DESIGN OPTIMIZATION WITH ANSA MORPH

[1]Tobias Eidevåg*, [1]David Tarazona Ramos*, [1]Mohammad El-Alti

[1]Alten AB, Sweden

KEYWORDS –
Morphing, CAE workflow, Optimization, Automation, DOE, Regression, CFD, FEM, Python

ABSTRACT –
During the last decades, the use of FEA for solving mechanical problems has experienced an exponential growth, both in the fields of fluid and solid mechanics. A broad range of FEA tools can be found nowadays in the market for modelling, analyzing and processing the results. In this work, an optimization workflow for product design was developed, based on connecting ANSA, a CAE modelling tool and the postprocessor META together, concretely taking advantage of the MORPH feature of ANSA. By generating Design of Experiments, a response surface can be calculated and the optimal values of the design variables can be decided. The presentation is divided into two parts which will show the method applied to different industrial applications. The first part is in the field of fluid dynamics and is in collaboration with AB Volvo Penta. The design of a turbo inlet pipe is optimized using the CFD solver FLUENT together with ANSA and META. The second part is within the field of solid mechanics where a cable drum for off shore operations designed by Svensson Group is optimized regarding geometrical parameters in order to provide a more robust and sustainable design. ANSA MORPH presents a very useful tool for improving design performance in a wide range of modelling approaches.

TECHNICAL PAPER -

1. INTRODUCTION

Shape optimization for industrial applications is now extensively used in the industry. Manufactures in different industries are gradually adopting optimization strategies and shape optimization in particular in their product development cycles. It is to develop more environmentally friendly products to meet the growing environmental requirements and the harder international competition. Shape optimization is also of importance in future product development.

Traditional engineering design is based on refining an initial solution and hope that the solution remains feasible when the design process is completed. Nevertheless, it occurs often that at certain point of the design chain, the solution is no longer feasible regarding any of the criteria and so, backtracking and rework is required. This problem has motivated the development of alternative design approaches, which intend to keep a more flexible solution space in order to avoid too restrictive designs. Probably the most known approach belonging to this group is the Set-Based Design approach, born and implemented in the 1990s for a more efficient way of building cars at Toyota (1).

In Set-Based Design, neither the initial design space nor the final one is made of single solutions but sets of alternatives. This makes the process more flexible, highly increasing the probability to obtain an optimal design and avoiding rework, but also resulting into a set of feasible alternatives which can be used in future projects.

Although there is a theory and established methodology behind Set-Based Design as it was identified in Toyota, the application of it depends on the type of project that needs to be assessed. One approach to implement Set-Based Design takes advantage of the continuous increasing of computational power of computers and computer clusters. This gives the opportunity to test sets of initial alternatives in an automated way, and find optimal solutions among them. In this paper, two different approaches for applying Set-Based Design on the fields of fluid dynamics and solid mechanics are presented, which combined mathematical optimization methods by keeping in mind the principles of Set-Based Design.

Surrogate models are adopted as the optimization approach. The specific model is the polynomial response surface methodology (RSM). The choice of this optimization strategy is mainly because its robustness and that the aim is to find a region of feasible designs instead of using the usual gradient-based local minimum/maximum seeking optimization strategies. It is the understanding of the response on each parameter on the system that makes this algorithm useful in physical analysis.

The optimizations were performed using Ansa 15.2.4 primarily based on using the features in MORPH and an Optimization Task Manager under Task Manager. The optimization task manager makes the process of generating designs fully automated and therefore do not require any user input when the design space is defined. The general method for the design optimization is visualized in Figure 1.

