conditions were specified.A 0.1 MPa external pressure loading was appended,and the step information was set. The model, ready to be solved, was then exportedasan ABAQUS inp file.

When the solution was generated, the μETA for post-processing software was used to analyse theresults. The maximum von misses stress was about 110 MPa. Because the vacuum vessel is made of aluminium alloy, the design strength should be set to 115MPa. So there was still some space for optimization.

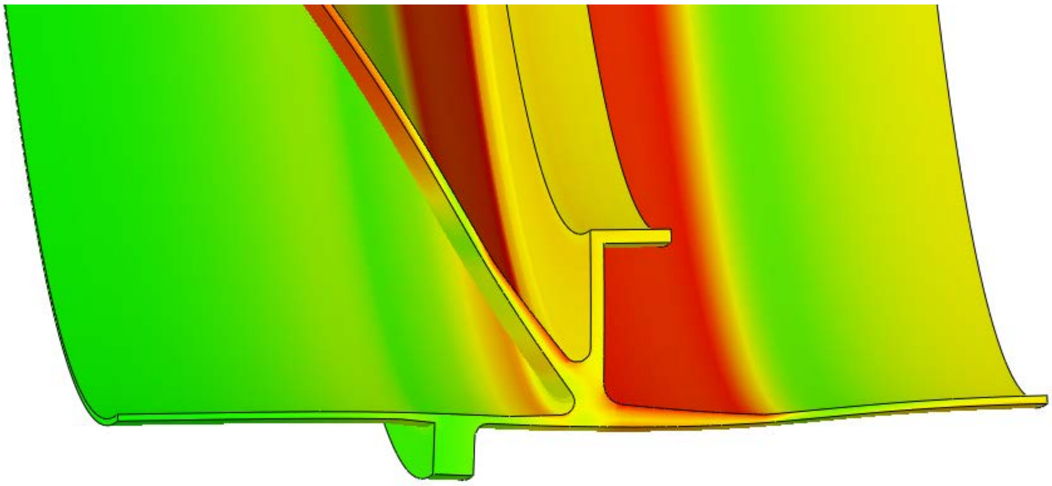

Figure 3 –VonMisses stress

AMORPH model was established, so that the shape of the finite element model could be changed freely. In the ANSA environment,the morph Module was switched to2DMORPH functionsin order to create2D morph boxes.The 2D morph boxes where adjusted to fit the finite element model well, and thenthe finite element mesh was LOADED into the Morph box.

Theoptimizationtaskwas defined in ANSA's Task Manager.Morph parameters were mapped to design variables.Session commands where set to output a mass.log file which contains the model's mass information. A DV File's path was set so that ANSA would generate a file that contained the design variables.A FE output file path and file name was given.When the Task Manager's task was executed, ANSA generated a DV file, a mass_log file, and an inp file corresponding to the design variables.

After set up the deck, a session file was generated by μETA via recording the μETA's functions used. Consiquentlya session_result file was generated from which the maximum Von misses stress was fetched out.

The Isight softwarewas set up to for the optimization.In Isight, three Simcode components on the workflow line were created, and renamed with ANSA, ABAQUS and μETA. In ANSAcomponent, the DV_file.txt was selectedas data source;its design variables' current value was chosen and set as input parameters. The mass.log was selected as data source,and its' mass information output parameters were set. Then, the command linewas set,so that Isight could call ANSA and run the Optimization task automatically.

In ABAQUS component, the command for ABAQUS at the Command Line field was entered. In μETA component, the responses.ses.results were selected as data source,and its' maximum stress output parameters were also defined. The command line to call μETA automatically was also set.

In the optimize component, the NLPQL solver was selected and the upper and lower value of design variables were set, together with the objective variable and the working directory.

After running the optimization task the result were generated. Figure-4 shows the section before and after optimization:

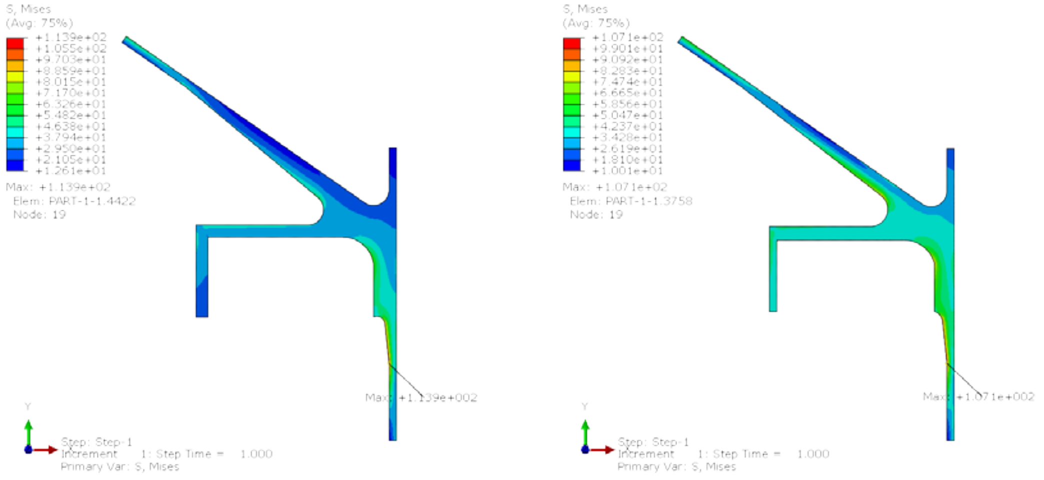

Figure 4 –section before and after optimization

Table 1 shows the calculation summery:

Section	area(mm)			Misses stress(MPa)		
	Before	after	%	Before	after	%
A	5881.0	5058.0	-14.0	65.4	114.6	21.6
B	6265.3	5687.6	-9.2	103.6	108.0	4.2
C	5618.6	5661.4	0.7	108.09	106.6	-1.4
D	5247.0	4626.0	-11.8	113.9	107.1	-6.0

Table 1 section before and after optimization

X. CONCLUSIONS

Undertaking the optimization task with ANSA and Isight, we have got an optimal set of parameters to meetthe requirements of the weight loss plan without loss of structure strength.This can reduce the cost of material and make the product lighter, thus we can control the vacuum vessel easier.

REFERENCES

(1) ANSA version 12.1.5 User's Guide, BETA CAE Systems S.A., July 2008

(2) μETA PostProcessor version 6.2.0. User's Guide, BETA CAE Systems S.A., June 2008