Development of a design optimization process using the example of a valve geometry.

Davide Corbo

Airbus Defence and Space

Abstract

The efficient use of materials is really important in many different settings, especially in the aerospace industry. Structures are subjected to many extreme conditions and at the same time, must be light as much as possible. For example when constructing a satellite, its support structure should not use more than a certain amount of material, but should be stiff enough to carry out all its devices. The intention of the present work is to realize an efficient design optimization process, in order to improve any kind of structural component. The optimization aims to reduce the structural weight as much as possible, while keeping the right mechanical behaviour of the structure. The design optimization process is developed by using the parametric optimization approach based on the Design of Experiments (DoE) and the Response surface method (RSM). The structural component in exam is the oxygen/hydrogen balancing nozzle TEO/TEH situated on the upper stage ME of the launcher Ariane 5. The optimization is carried out in two different ways. First of all, the TEx valve geometry is optimized all inside the software *ansys workbench* thanks to the optimization tool *ansys designXplorer*. The second improvement is made by using *ansys workbench* as solver and the software *optiSlang* for the sensitivity analysis and the optimization. Finally the best solution is chosen and the work ends with the conclusions and the future developments.

Keywords: Parametric design optimization, DoE, OptiSlang inside ansys workbench, ansys designXplorer, Response surface method, sensitivity analysis, Shape optimization.

1 Mathematical Optimization Problem

Optimization means to find the best possible solution under given constraints. The goal of the optimization problem is usually some sort of maximization or minimization, like minimization of the mass or maximization of stiffness. In the following, the general mathematical optimization problem is shown, (in this case is formulated as minimization of the objective function) subject to constraints.

$$\begin{gathered}\text{Find } \boldsymbol{x} = \begin{Bmatrix} x_1 \\ \vdots \\ x_n \end{Bmatrix} \text{ which minimizes } f(\boldsymbol{x}) \\ \text{Subject to } \begin{cases} g_i(\boldsymbol{x}) \leq 0, i = 1,2,\ldots,m \\ h_j(\boldsymbol{x}) = 0, j = 1,2,\ldots,n \end{cases}\end{gathered} \tag{1}$$

Where $\boldsymbol{x}$ is the vector of design parameters and $f(\boldsymbol{x})$ is the objective function. The functions $g_i(\boldsymbol{x})$ and $h_j(\boldsymbol{x})$ are the inequality constraint function and the equality constraint function respectively and they define the constraints of the problem. For this reason it is called a *constrained optimization problem.*

2 Parametric design optimization

For parametric design optimization the paper discuss an approach based on the DoE and the RSM to improve the design and to carry out a fully parametric optimization process. The initial design is parametrized and the user decides which dimensions can be changed in which variation window of each parameter, in order to modify the shape of the structure during the optimization process. The second step is to setup the simulation sequence in order to investigate the mechanical behaviour of the structure and to extract the output parameters, such as: stresses, displacements or eigenfrequencies. Then the DoE generates a set of design points which represent possible combinations of the input variables. Each design point represents a specific shape of the structure and all of them must be solved in the simulation model. Once all the design points are solved and the outputs are extracted, the RSM allows to express the variation of each output parameter as explicit function to the variation of the input parameters. In this way it is possible to investigate the relationships between variation of the input and output parameters. The user can now understand the model behaviour and explore improvement possibilities for the optimization process. Moreover a sensitivity analysis is carried out in order to identify the most influent input parameters, reducing the optimization problem and improving the accuracy and efficiency of the RSM approach. Finally objectives and constraints are defined and the optimization algorithm is called to find the best design improvement, which satisfies goals and constraints.

3 TEO/TEH Valve geometry

The geometry in exam is the oxygen and hydrogen balancing nozzle TEO/TEH. This valve is situated in the upper stage ME of the launcher Ariane 5. This component is integrated inside the ELS, symmetrically positioned to the oxygen/hydrogen purge connector CPO/CPH and it provides longitudinal thrust to balance TCPO/TCPH. At the other side the TEO/TEH is connected to the EPC attachment bracket via a rigid rod. In the Fig. 1 is illustrated the complete system and the red mark put in evidence the valve geometry in exam.

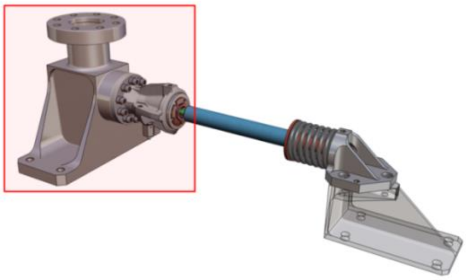


Figure 1 *TEO/TEH Complete system.*

The material used is the Aluminum 3.3214 and the proprieties are shown in the Tab.1. This material is heat-treatable aluminum alloy of medium strength, used especially in applications requiring good weld ability and corrosion resistance.

Material	**Temper**	**E(MPa)**	**G(MPa)**	α**(1/K)**	R_{p02}**(MPa)**	R_M**(MPa)**	ρ **(g/cm^3)**
Al	T6	63300	26200	2.28E-5	230	255	2.71

Table 1 *Material proprieties.*

3.1 Design constraints

The first step is to create the parametric model on the initial design, in order to change the shape of the structure during the optimization process. In the Fig. 2 the initial design of the Tex valve is illustrated.

Figure 2 *Initial design.*

Any geometry has to fulfil the follows design constraints:

- The diameter of the bore holes of the upper flange must be 8.3 *mm*.
- The edge distance (from the bore hole's outer diameter to the edge) must be at least 3.5 *mm* for the upper flange.
- The diameter of the bore holes of the flange towards the nozzle must be 5.2 *mm*.
- The edge distance (from the bore hole's outer diameter to the edge) must be at least 2.3 *mm* for the flange towards the nozzle.
- All the internal surface of the valve cannot be changed in order to respect the fluid dynamic requirements, see Fig. 3.

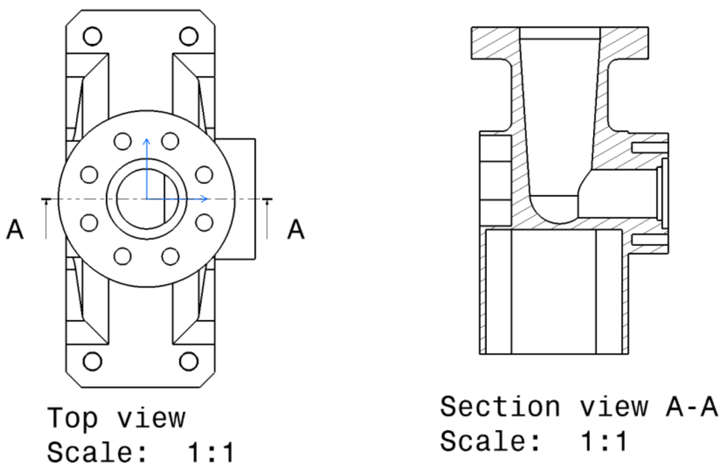


Figure 3 *Section of the valve.*

3.2 Input design parameters

The parametric model is made by using sketches and planes inside *ansys design modeler*. In this way all the dimensions generated, are automatically selectable as input parameters for the optimization process. In the following the input design variables are described.

- **Thickness upper flange**
  By increasing the parameter, the thickness of the upper flange decreases, see Fig. 4a.

- **External diameter upper flange**
  To reduce the mass, the value of the external diameter of the upper flange is reduced, see Fig 4b.

- **External diameter interface towards nozzle**
  See Fig. 4c.

- **Thickness base**
  See Fig. 4d.

- **Central pocket**
  The central part of the geometry is the only area where it is possible to create a pocket. By drawing the sketch shown in Fig. 4e, the length and the radius of the pocket are selected as input parameters in order to change the shape of the pocket during the optimization process.