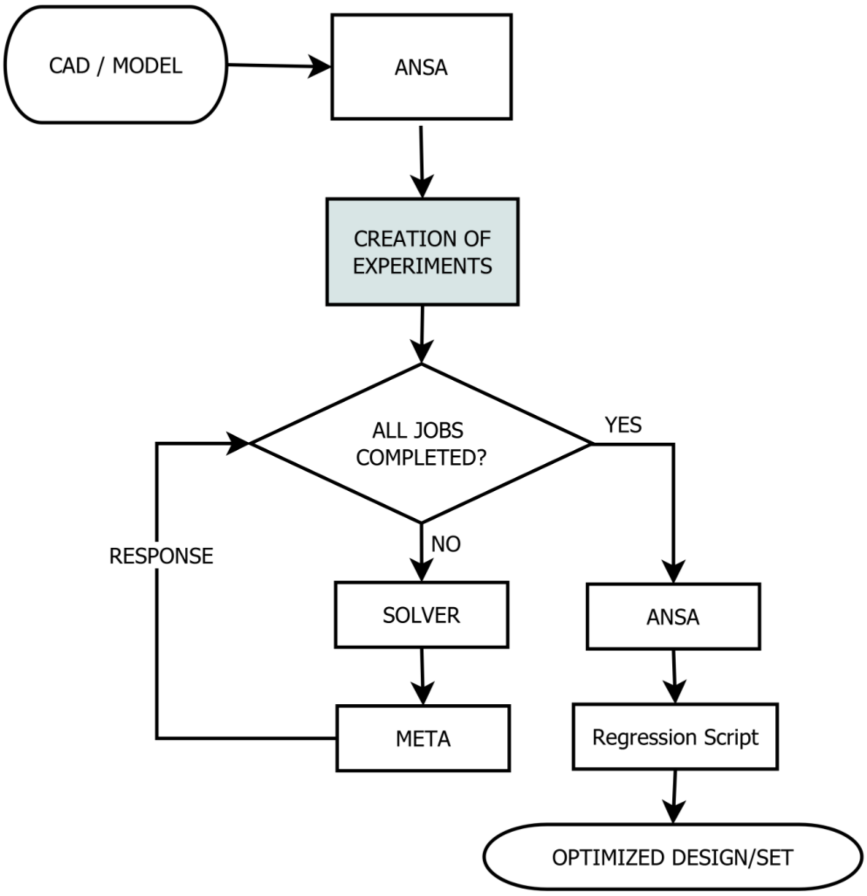


Figure 1 - Method Flow Chart

Once the model has been set up, either from a CAD file or already built model, the different experiments are created. This step is performed within the Optimization Task Manager, where all the design variables are specified and linked to the different operations. These operations depend on the case to assess.

2. TURBO INLET – FLUID DYNAMICS CASE

The Geometry and Computational Model

The study is performed on an inlet pipe of a turbo unit. The pipe geometry is composed of two main parts: a 90° bend and a 180° bend. The baseline geometry is shown in Figure 2. In this paper the Ansys Fluent CFD code was used to perform the numerical simulations. The fluid temperature is set to 25°C at the inlet. The inlet mass flow is set to 0.33 kg/s which correspond to an engine running speed at 2500 rpm. The pipe walls have no-slip and adiabatic boundary conditions. The k-ω SST turbulence model where used with second order discretization schemes and the simulations where done in compressible mode since some velocities become close to 0.3 in Mach number. The k-ω SST turbulence model requires a fine mesh resolution especially in the near-wall region. Therefore 10 layers in near wall region are generated with a minimum cell layer of 0.05 mm height. This gives y^+ values below 5 in most parts of the computational domain. The computational mesh comprise of 2.2 million penta and polyhedral cells for the baseline geometry.

