

CUSTOMER STORY // AEROSPACE INDUSTRY

OPTIMIZATION OF VALVE GEOMETRIES IN THE ENGINE SYSTEM OF THE ARIANE 5

optiSLang is used to generate a transferable automatic and efficient RDO process to reduce the weight of casted valve nozzles as components in the upper stage Midlife Evolution (ME) of the launcher Ariane 5.



Fig. 1: Upper stage Midlife Evolution (ME) of the launcher Ariane 5

Introduction

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 as light as possible. In this article, a highly automated optimization process for the weight reduction of casted support structures will be presented. The structural component to be examined is the oxygen/hydrogen balancing nozzle (TEO/TEH) situated on the upper stage Midlife Evolution (ME) of the launcher Ariane 5 (see Fig.1).

The optimization is carried out using ANSYS Workbench as a solver and the software optiSLang for the sensitivity analysis and optimization. After finalizing the first optimization, the workflow is tested on a second structure.

Parametric design optimization

For a parametric design optimization, this article discusses an approach based on the Design of Experiments (DoE) and the Response Surface Method (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 the setup of the simulation sequence in order to investigate the mechanical behavior 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 pa-

parameter as an explicit function to the variation of the input parameters. In this way, it is possible to investigate the correlation between variation of the input and output parameters. The user can now understand the model behavior and explore improvement possibilities for the optimization process. Moreover, a sensitivity analysis is carried out in order to identify the most influential input parameters, to reduce the optimization problem and to improve the accuracy and efficiency of the RSM approach. Finally, objectives and constraints are defined and the optimization algorithm is chosen to find the best design improvement which satisfies goals and constraints.

TEO/TEH Valve geometry

The geometry in exam is the TEO/TEH. This valve is situated in the upper stage ME of the launcher Ariane 5. This component is integrated inside the Elongated Lower Skirt (ELS), symmetrically positioned to the oxygen/hydrogen purge connector (CPO/CPH) and it provides longitudinal thrust to balance the nozzle (TCPO/TCPH). At the other side, the TEO/TEH is connected to the Cryogenic propulsive stage (EPC) attachment bracket via a rigid rod. The complete system and the valve geometry in exam marked in red is illustrated in Fig. 2.

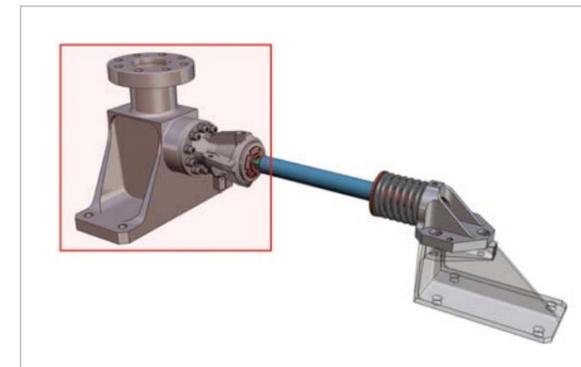


Fig. 2: TEO/TEH Complete system

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

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 Fig. 3, the initial design of the Tex valve is illustrated.



Fig. 3: Initial design

Input design parameters

The parametric model is generated inside ANSYS Design Modeler by using sketches and plans. In this way, all the generated dimensions 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 next page)
- External diameter upper flange
- Mass - for reduction the value of the external diameter of the upper flange is decreased (see Fig. 4b next page)
- External diameter interface towards nozzle (see Fig. 4c next page)
- Thickness base (see Fig. 4d next page)
- 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 (see next page), 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.

Simulation model

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

Furthermore, it must be considered that a pressure load has to be applied on all the internal surfaces of the structure (see Fig. 6 next page).

Material	Temper	E (MPa)	G (MPa)	α (1/K)	$R_{p0.2}$ (MPa)	R_M (MPa)	ρ (g/cm ³)
Al	T6	63300	26200	2.28E-5	230	255	2.71