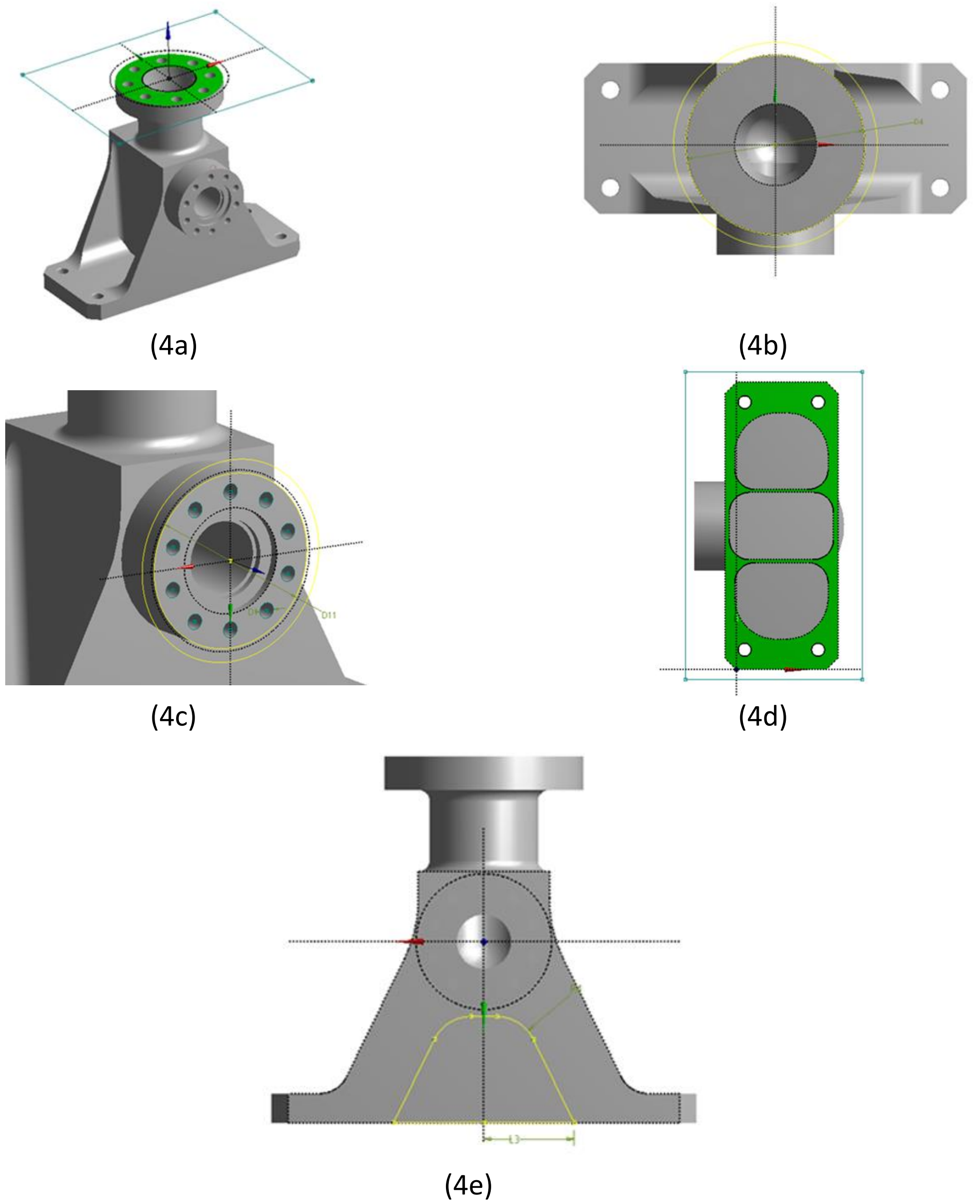


Figure 4 *Input design parameters.*

Tab. 2.shows the defined ranges if input parameter variation

Input Parameter	**Min Value [*mm*]**	**Max value [*mm*]**
Length Pocket	37	62
Radius Pocket	10	22
Diameter upper flange	80.5	88
Diameter flange towards nozzle	56	60
Cut material from upper flange	0.1	4
Cut material from base	0.1	5

Table 2 *Range input parameters.*

3.3 Simulation model

The structure is subjected to several forces and moments which are defined in their coordinate system as shown in Fig. 5. The definition of the load vector orientations leads to 64 possible load case combinations, according to the eq. (2).

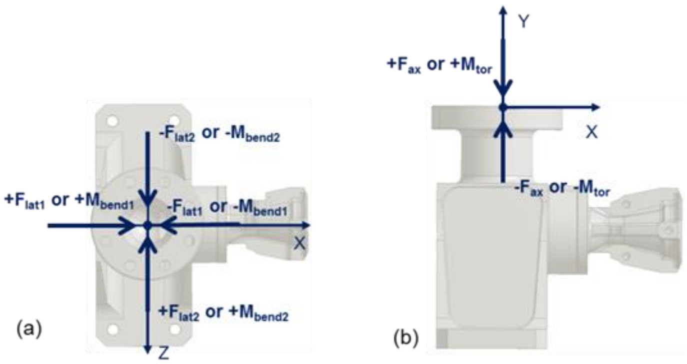


Figure 5 *Orientation of load vectors: a) lateral force or bending moment; b) axial force or torsional moment.*

$$\begin{bmatrix} F_y & F_z \\ +1 & 0 \\ -1 & 0 \\ 0 & +1 \\ 0 & -1 \end{bmatrix} \times \begin{bmatrix} F_x \\ +1 \\ -1 \end{bmatrix} \times \begin{bmatrix} M_y & M_z \\ +1 & 0 \\ -1 & 0 \\ 0 & +1 \\ 0 & -1 \end{bmatrix} \times \begin{bmatrix} M_x \\ +1 \\ -1 \end{bmatrix} \hat{=} 4 \times 2 \times 4 \times 2 = 64 \qquad (2)$$

Furthermore a pressure load applied on all the internal surface of the structure must be considered, see Fig. 6.

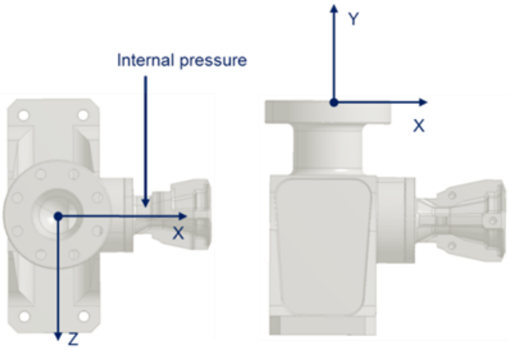


Figure 6 *Pressure load.*

The structure is constrained at its 4 interface points with fixed constraints towards the ELS. In the Fig. 7 the positions of the fixed constraints S1, S2, S3, S4 are illustrated.

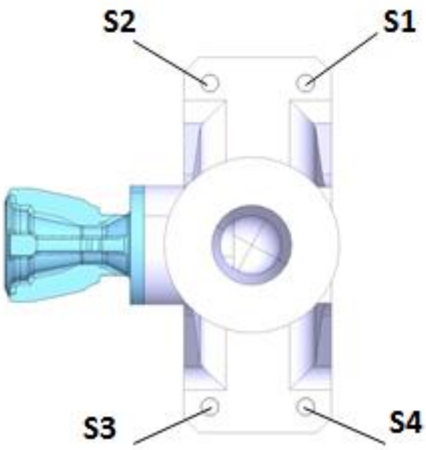


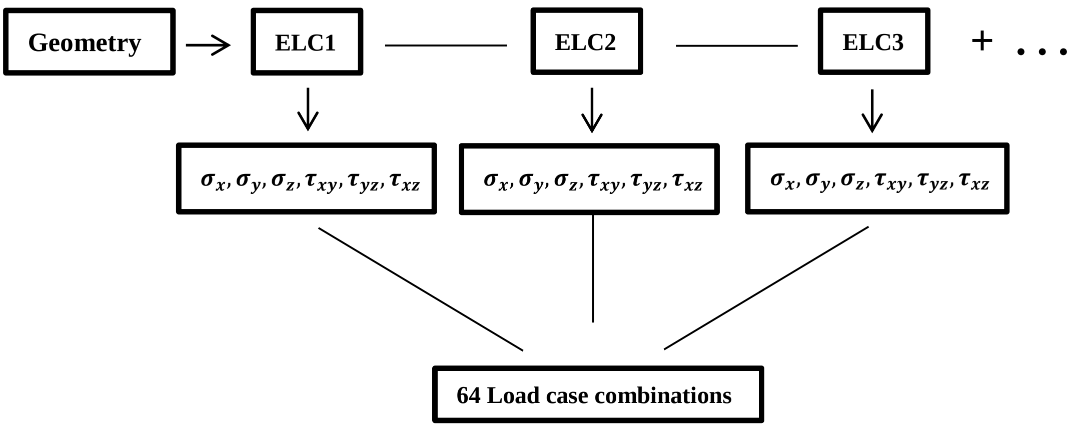
Figure 7 *Fixed constraints location.*

To reduce the computational time, 7 elementary load cases (ELCs) are solved and then all the 64 LCs are calculated from the post-processed results of the ELCs, by linear superposition of the nodal stress. Reversed loads of an ELC are obtained by scaling the results of the respective ELC with the factor of -1. In the Tab. 3 the load case combinations matrix is presented.

LCs	Pressure	FAX	FLAT1	FLAT2	MROT	MBEND1	MBEND2
LC1	1	1	1	0	1	1	0
LC2	1	-1	1	0	1	1	0
LC3	1	1	-1	0	1	1	0
:	:	:	:	:	:	:	:
LC64	1	-1	0	-1	-1	0	-1

Table 3 *Load case combinations.*

The V.M stress for all the nodes of the structure are calculated according to the specific LC combination. From each ELC the stresses (σ_x ,σ_y ,σ_z ,τ_{xy}, τ_{yz} ,τ_{xz}) are extracted, as illustrated below.



For each stress, a specific identification name is given:

$$\left(\sigma_{xi}\,, \sigma_{yi}\,, \sigma_{zi}\,, \tau_{xyi}\,, \tau_{yzi}\,, \tau_{xzi}\,\right) \qquad for\ i = 1,2,3, \dots n$$

Where *n* is the number of ELC.