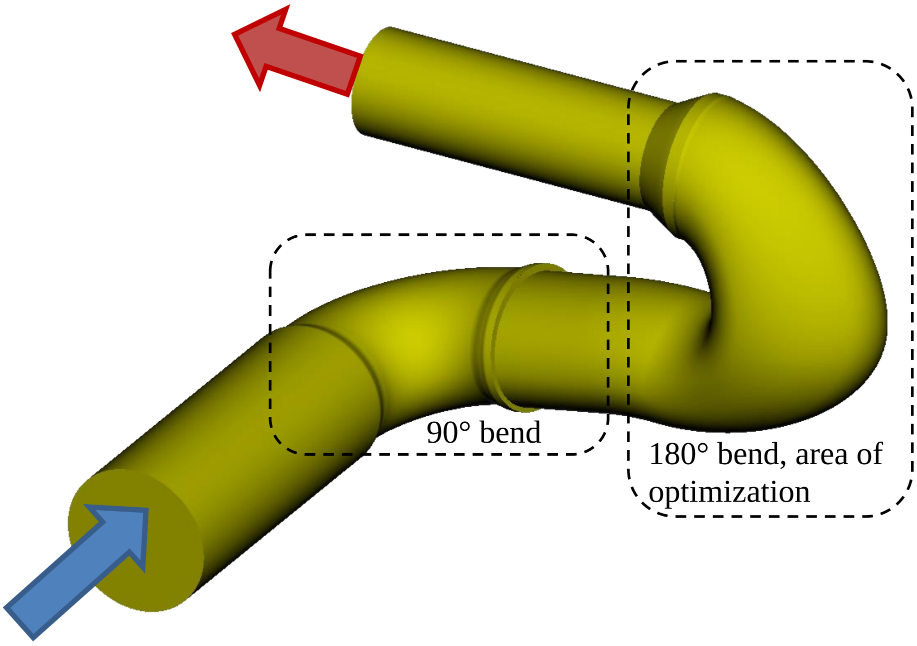


Figure 2 – Baseline Geometry

Optimization Approach

The focus for this optimization is to minimize the total pressure drop by changing the 180° bend. The optimization method is a polynomial Response Surface Methodology (RSM) and the idea is to explore several variables relationship with a response variable. The idea of RSM is to build an empirical model of the true response surface of the system. The true response surface is governed by physical laws. The data obtained from the design candidates are used to build a mathematically best fitting model. A second-order polynomial model has been adopted to capture non-linearities. The model also includes interaction terms of the different parameters. Here the total pressure drop will be the response for each experiment. The experiments are chosen according to the “Design of Experiment” (DOE) method called faced-centered composite design as presented in (1) and (3). This method generates 2^n+2n+1 experiments where n is the number of design variables. For this optimization four design variables where chosen which with the chosen method generates 25 experiments. In order to smooth geometrical changes, the baseline geometry was surrounded by morphing boxes and the design variables were defined as sets of control points near the design variables positions. The positions and allowed directional change for each design variable are visualized in Figure 3 together with the surrounding morphing boxes. The morph optimizations are performed on the geometry faces of the model and thereafter the mesh is generated using a python script that runs the meshing commands in Ansa. The process of generating experiments is fully automated in Ansa meaning that all experiments are sequentially generated without the need of manual input. The design values for each experiment together with the response (Total Pressure drop ΔP in the system) from running the CFD simulation on the design is shown in Table 1.

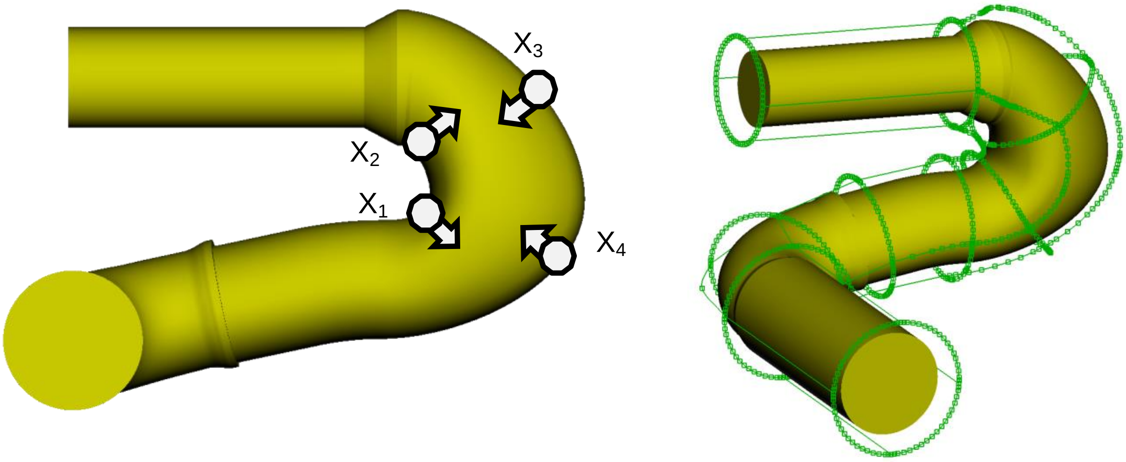


Figure 3 – Design variable locations and surrounding morphing boxes. The arrows display the positive direction for each design variable.

Exp	X_1	X_2	X_3	X_4	Y (ΔP)
1	0	0	0	0	1015
2	-1	-1	-1	-1	1175
3	-1	-1	-1	1	3007
4	-1	-1	1	-1	1110
5	-1	-1	1	1	1547
6	-1	1	-1	-1	1629
7	-1	1	-1	1	1945
8	-1	1	1	-1	1401
9	-1	1	1	1	1579
10	1	-1	-1	-1	1181
11	1	-1	-1	1	1969
12	1	-1	1	-1	1565
13	1	-1	1	1	1804
14	1	1	-1	-1	1265
15	1	1	-1	1	1676
16	1	1	1	-1	1032
17	1	1	1	1	1085
18	-1	0	0	0	1199
19	1	0	0	0	1142
20	0	-1	0	0	1023
21	0	1	0	0	1081
22	0	0	-1	0	1235
23	0	0	1	0	949
24	0	0	0	-1	942
25	0	0	0	1	1252

Table 1 – Design values and total pressure drop response for each experiment.