Table 1: Material properties

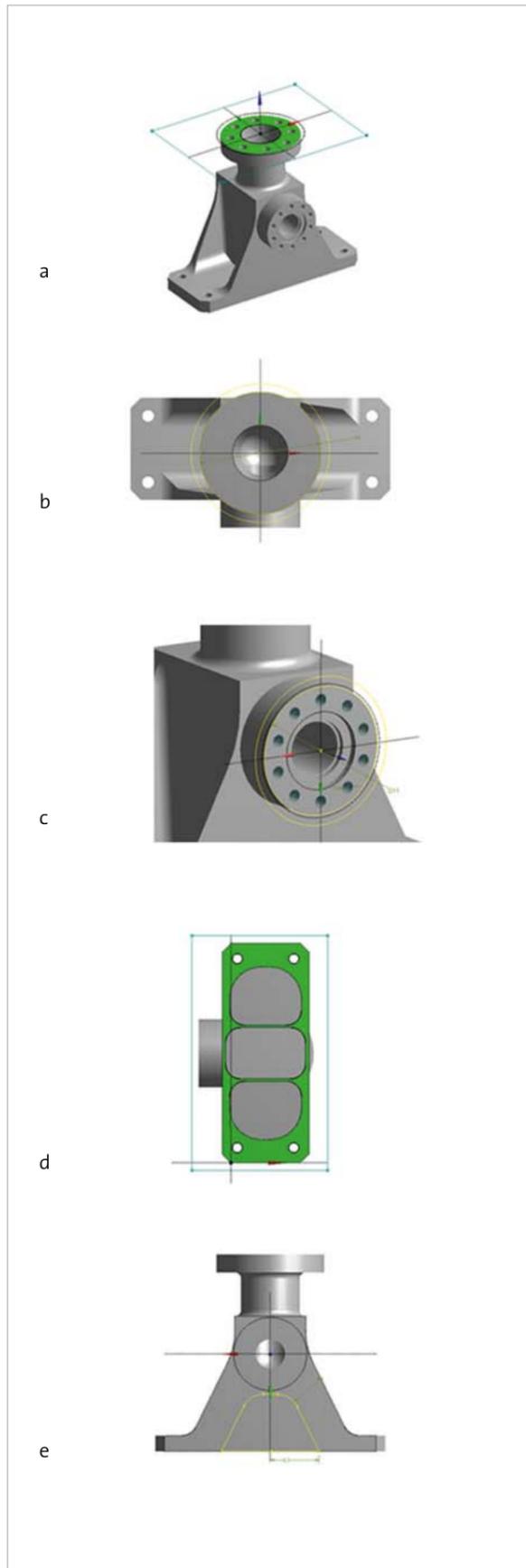


Fig. 4: Input design parameters

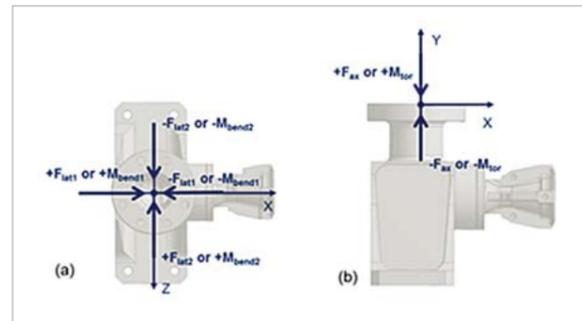


Fig. 5: Orientation of load vectors – (a) lateral force or bending moment / (b) axial force or torsional moment.

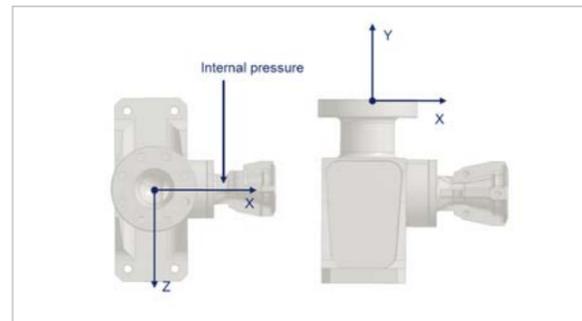


Fig. 6: Pressure load

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

Tab. 2: Defined ranges for input parameter variation

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

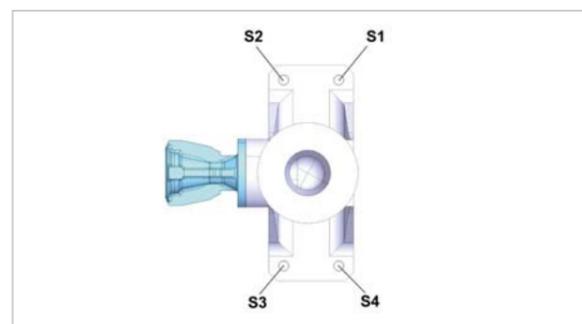


Fig. 7: Fixed constraints location

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 using linear superposition of the nodal stress.