Once the specific identification name is assigned, the stresses are combined with respect to the specific load case combination and the equivalent V.M stress is calculated by using the eq. (3).

$$\sigma_{V.M} = \sqrt{\frac{1}{2} \cdot \left[\left(\sigma_x - \sigma_y\right)^2 + \left(\sigma_y - \sigma_z\right)^2 + (\sigma_z - \sigma_x)^2 + 6 \cdot \left(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2\right)\right]} \tag{3}$$

As an example, the first load case combination LC1 = 1110110 is considered, which means "ELC1+ELC2+ELC3+ELC5+ELC6". By inserting the eq. (4) in *ansys WB*, it is possible to calculate the V.M stress for all the nodes of the structure.

$$\sigma_{V.M} = \sqrt{\frac{1}{2}\cdot\begin{bmatrix} \left((\sigma_{x1}+\sigma_{x2}+\sigma_{x3}+\sigma_{x5}+\sigma_{x6}) - (\sigma_{y1}+\sigma_{y2}+\sigma_{y3}+\sigma_{y5}+\sigma_{y6})\right)^2 + \cdots \\ \left((\sigma_{y1}+\sigma_{y2}+\sigma_{y3}+\sigma_{y5}+\sigma_{y6}) - (\sigma_{z1}+\sigma_{z2}+\sigma_{z3}+\sigma_{z5}+\sigma_{z6})\right)^2 + \cdots \\ \left((\sigma_{z1}+\sigma_{z2}+\sigma_{z3}+\sigma_{z5}+\sigma_{z6}) - (\sigma_{x1}+\sigma_{x2}+\sigma_{x3}+\sigma_{x5}+\sigma_{x6})\right)^2 + \cdots \\ 6\cdot\left(\begin{matrix}(\tau_{xy1}+\tau_{xy2}+\tau_{xy3}+\tau_{xy5}+\tau_{xy6})^2 + (\tau_{yz1}+\tau_{yz2}+\tau_{yz3}+\tau_{yz5}+\tau_{yz6})^2 + \cdots \\ (\tau_{xz1}+\tau_{xz2}+\tau_{xz3}+\tau_{xz5}+\tau_{xz6})^2\end{matrix}\right)\end{bmatrix}} \tag{4}$$

Obviously the equation changes for each of the 64 LC. Once all the combinations are computed, it is necessary to assign a specific identification name for every LC:

$$LC_i \qquad for\ i = 1,2,3,\dots n$$

Where n is the number of LC.

Finally by using the following expression it is possible to extract the worst LC:

$$\max(LC_1, LC_2 LC_3, \dots, LC_{64})$$

Finally the maximum V.M stress for the worst LC is selected as an output parameter.
For each design point the following operations are performed:

- The ELCs are solved and the stresses resulting are extracted.
- The 64 LC combinations are calculated from the post-processed results.
- The worst LC combination is considered and the maximum V.M stress is selected as an output parameter.

Furthermore, in the simulation model, the modal analysis in clamped configuration must be performed in order to calculate the first eigenfrequency in the range of 0 to 2000 *Hz*. By performing these operations, the user can investigate the following three output parameters during the optimization process:

- Mass value
- The max V.M stress for the worst LC
- The first eigenfrequency

The optimization aims to reduce the mass of the structure as much as possible while keeping the maximum V.M stress under the yield value and the first eigenfrequency over a specific value. Goals and constraints for the optimization process are shown in the Tab. 4.

Objectives and constraints		
Output Parameters	**Objective**	**Constraint**
Mass Geometry	Reduce as much as possible	No constraint
Max V.M stress	No objective	< 255 MPa
1st Frequency	No objective	≥400 Hz

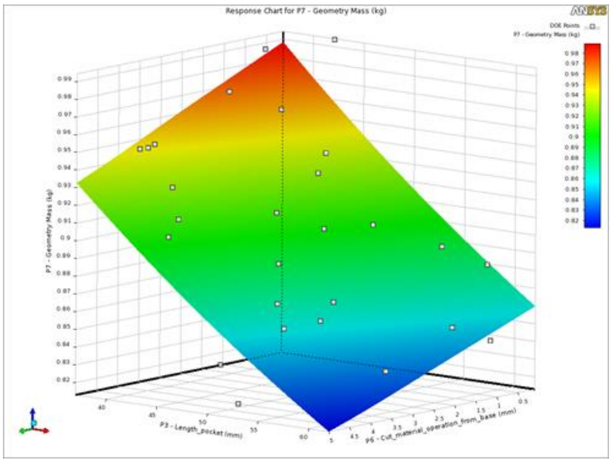
Table 4 *Goal and constraints.*

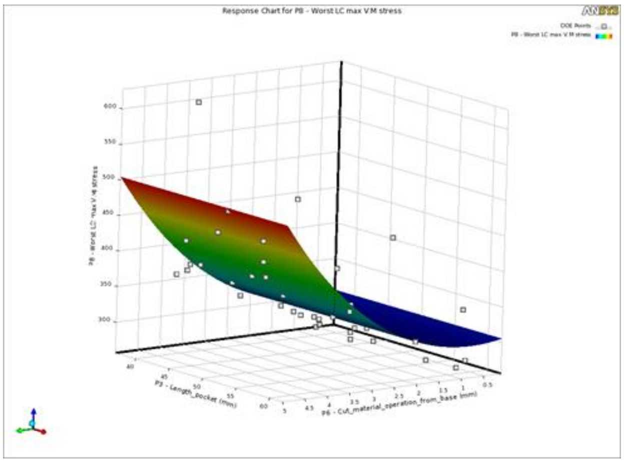
4 Optimization inside Ansys WB DesignXplorer

Ansys Wb DesignXplorer is based on DoE and the RSM. This together with various optimization methods helps to develop an optimized structure based on selected input and output parameters. The first step of the optimization is to create the DoE, which is a technique used to scientifically determine the location of sampling points. There are a wide range of DoE methods available in engineering literature, but these techniques all have one common characteristic: they try to locate the sampling points such that the space of random input parameters is explored in the most efficient way, or obtain the required information with a minimum of sampling points. In this optimization, in order to scan the design space, from 6 input parameters, 45 design points are generated by using the central composite design method (CCD). All the design points must be solved in the simulation model and the output parameters are extracted.

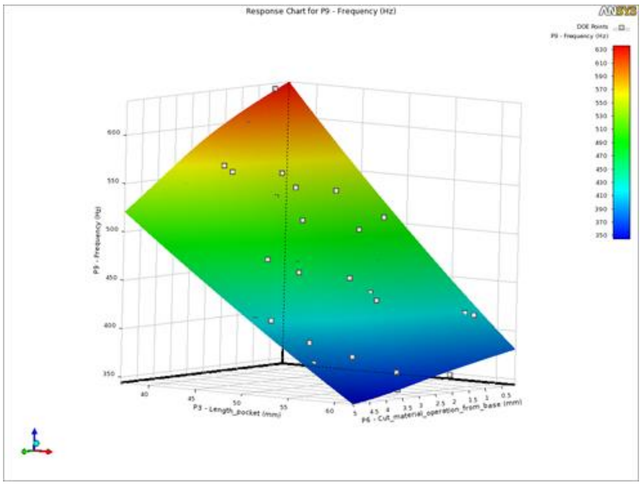
4.1 Sensitivity analysis inside Ansys WB DesignXplorer

Once all the design points are computed in the simulation model, the variation of output parameters are described in terms of the input parameters. Thanks to the RSM, all the response surfaces are built from the DoE in order to quickly provide the approximated values of the output parameter variation, without having to perform other finite element analysis. Obviously the accuracy of the response surface depends on the complexity of the variations of the solution and on the number of the design points. Once all the response surfaces are created, it is possible to look at the graphical representation, which allows to see how changes to each input parameter affect a selected output. In the Fig. 8 is illustrated the three-dimensional graphic which allows to view how the variations of two inputs impact on all the output parameters.