A quadratic model of the design variables influence on the pressure drop is defined as:

$$y = \beta_0 + \sum_{i=1}^{n} \beta_i x_i + \sum_{i=1}^{n} \sum_{j>i}^{n} \beta_{ij} x_i x_j + \sum_{i=1}^{n} \beta_{ii} x_i^2 + \epsilon$$

where βs are the regression coefficients, x_i is the i^{th} design variable and n is the number of design variables. The β coefficients need to be estimated and this was done by using the

Ordinary least squares method. When having the estimated coefficients a multivariable minimization can then be performed and in this study the optimization algorithm Limited-memory BFGS was used. A minimum of the quadratic model is then found as

$$x_1 = 0.009 \; x_2 = -0.239$$
$$x_3 = 0.060 \; x_4 = -0.970$$

With these design values a new design was created and a CFD simulation on this design was performed. The total pressure drop for the new design was 920 Pa which is a decrease of 9.4 % from the baseline geometry which had a pressure drop of 1015 Pa.
When performing the fit there is a risk that outliners from the experiments have negative influence on the regression fit. The DFBETAS is a standardized measure that shows how much an estimated coefficient would change if an experiment would be removed, see (4) for more details. Let b be the estimated β values and rearrange them from $b_1, b_2, b_3, b_4, b_{11}, b_{12}$ to $b_1, b_2, b_3, b_4, b_5, b_6$; let $b(i)_j$ be the estimated β_j value after observation *i* has been removed; let $s(i)^2$ be the variance estimate after deleting the ith observation; Let X be the design matrix. The DFBETAS is defined as:

$$DFBETAS_j = \frac{b_j - b(i)_j}{s_{(i)}\sqrt{(X'X)_{jj}}}$$

For this case it was noticed that experiment 3 and 7 had high DFBETAS values compared with other experiments. These two experiments also have high pressure drops and are therefore dropped in the regression fit. With these dropped a new minimum where found as:

$$x_1 = 0.101 \; x_2 = -0.069$$
$$x_3 = 0.214 \; x_4 = -0.707$$

Running the CFD simulation on this geometry gave a total pressure drop of 902 Pa which is a decrease of 11.1 % compared with baseline geometry. The summary of the regression coefficients, t statistics and confidence interval can be seen in Table 2.

	Estimated Value	t statistic	95 % Confidence Interval	
β_0	976.7	29.1	898.3	1053.0
β_1	-61.0	-2.6	-114.9	-7.0
β_2	-13.5	-0.7	-59.6	32.7
β_3	-124.0	-5.3	-178.0	-70.1
β_4	210.3	9.0	156.3	264.3
β_{11}	201.7	4.2	89.9	313.6
β_{12}	-163.8	-7.6	-213.3	-114.3
β_{13}	45.9	1.8	-12.7	104.6
β_{14}	-30.8	-1.2	-89.4	27.9
β_{22}	82.8	1.7	-29.1	194.6
β_{23}	-97.3	-4.5	-146.8	-47.8
β_{24}	-55.7	-2.6	-105.2	-6.2
β_{33}	123.0	2.5	11.1	234.8
β_{34}	-103.9	-4.1	-162.6	-45.3
β_{44}	128.1	2.6	16.2	240.0

Table 2 - Regression Summary

A comparison between the baseline design and the optimal design can be seen in Figure 4.

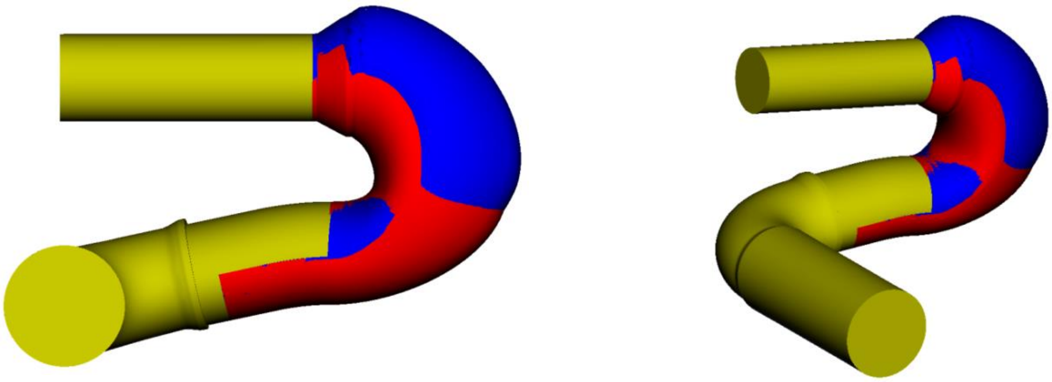


Figure 4 – Differences between the designs where red is Baseline design and blue is optimized design. Yellow color means no difference between the designs.

A comparison between the velocity profiles in a plane between the baseline design and optimized design can be seen in Figure 5. High velocity gradients result in large pressure drops in the system. It can be seen in Figure 5 that the velocity gradients near the inner radius turn have decreased for the optimized design compared with the baseline.