The von Mises equivalent stress for all nodes of the structure are calculated according to the specific LC combination. Finally, the maximum stress for the worst LC 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
- Maximum stress for the worst LC
- First eigenfrequency

The optimization aims at reducing the mass of the structure as much as possible while keeping the maximum stress under 225 MPa and the first eigenfrequency over 400 Hz.

Optimization in optiSLang

optiSLang provides a workflow for the automatic identification of relevant input and output parameters and quantifies the forecast quality of the response surfaces with the help of the Coefficient of Prognosis (CoP) and the Metamodel of Optimal Prognosis (MOP) workflow. To achieve an efficient optimization and reliable parameter reduction, a predictable prognosis quality of the response surfaces is incredibly important. With the availability of an automatic parameter reduction, optiSLang allows a “no run too much” philosophy 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.

Sensitivity analysis in optiSLang

There is an integrated version of optiSLang inside ANSYS Workbench available 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 defines the optimization problem by assigning the specific range for each input parameter as well as goals and constraints for the outputs. 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 indicated by the Coefficient of Prognosis measure are computed, the Metamodel of Optimal Prognosis detects the optimal approximation model using the optimal subspace of important variables for each specified solver response. The software directly shows only the most influential design variables for each output parameter. In the following, for each output parameter, the optimal approximation model and the most significant input parameters are identified (see Fig. 8 next page).

Looking at the graphs in Fig. 8 (see next page), it is interesting to note that the length of the pocket is at the same time the most influential input parameter regarding the mass and the first frequency reduction. This means that the optimization will be the best compromise between goal and constraints.

Optimization

Using an optimization wizard, optiSLang automatically suggests the most appropriate optimization algorithm in order to find the best design which satisfies goals and constraints. Here, the NLPQL is suggested as the most appropriate optimization algorithm. The quality of results obviously depends on the accuracy of the approximation which is influenced by the number of design points and the approximation functions used to generate all response surfaces. The algorithm converges after N= 91 design evaluations. The best design (#88) with its input parameters is shown in Fig. 9 left (see next page) with the associated responses shown in Fig. 9 right (see next page).

The best design is automatically verified in the ANSYS simulation model. In Fig. 10 (see next page), the optimum design is shown and compared to the basic geometry.

The optimization provides a final design which presents the minimum value for the thickness and the diameter of the flange according to the design constraints. In Tab. 3 (see next page), the geometrical characteristics and the mechanical performances of the optimum design are compared to the basic geometry values. The percentage of decreases or increases of the output parameters are also shown.

It is proved 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 first frequency. However, all outputs satisfy the constraints. Furthermore, optiSLang allows working with much more than the investigated 6 input parameters without changing the process. The optimization loop in optiSLang is highly automated. The software independently reduces the optimization problem by choosing the best approximation model in