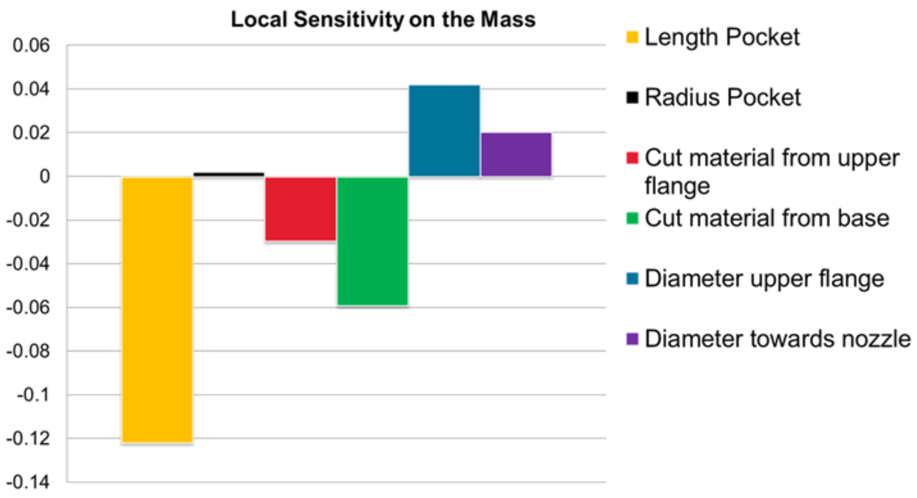
(8a) (8b)



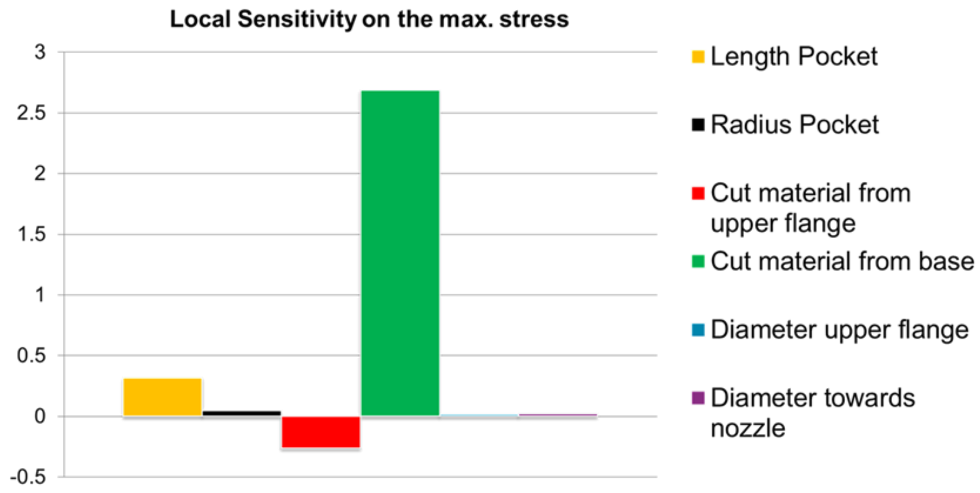
(8c)

Figure 8 Response surface mass (a); response surface V.M stress (b); response surface 1st frequency (c).

The relationships between the inputs and the outputs are investigated in order to understand the model behaviour and explore all the improvement possibilities for the optimization process. For example by looking at Fig. 8, it is understandable that the length of the pocket really influences the mass value and the 1st eigenfrequency. On the other hand, the operation of cut material from the base is really influential on the maximum V.M stress. Once all the response surfaces are generated, the software provides several tools for the sensitivity analysis. The sensitivity analysis allows understanding which input parameters are the most influential and which are not. This is really important in order to reduce the problem, to improve the response surface accuracy and to reduce the computational time. The local sensitivity analysis of the input parameters on the outputs is illustrated in the Fig. 9.



(9a)



(9b)

Figure 9 Local sensitivity analysis.

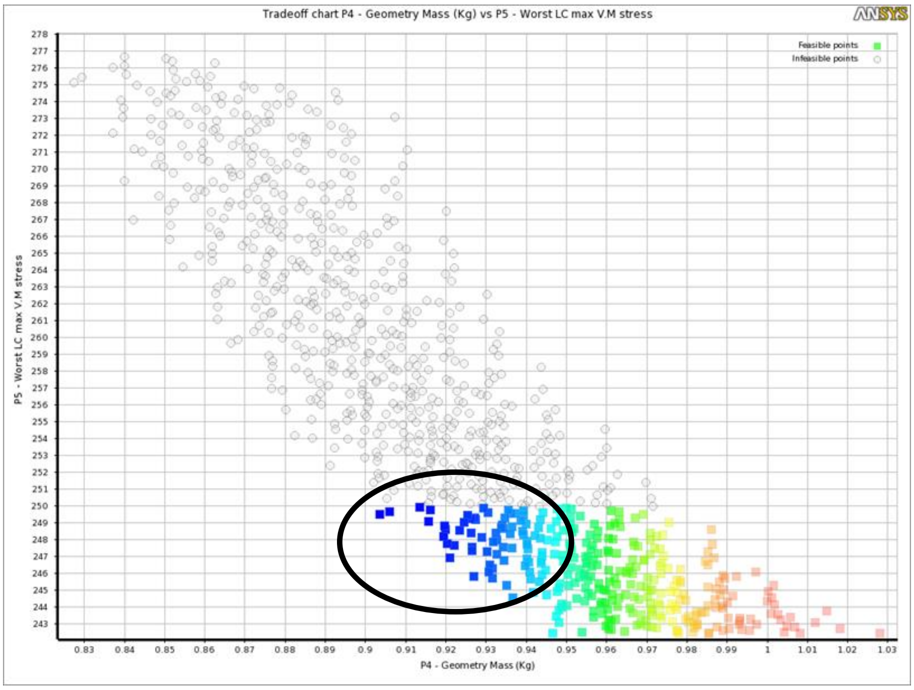
When looking at the graph in the Fig. 9a it is possible to see that the most influential input parameter on the mass value is the length of the pocket. When this input parameter increases, the mass value decreases really fast. Moreover it is understandable that the radius of the pocket and the external diameter towards the nozzle do not influence the mass value. On the other hand the graph in the Fig. 9b shows that the most influential input parameter on the maximum V.M stress is the operation “cut material from base”. This means that even a slight reduce of the thickness of the base, will result in high values of stress.

4.2 Optimization

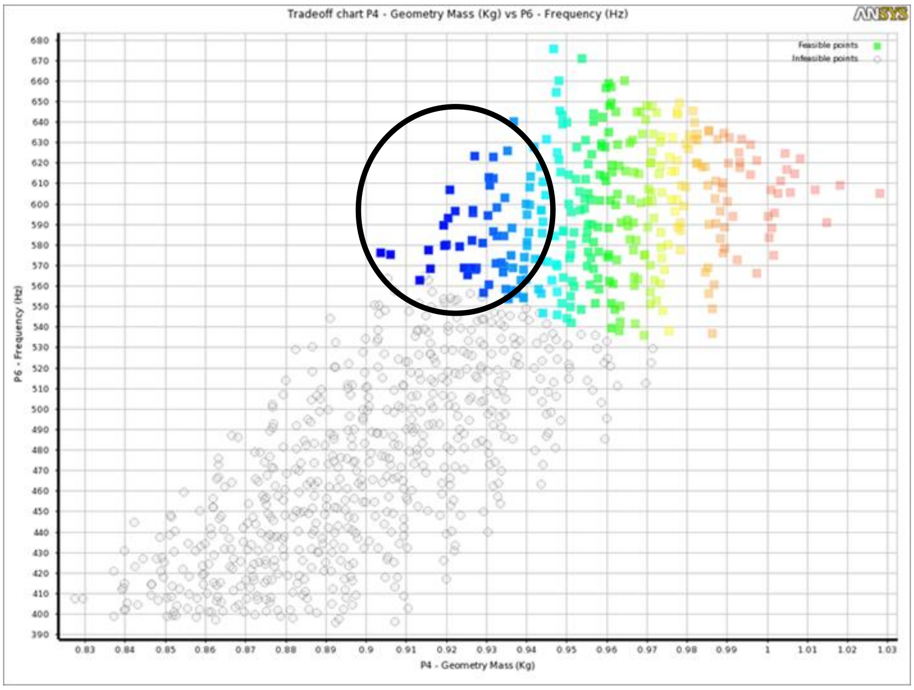
The optimization is performed by considering only the most important input parameters. The sensitivity analysis described above shows that it is possible to reduce the number of input parameters from 6 to 4. Another important consideration is that the operation “cut material from base” increases the maximum V.M stress too much. To keep the stress as low as possible, even this input parameter is removed and the input parameters considered for the optimization are the following:

- Pocket length
- Cut material from upper flange
- Diameter upper flange

In order to find the best design, the kind of optimization algorithm must be manually chosen in *ansys designXplorer*. The NLPQL (Nonlinear Programming by Quadratic Lagrangian) is the most appropriate method when the optimization problem consists of continuous input parameters and one objective function. Once the optimization is performed the software allows checking all the generated samples (see Fig. 10). The colours applied to the points represent the goodness of the samples, from red (the worst) to blue (the best points). The samples which do not respect the fixed constraints are represented without a colour. In Fig. 10a for each sample the mass value and the max V.M stress are shown. The area of interest is put in evidence by a black circle. The same is done in the Fig. 10b by comparing the mass value with the first eigenfrequency.



(10a)



(10b)

Figure 10 a)Trade-off mass vs. worst LC b)Trade-off mass vs. frequency mode1.