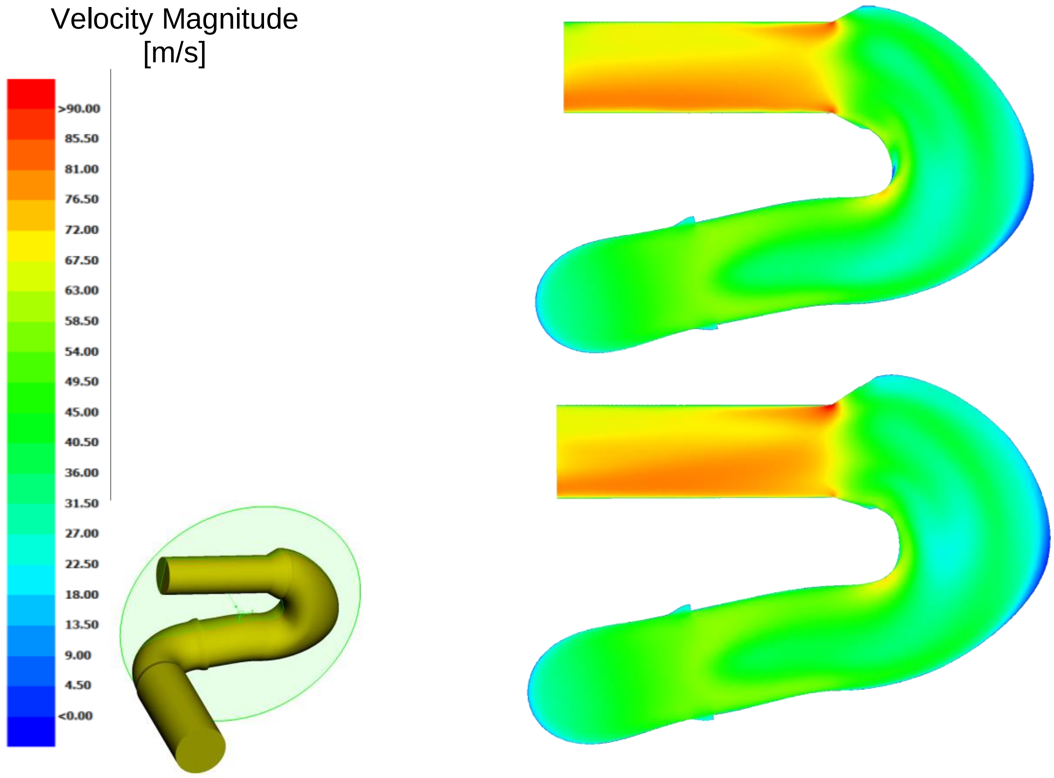


Figure 5 – Velocity magnitude in cross-section plane of turbo pipe for baseline design (at top) and optimized design (at bottom)

With the quadratic model the design space can be analyzed even further. With a limit for the pressure drop one can calculate the set of designs that is below this limit. How robust a design is with a change in design variable can also be an important aspect. A visualization of the design space can be seen in Figure 6 where two variables for each plot are varied and the other two are fixed. The figure visualizes that design variable 4 has a strong influence on the total pressure drop and that it is beneficial with negative value.