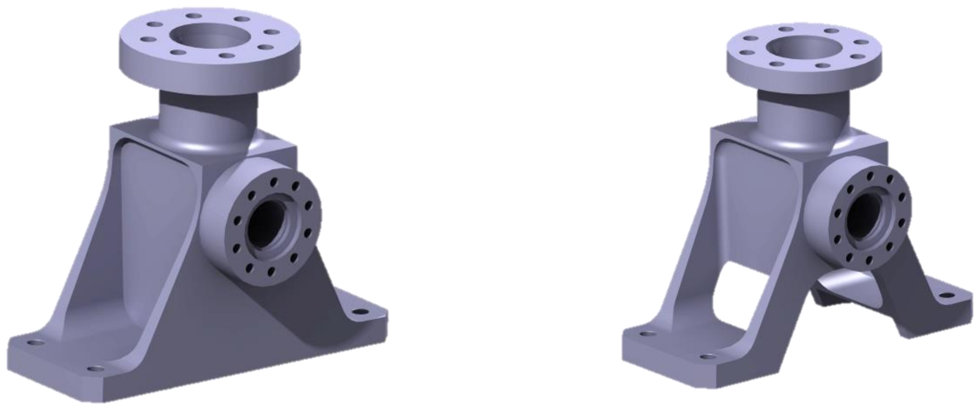
The software provides 3 candidate points with the predicted values of the outputs and the mass percentage reduction (the initial value of mass is 1.155 Kg). The user can choose the best candidate point and verify it in the simulation model. In the Tab. 5 the 3 candidate points are presented. Obviously the best design is the candidate point 1 because it presents the greatest mass reduction, respecting all the constraints.

	Candidate Point 1	**Candidate Point 2**	**Candidate Point 3**
Input parameters			
Length Pocket [mm]	**44.328**	**44.694**	**43.711**
Cut material upper flange [mm]	**3.846**	**2.7228**	**3.6354**
Diameter flange [mm]	**80.614**	**80.658**	**82.899**
Output parameters			
Max V.M stress (MPa)	**249.5**	**249.9**	**248.88**
Frequency Mode 1 (Hz)	**576.68**	**563.26**	**580.06**
Mass valve geometry (Kg)	**0.9037**	**0.913**	**0.919**
Mass reduction	**-21.75%**	**-20.95%**	**-20.43%**

Table 5 Candidate points.

4.3 Best design in ansys

The candidate point 1 is manually verified in the simulation model. In Fig. 11 the optimum valve geometry is compared to the initial design.



Initial Design **Final Design**

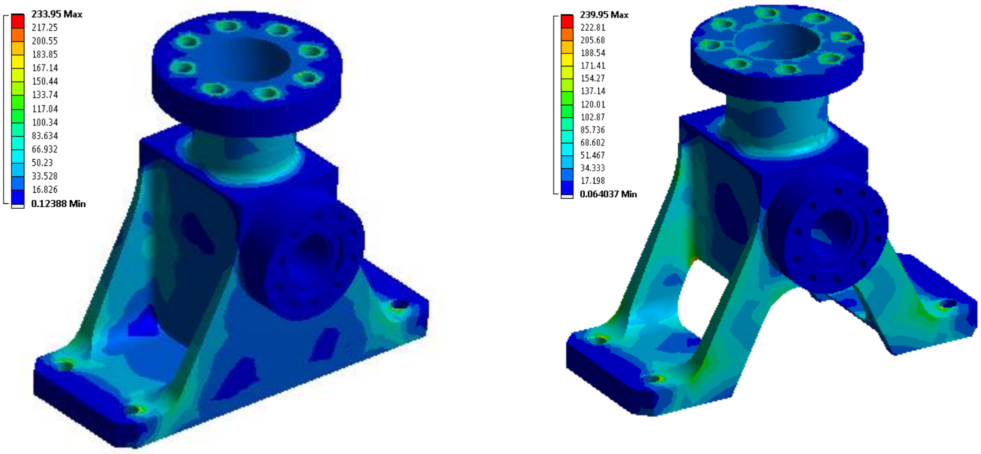
Figure 11 Initial design and final design.

By looking at the optimized geometry, one can see how the valve presents a reduced thickness and diameter of the upper flange. Furthermore the pocket created in the central area has the right length so the value of the first eigenfrequency does not decrease too much. In Tab. 6 the input and output values before and after the optimization are shown.

		Basic Design	**Optimum Design**	
Inputs	Diameter upper flange	88 *mm*	80.61 *mm*	
	Length pocket	*no pocket*	44.32 *mm*	
	Radius pocket	*no pocket*	20 *mm*	
	Diameter flange towards nozzle	60 *mm*	57 *mm*	
	Cut material from upper flange	*no*	3.84 *mm*	
Outputs	Total Mass	1.155 *Kg*	0.903 *Kg*	**21.8 % (-)**
	Maximum stress	233.9 *MPa*	239.9 *MPa*	**2.56 % (+)**
	1^{st} Frequency	683 *Hz*	576.45 *Hz*	**15.6 % (-)**

Table 6 Results optimization in *ansys*.

Looking at the table above it is possible to see that the mass reduction achieved is around 22%. Furthermore by looking at the other output parameters, it is possible to notice that the maximum V.M stress for the worst LC increases, while remaining lower than 255 *MPa*. Compared to the initial value the 1^{st} eigenfrequency decreases but is still over 400 *Hz*. The final design shows that it is possible to meet all the mechanical requirements, reducing significantly the weight of the structure. In Fig. 12 it is possible to compare the equivalent stress distribution for the worst LC between the basic geometry and the optimum design.



Initial Design **Final Design**

Figure 12 Comparison of the equivalent stress distribution, initial vs. optimum design

However, the design optimization process (performed completely inside *ansys WB*) presents several limitations:

- The optimization process is not entirely automatic
- The number of input parameters must be reduced manually
- The best approximation model and the related DOE schemata to generate the response surfaces must be chosen by the user
- The optimization algorithm to find the best design must be chosen by the user

These limitation become significant if the number of parameter to be optimized rise. In this case only 6 parameter are varied, but if it is necessary to optimize a much more complex geometry and consider a large amount of design variables, *a user would like to have a kind of automatic procedure to identify important parameter and reduce the optimization problem*.

5 Optimization in optiSLang

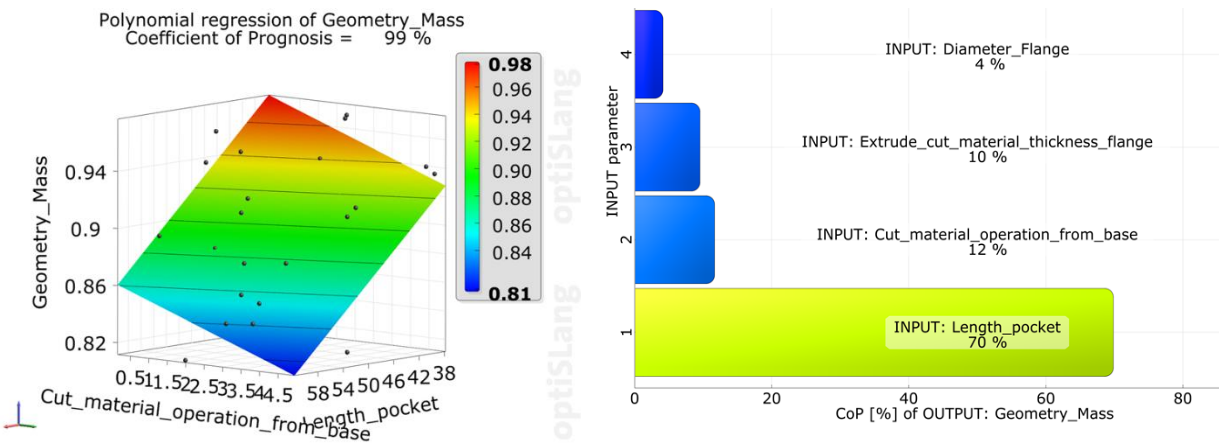
OptiSLang is an general purpose software for sensitivity analysis, multiobjective and multidisciplinary optimization, robustness evaluation, reliability analysis and robust design optimization. This software automatically identifies the relevant input and output parameters and quantifies the forecast quality with the help of the Coefficient of Prognosis (CoP) and the Metamodel of Optimal Prognosis (MoP) workflow. To achieve an efficient optimization, a predictable prognosis quality is really important. With the availability of an automatic parameter reduction the philosophy allows “no run too much” in order to minimize solver calls. Furthermore *optiSLang* automatically selects the appropriate algorithms for the optimization and supports the interfacing to almost any software tool which is used in virtual product development.

5.1 Sensitivity analysis in optiSLang

OptiSLang is connected to the simulation model created in *ansys workbench*, where the following steps have already been performed:

- Parametric Model.
- Definition of input parameters.
- Simulation Model.
- Definition of output parameters.

After dropping the sensitivity wizard on the project page OptiSLang automatically shows all parameters defined in *ansys*. The user can define the optimization problem by assigning the specific range for each input parameter and goals and constraints for the outputs, respectively according to the Tab. 2 and Tab. 4. The first and most important step for a successful and efficient optimization procedure is to analyze the global sensitivities of the design parameters of the initial design. By performing an optimized Latin hypercube sampling (LHS) with N=45 design points, the design space is scanned. Once all the design points are computed, thanks to the Coefficient of Prognosis measure, the Metamodel of Optimal Prognosis detects for each specified solver response, the optimal approximation model using the optimal subspace of important variables. The software shows directly for each output parameter only the most influential design variables. In the following, for each output parameter, the optimal approximation model and the most significant input parameters are shown (see Fig. 13).