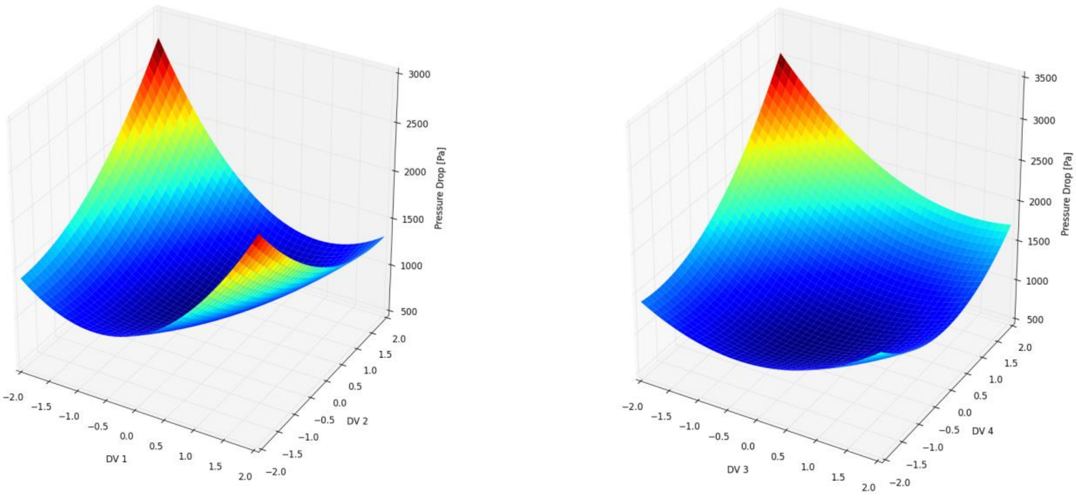


Figure 6 – Response surface fit

3. CABELDRUM – SOLID MECHANICS CASE

The Geometry and Computational Model

For this case, a cable drum concept for transport operations within the oil and gas industry is analyzed. The drum is composed of a main cylindrical body and eight spokes at each side connected to the circular girders which define the outer border of the reel. The diameter of the baseline cylindrical body is 2350 mm whilst the outer diameter is 4075 mm. During transport of the loaded drum on a truck, an extreme load case can be identified when a lateral acceleration from quick turning maneuvers pushes the cable against the spokes. This load case highly affects the behavior of the spokes, leading to buckling and yielding of these components of the structure. The spokes are rectangular hollow section profiles and their strength can be directly identified with the thickness of the flanges and their length, if the shape of the spoke wants to be kept unchanged. The aim of the optimization process is to optimize the design of the spokes for the aforementioned load case. This can give an optimal solution for drums which are able to carry different amounts of cable.

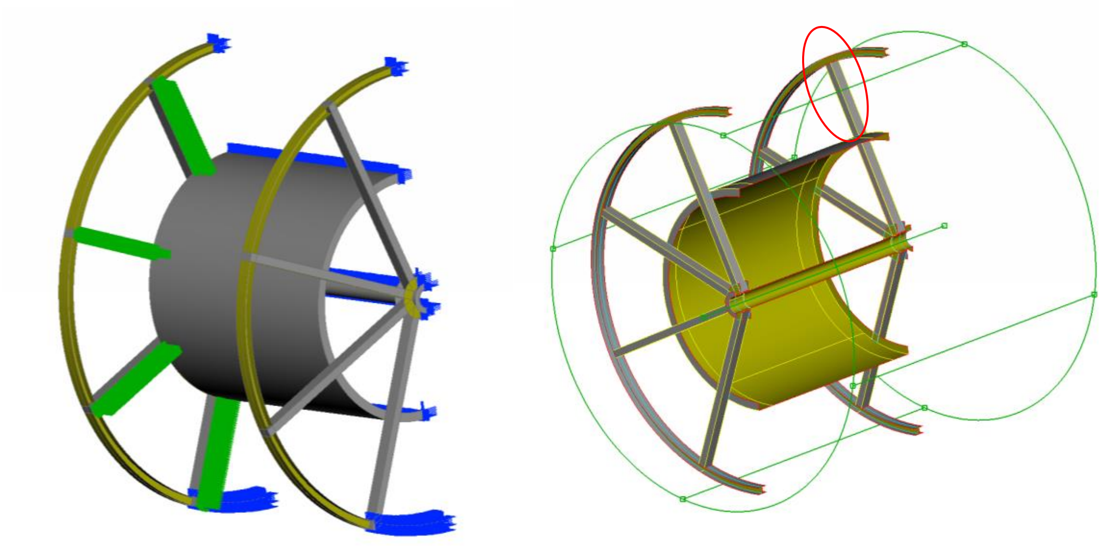


Figure 7 – Model of the cable drum highlighting boundary conditions and loads on the spokes (left) and model showing the morphing box and the purpose of the extend operation (right).

After a mesh convergence study, the drum is modeled with 2nd order shell elements with an average length of 8 mm, and only half of the drum is modeled due to the symmetry of the problem. As meshing algorithm, mapped meshing has been used where possible. All connections are considered full interaction and implemented by merged edges; nevertheless the singularities at the boundaries have been treated to avoid extreme values from modeling issues. For the sake of simplicity, the drum is fixed at the bottom part of the outer circular profiles. Structural steel S355 is used and gravity effects are considered. The aforementioned load case is implemented by applying a load equivalent to half the weight of the cable (as stated in TSVFS 1978:10 and VVFS 1998:95) on the inner surface of the spokes at one of the sides, see Figure 7.

Optimization Approach

The aim of the optimization process is to determine the optimal thickness of the spokes flanges for different spokes lengths. The response of the analyses is the utilization ratio at the spokes, defined as the maximum stress over the material's yielding limit The same optimization method as in the fluid dynamics case is used i.e. the polynomial Response Surface Methodology (RSM). Nevertheless, instead of minimizing the response, the goal is to find fully utilized designs, i.e. utilization ratio equals to 1. As design variables the thickness of the spokes and the extended length of the spokes from the length of the original design are considered. The DOE in this case is Uniform Latin Hypercube with 30 designs and 1 seed.

The intervals used to create the experiments are [3, 12] mm for thickness and [0, 500] mm for extended length. In order to build each experiment, the outer beams are surrounded by a cylindrical morphing box with its outer radius as design variable, which allows moving them radially by keeping their shape. The spokes are then extended up to the beams by means of a script. The load on the spokes is also varied according to the length of the spokes, since longer spokes intend to carry more cable. Both morphing box and extend operations affect the topological features of the drum, see Figure 7, and hence, the mesh is generated after every alternative is built. The meshing is performed by using a python script which tries first mapped algorithm on the whole model, then free meshing on the regions where the mapped algorithm is not possible to be used and finally reconstructs the mesh to improve its quality. At last, the thickness of the flanges is determined by a direct design variable pointing to the thickness of the shell element property of the spokes. This case shows some of the capabilities to be included within an optimization task in ANSA, i.e. morphing parameters directly controlling the shape of morphing boxes, user scripts which can be linked to the design variables, and design variables directly defining element properties parameters. A flowchart clarifying the process and how the design variables affect each operation is shown in Figure 8.

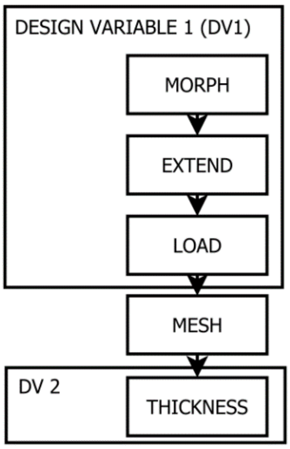


Figure 8 – Cable drum optimization task.

Again, the process is fully automated. The different experiments and results for utilization ratio at the spokes can be seen in Table 3.

Exp	L (mm)	t (mm)	UR
1	190	7.66	0.87
2	293	11.07	0.79
3	69	8.59	0.64
4	172	4.55	1.28
5	500	10.45	1.09
6	0	3.00	1.32
7	310	6.10	1.25
8	17	3.31	1.26
9	379	10.76	0.91
10	241	7.97	0.92
11	86	5.17	0.98
12	34	8.90	0.59
13	52	9.83	0.56
14	466	4.86	1.88
15	103	10.14	0.61
16	121	5.79	0.95
17	431	3.62	2.36
18	138	3.93	1.37
19	155	11.69	0.60
20	207	6.41	1.02
21	362	12.00	0.82
22	414	4.24	1.97
23	224	8.28	0.87
24	259	11.38	0.73
25	276	5.48	1.29
26	328	6.72	1.19
27	345	7.03	1.18
28	448	7.34	1.33
29	397	9.21	1.05
30	483	9.52	1.15

Table 3 – Design values and utilization ratio response for each experiment.

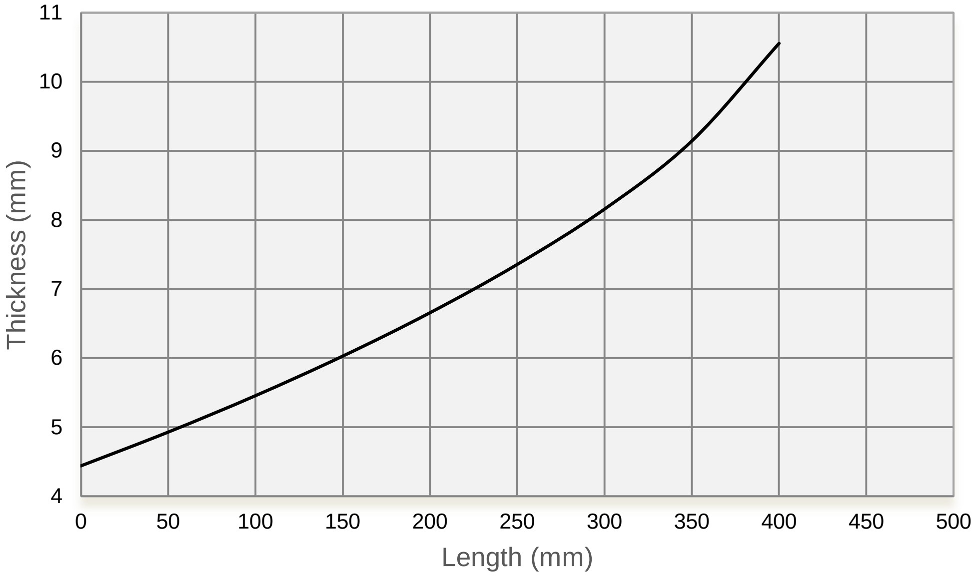
For this case, the same quadratic model as for the fluid dynamics case is used, whose coefficients are solved by means of Ordinary least square method, corresponding the index 1 to the extended length and the index 2 to the thickness of the flanges:

$$y = \beta_0 + \sum_{i=1}^{n} \beta_i x_i + \sum_{i=1}^{n} \sum_{j>i}^{n} \beta_{ij} x_i x_j + \sum_{i=1}^{n} \beta_{ii} x_i^2 + \epsilon$$