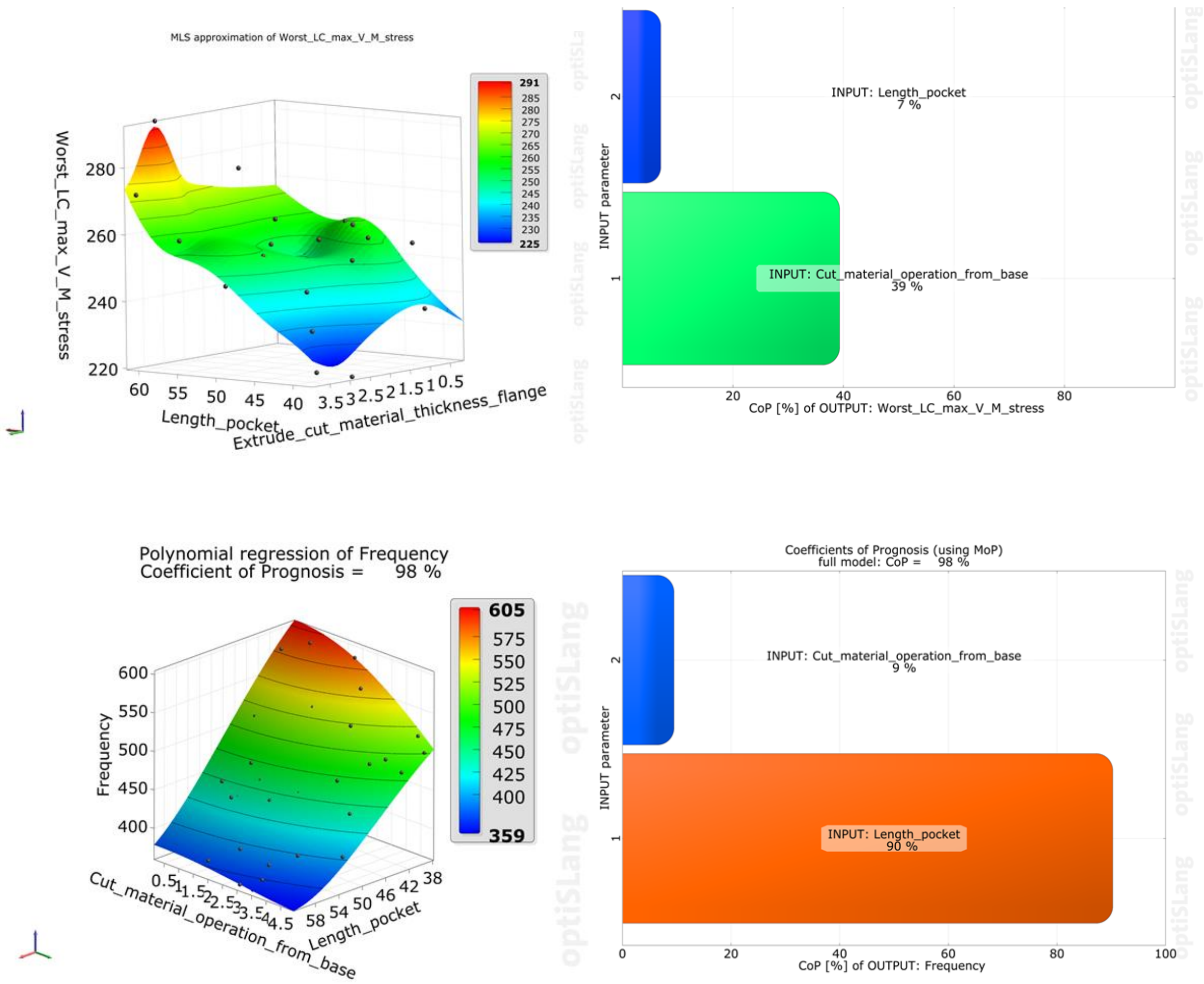
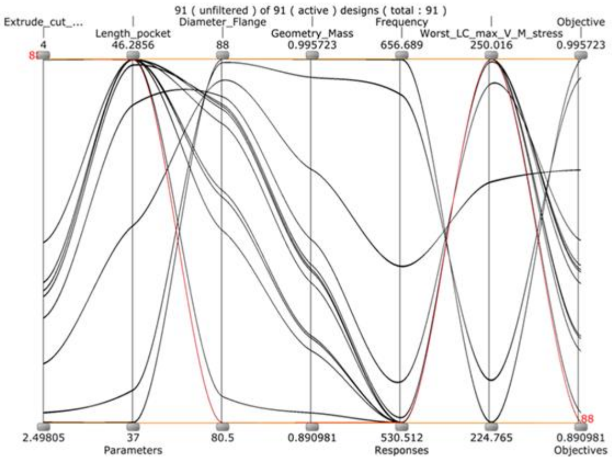


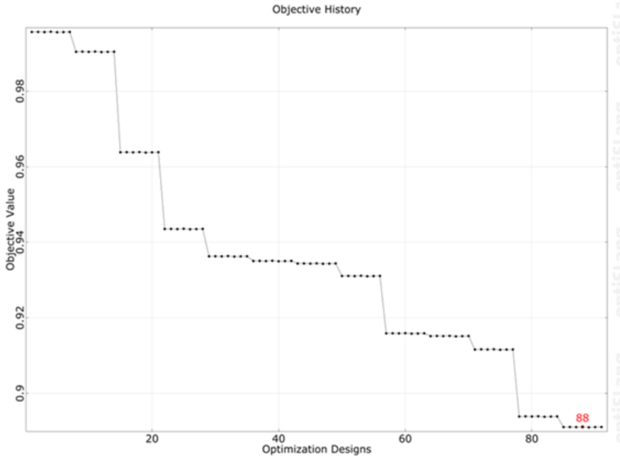
Figure 13 Most influent input parameters for each output.

Looking at the graphs above, it is interesting to note that the length of the pocket is at the same time the most influent input parameter on the mass and the 1st frequency reduction. This means that the optimization will be the best compromise between goal and constraints.

5.2 Optimization

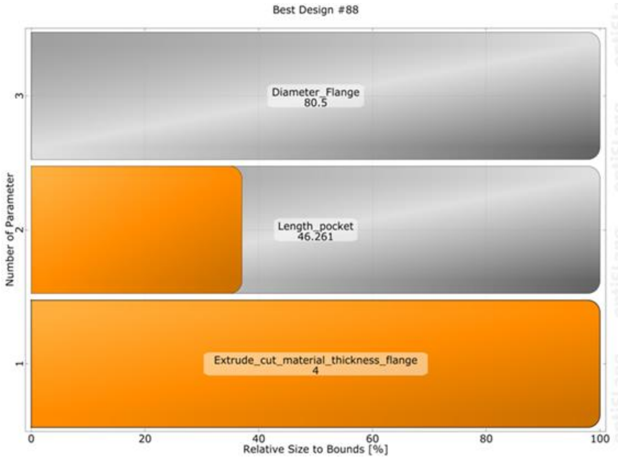
Using a optimization wizard according to the specific optimization problem, *OptiSLang* automatically suggests the most appropriate optimization algorithm in order to find the best design, which satisfies goals and constraints. The most appropriate optimization algorithm suggested here is the NLPQL. Obviously the quality of results depends on the accuracy of the approximation, which is influenced by the number of design points and the kind of approximation functions used to generate all response surfaces. The algorithm converges after N= 91 design evaluations which are represented in the Fig. 14a, where the red line indicates the best design. *OptiSLang* provides the objective history, where it is possible to check the position of the best design during the optimization process (see Fig. 14b). The best design (#88), with its input parameters, is shown in Fig. 14c with the associated responses of Fig. 14d.

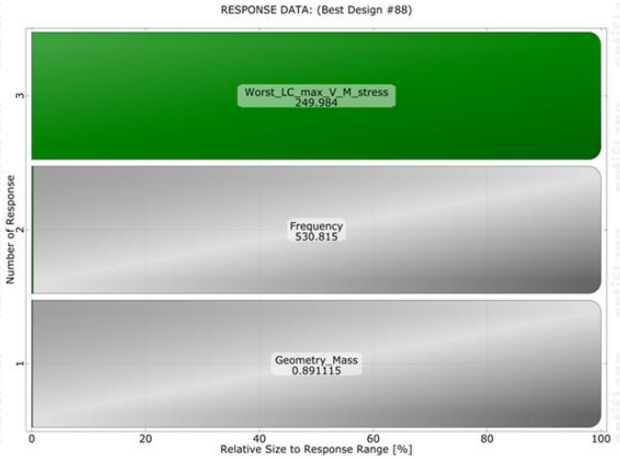




(14a)

(14b)





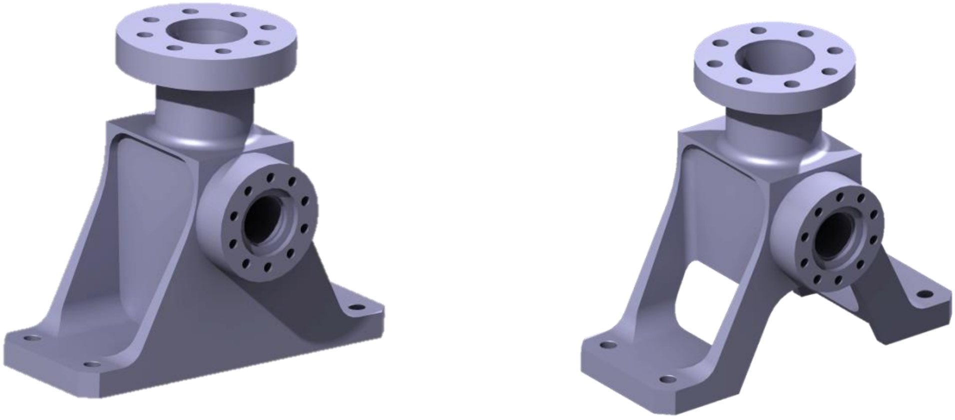
(14c)

(14d)

Figure 14 (a) Best design and all the samples generated; (b) Objective history; (c) Input parameters best design; (d) Predicted output parameters best design.

5.3 Best design in optiSLang

The best design is automatically verified in the simulation model. In Fig. 15, the optimum design is shown and compared to the basic geometry.



Initial Design **Final Design**

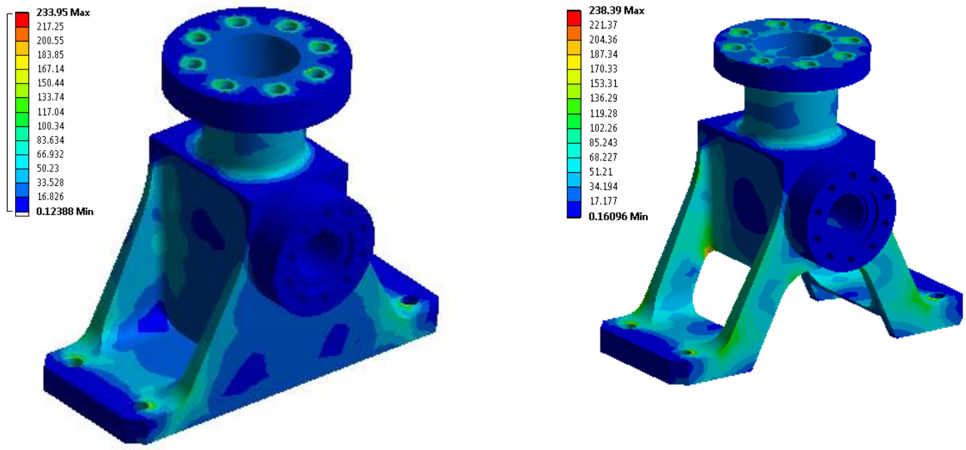
Figure 15 Initial design and final design.

The optimization in *optiSLang* provides a final design which presents the minimum value for the thickness and the diameter of the flange according to the design constraints. Furthermore the length of the pocket is bigger than the previously obtained value in *ansys*. In Tab. 7 the geometrical characteristics and the mechanical performances of the optimum design are compared to the basic geometry values. Moreover the percentage decreases or increases of the output parameters are shown.

		Basic Design	**Optimum Design**	
Inputs	Diameter upper flange	88 *mm*	80.5 *mm*	
	Length pocket	*no pocket*	46.26 *mm*	
	Radius pocket	*no pocket*	20 *mm*	
	Diameter flange towards nozzle	60 *mm*	57 *mm*	
	Cut material from upper flange	*no*	4 *mm*	
Outputs				
	Total Mass	1.155 *Kg*	0.89 *Kg*	**22.9 % (-)**
	Maximum stress	233.9 *MPa*	238.3 *MPa*	**1.88 % (+)**
	1st Frequency	683 *Hz*	554.48 *Hz*	**18.81 % (-)**

Table 7 Results optimization in *optiSLang*.

The table above shows that *optiSLang* allows to obtain a mass reduction of around 23 %. In this case the final geometry also has a bigger value of stress and a lower value of the 1st frequency, but all the outputs satisfy the constraints. The optimum design provided by *optiSlang* is slightly better than the previous obtained in *ansys*. Furthermore *optiSlang* allows working with more much more than the investigated 6 input parameters without changing the process. The optimization loop in *optiSLang* is almost entirely automatic, because the software automatically reduces the problem, chooses the best approximation model in order to build the response surfaces and suggest the most appropriate optimization algorithm. Due to these reasons, performing the optimization in *optiSLang* was choosen to be the best option available in order to optimize any kind of structural component. In the Fig. 16 the equivalent stress distribution for the worst LC, before and after the optimization is compared.



Initial Design **Final Design**

Figure 16 Comparison of the equivalent stress distribution, initial vs. optimum design

6 Validation

The validation aims to demonstrate that it is possible to optimize even much more complex geometries with a large amount of load cases. The complex geometry in exam is the "pressurization and degassing plate for the Hydrogen tank" PPDRH. Optimization goal and constraints are the same described previously, see Tab. 4. The geometry presents 5 external mechanical interfaces and the simulation model consists of 320 load case combinations plus the modal analysis in clamped configuration.

6.1 Input parameters

The parametric model is made inside *ansys design modeler*. In this case the choice of the input parameters is limited because the geometry presents a lot of design constraints. First of all where it is possible, several pockets are generated in order to remove material, see Fig. 17.

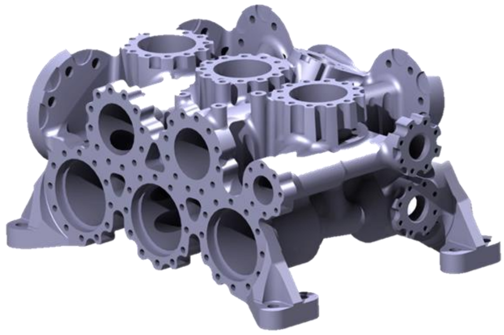


Figure 17 Pockets created on the basic geometry.

One of the most interesting questions of the user is to know how much material can be saved from the supports and behind the external mechanical interfaces. For this reason for the optimization process, the following input parameters are considered, see Fig. 18.

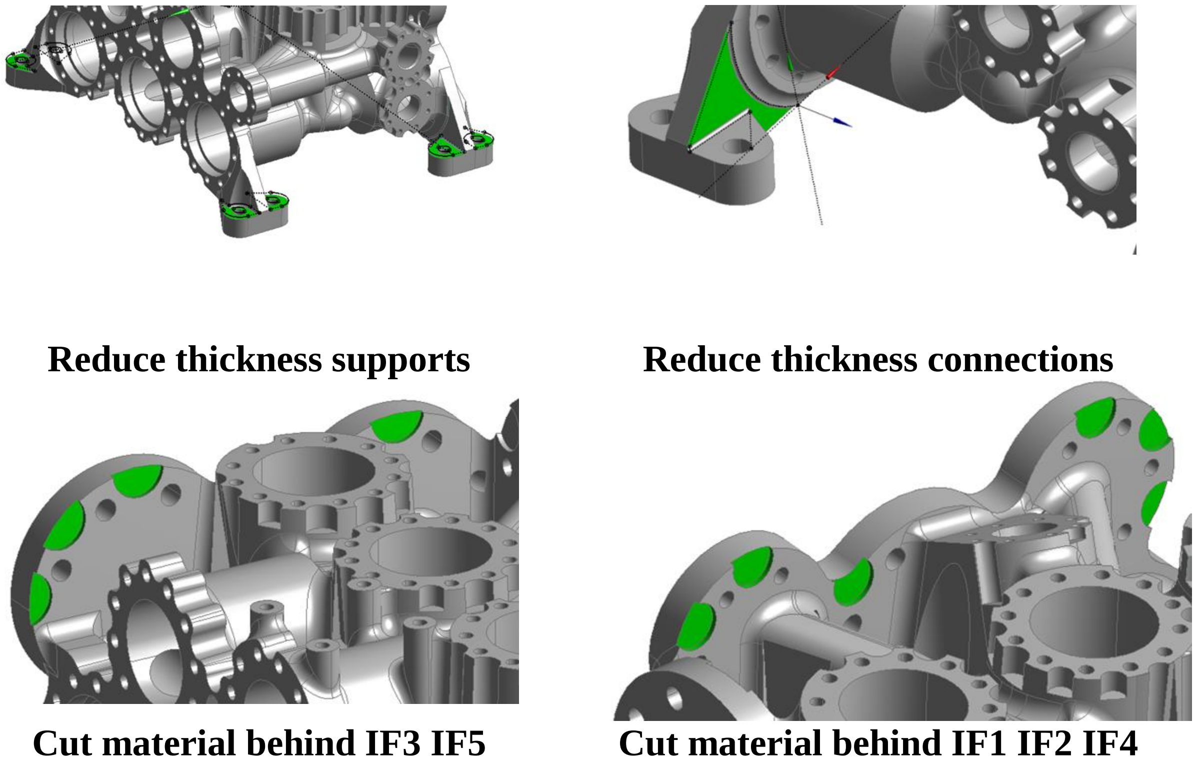


Figure 18 Input parameters.

The number of input parameters is 4. In the following the minimum and maximum values for each one are shown.

Input Parameter	Min Value [*mm*]	Max value [*mm*]
Cut material supports	1	12
Cut material connections	1	7
Cut material IF3 IF5	2	11
Cut material IF1 IF2 IF4	2	11

Table 8 Range input parameters.