	Estimated Value	t statistic	95 % Confidence Interval	
β_0	2.1620	30.655	2.016	2.308
β_1	0.0026	10.162	0.002	0.003
β_2	-0.3416	-16.421	-0.385	-0.299
β_{11}	1.522e-06	3.345	5.83e-07	2.46e-06
β_{12}	-0.0002	-10.175	-0.000	-0.000
β_{22}	0.0180	12.773	0.015	0.021

Table 4 – Regression Summary

Once the response surface is calculated, different results can be extracted depending on the needs. As aforementioned, this case aims to find designs where the spokes are fully utilized, which can be done by seeking designs with utilization ratio equals to 1 see Figure 9 (top). Above certain length value i.e. 400 mm, the response surface shows no fully utilized designs. Hence widening the design space might be necessary to assess designs lying over this region. Regarding concept development, the feasible solutions space, see Figure 9 (bottom), provides with useful information about expected results prior to manufacturing, i.e. for a certain length, an estimate of thickness needed can be obtained.



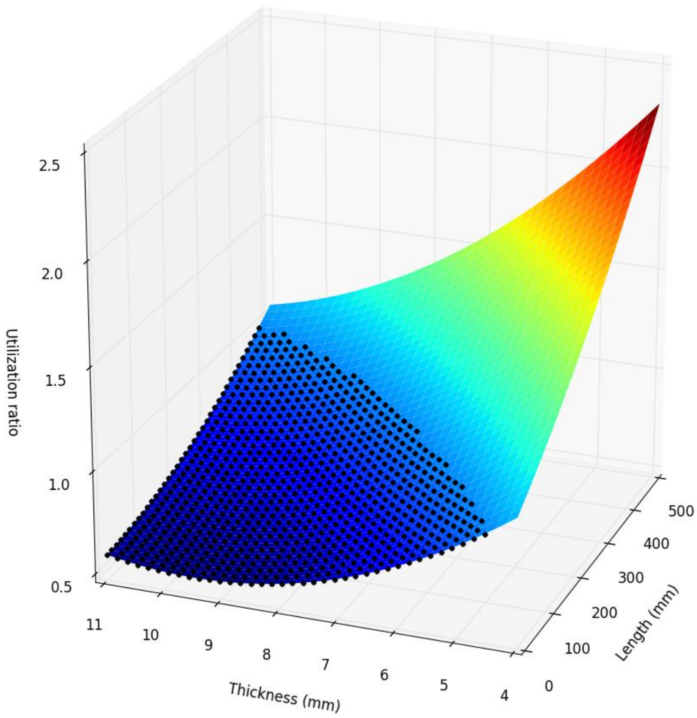


Figure 9 – Fully utilized designs (Top) and response surface and feasible designs (Bottom).

5. CONCLUSIONS

In this paper, an optimization process was carried out in ANSA and META. The automated process of generating DOE in ANSA and connect them to the solvers is important for engineering workflow. It was possible to improve designs using this approach and have a feasible design region of optimal designs for both cases. As improvements for ANSA and META we conclude that it would be beneficial to run the different design candidates in parallel since DOE and RSM main advantage is that all candidate designs can be run parallel in time. The model does not require information between the candidates during the simulation time. This is very useful when running experiments or long CFD computations. In this paper this issue was manually solved using different scripts than the ones generated by ANSA. Another improvement is to include optimization analysis, statistics and plots directly in META to streamline the whole optimization process.

REFERENCES

(1) Luis, S., Tarazona, D. Applicability of Set-Based Design on Structural Engineering. 2014. Chalmers University of Technology.

(2) Krajnovic, S. Aerodynamic optimization of vehicles using computational fluid dynamics and response surface methodology. Science & Motor Vehicles Belgrad 2007 paper NMV0724.

(3) El-Alti, M., Kjellgren, P. and Davidson, L. Shape optimization and active flow control of truck-trailers for improved aerodynamics using Large-Eddy Simulation and Response Surfaces. Direct and Large-Eddy Simulation IX pp 405-410.

(4) Belsley, D. A., Kuh, E., and Welsch, R. E. (1980), Regression Diagnostics, New York: John Wiley & Sons.

(5) ANSA version 15.1.x User’s Guide, BETA CAE Systems S.A., June 2014