6.2 Simulation model

In the validation the most difficult challenge is to create the simulation model, because for each design point the structure must be verified according to 320 load case combinations. To reduce the computational time, for each interface, 6 ELCs are solved and the results are combined in order to obtain all the load case combinations. In the Fig. 19 the work flow of the simulation model is shown.

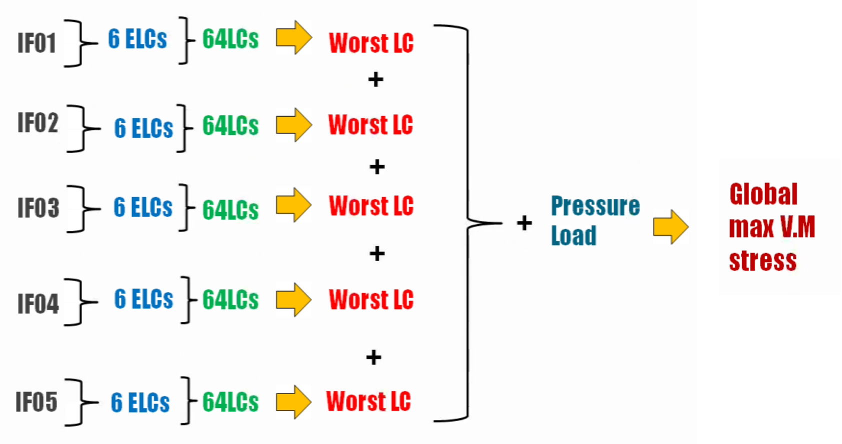


Figure 19 Load case combinations.

From each interface the stresses are extracted for all the ELCs:

$$(\sigma_{xij}, \sigma_{yij}, \sigma_{zij}, \tau_{xyij}, \tau_{yzij}, \tau_{xzij})$$

Where *i* is the number of interface and *j* is the number of ELC.

As an example, in the following the equation to calculate the V.M stress for the first interface and the first load case combination are shown.

LC1 1110110 ⇨ ELC1 + ELC2 + ELC3 + ELC5 + ELC6

⇩

$$\sigma_{V.M} = \sqrt{\frac{1}{2}\cdot\left[\begin{array}{c} \left((\sigma_{x11}+\sigma_{x12}+\sigma_{x13}+\sigma_{x15}+\sigma_{x16})-(\sigma_{y11}+\sigma_{y12}+\sigma_{y13}+\sigma_{y15}+\sigma_{y16})\right)^2+\cdots \\ \left((\sigma_{y11}+\sigma_{y12}+\sigma_{y13}+\sigma_{y15}+\sigma_{y16})-(\sigma_{z11}+\sigma_{z12}+\sigma_{z13}+\sigma_{z15}+\sigma_{z16})\right)^2+\cdots \\ \left((\sigma_{z11}+\sigma_{z12}+\sigma_{z13}+\sigma_{z15}+\sigma_{z16})-(\sigma_{x11}+\sigma_{x12}+\sigma_{x13}+\sigma_{x15}+\sigma_{x16})\right)^2+\cdots \\ 6\cdot\left(\begin{array}{c}(\tau_{xy11}+\tau_{xy12}+\tau_{xy13}+\tau_{xy15}+\tau_{xy16})^2+(\tau_{yz11}+\tau_{yz12}+\tau_{yz13}+\tau_{yz15}+\tau_{yz16})^2+\cdots \\ (\tau_{xz11}+\tau_{xz12}+\tau_{xz13}+\tau_{xz15}+\tau_{xz16})^2\end{array}\right)\end{array}\right]}$$

Once all the LCs are calculated, for each interface the worst LCs are extracted and added up. The global worst LC is obtained and the maximum V.M stress can be selected as output parameter. Furthermore the modal analysis in clamped configuration must be performed in order to calculate the 1st eigenfrequency in the range of 0 to 2000 Hz.

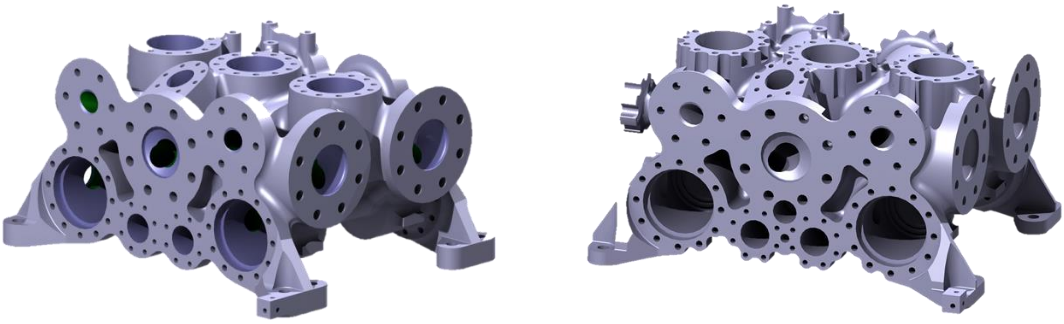
6.3 Final design

By using the LHS method, 25 design points are generated and all of them are computed in the simulation model. Once the DoE is solved, optiSLang carries out the sensitivity analysis and generates all the response surfaces thanks to the MoP. OptiSLang suggests the most appropriate algorithm for the optimization. The NLPQL is suggested because the number of inputs is low, the variables are continuous and the optimization problem presents one objective function. The algorithm generates 154 designs and the best one is found. The final design is verified and in the following the output parameters are presented and compared to the initial design.

		Initial Design	**Optimum Design**	
Inputs	Cut material IF5 IF3	no	6.97 *mm*	
	Cut material IF1 IF2 IF4	no	11 *mm*	
	Reduce thickness supports	no	3.96 *mm*	
	Reduce thickness connections	no	7 *mm*	
Outputs				
	Total Mass	7.057 *Kg*	6.5803 *Kg*	**6.75 % (-)**
	Maximum stress	239 *MPa*	238 *MPa*	**0.4 % (-)**
	1st Frequency	1150.7 *Hz*	1064.1 *Hz*	**7.52 % (-)**

Table 9 Results optimization.

By looking at the table above it is possible to see that all the constraints are respected and the mass value has been reduced by 6.75%. In the Fig. 20 the initial geometry and the final design are compared.



Initial Design **Final Design**

Figure 20 Initial design and final design, PPDRH.

The validation demonstrates that it is possible to perform the design optimization process even for much more complex geometry with a large amount of load cases. So the presented process can be considered as appropriate as standard procedure for optimization of structural components.

7 Conclusions

The aim of the thesis has been to develop an efficient design optimization process, in order to optimize any kind of structural component. The methodology chosen to perform the optimization has been the parametric approach based on the Design of Experiments and the Response Surface method. The parametric design optimization process has been successfully developed, implemented and validated. The design optimization has been applied to redesign a valve geometry with the objective to reduce the structural weight as much as possible. The optimization has been performed in two different ways and the solution *ansys+optiSLang* has been preferred to the optimization all inside *ansys WB*. The initial design has been optimized just using 3 input design variables and the mass has been significantly reduced by 23%. *OptiSLang* is safe to use, minimizes the user input, automatically reduces the problem and suggests the best optimization algorithm. The software allows working with more than 50 input parameters and this means that the same design optimization process can be applied in order to optimize much more complex geometries with a large amount of load cases. In conclusion the optimization process provided is efficient, flexible, suitable and allows to explore all the improvement possibilities in order to satisfy goals and constraints, in any kind of structural applications.

8 Outlook

The design parametric optimization process provided can be improved by using the parametric interface *ansys space claim direct modeler*, which allows to parametrize automatically any kind of basic geometry STEP file. This is really suitable when the basic geometry is too complex and when the user needs to use too many input design variables.
The optimization allows the user to perform a multi-objective optimization, thanks to the possibility to consider different kinds of analysis at the same time (such as: static structural analysis, modal analysis, fatigue analysis, thermal analysis, fluid dynamic analysis etc.). The power of this method is that the structural component can be improved in a multidisciplinary context, in order to obtain a product with high performances in several application fields.
Another important improvement could be to perform the robustness evaluation of the final design

Referenzen/Literaturangaben

D.C Montgomery. *Design and Analysis of Experiments.* John Wiley & Sons, NY,USA, 5th edition, 2001.

Robert L. Mason, Richard F. Gunst and James L. Hess. *Statistical Design and Analysis of Experiments – With Applications to Engineering and Science.* John Wiley & Sons, NY,USA, 2nd edition, 2003.

JOHANESS WILL & Thomas Most: Metamodell of Optimal Prognosis (MoP) – an Automatic Approach for User Friendly Parameter Optimization, 2009.

D. K. J. Lin & J. J. Peterson. *Statistical Inference for Response Surface Optima, Chapter 4 Statistical Inference for Response Surface,* University of Florida, USA, 2006.

A. E. Eiben & S. K. Smit. *Chapter 2 Evolutionary Algorithm Parameters and Methods to Tune Them*.

ANSYS. *Design Exploration User's Guide*.

OptiSLang manual.