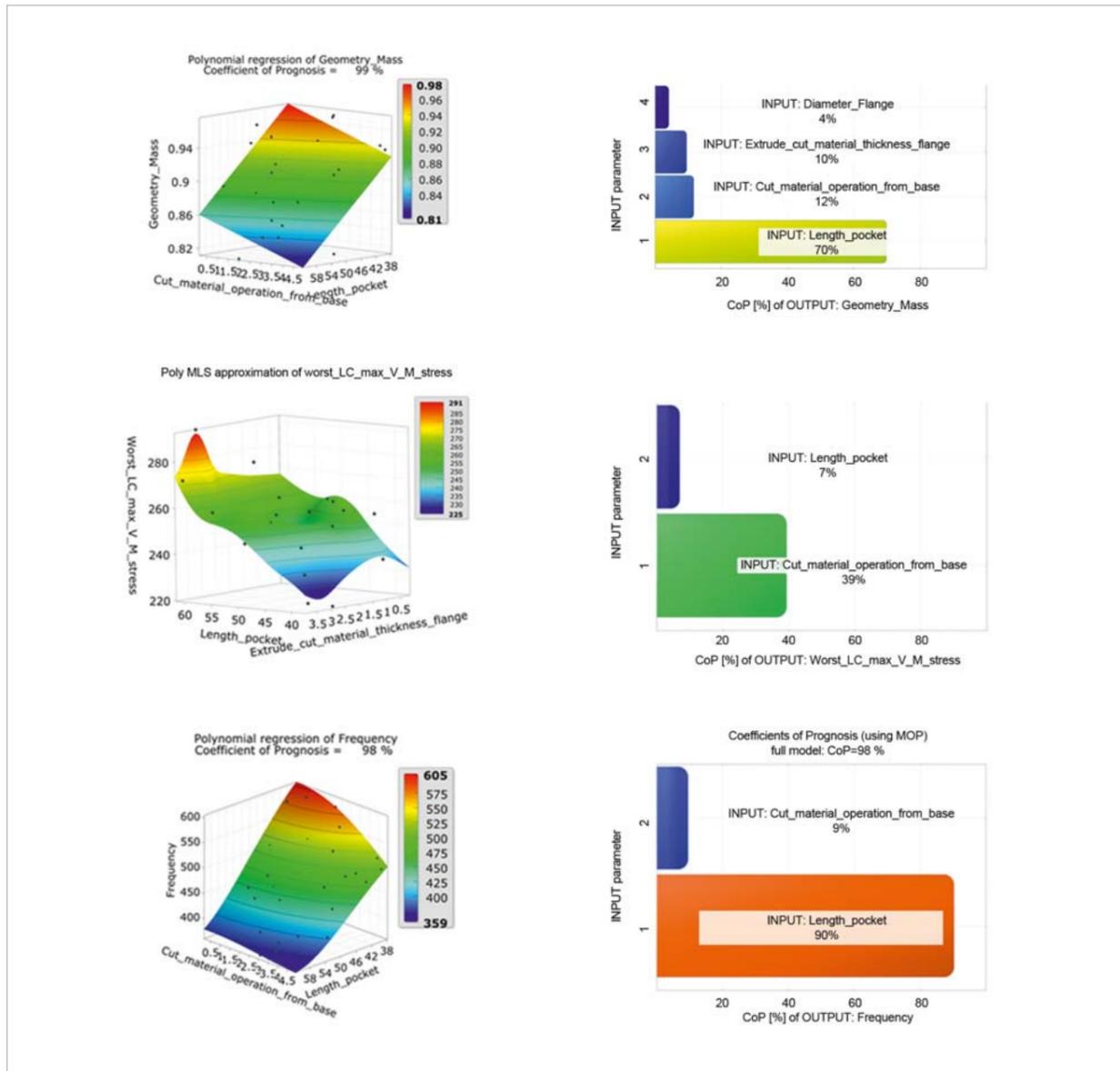


Fig. 8: Most influent input parameters for each output

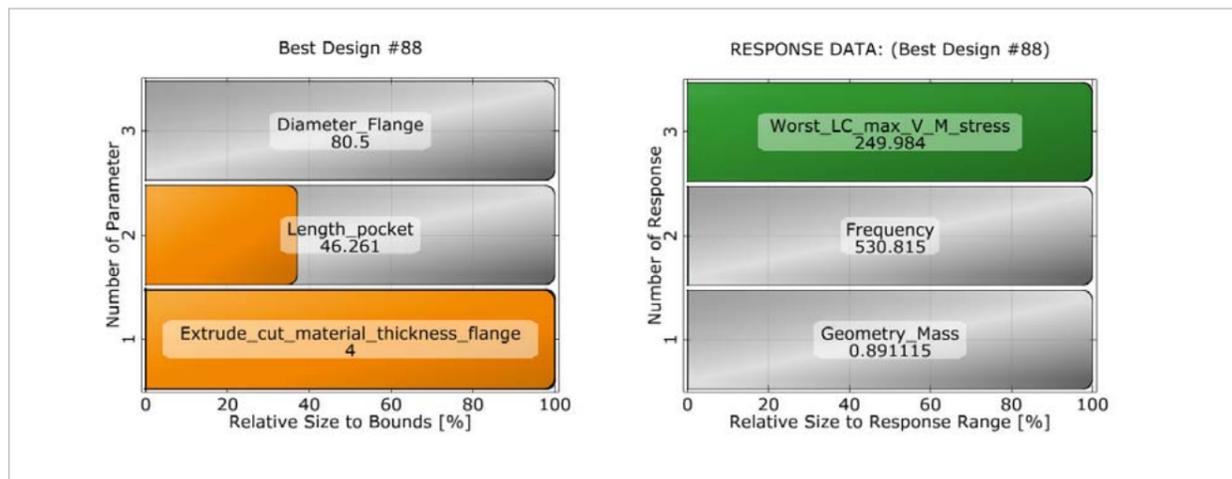


Fig. 9: Input parameters best design (left) and predicted output parameters best design (right)



Fig. 10: Initial design (left) and optimum design (right)

	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 = + 1,8 %
	1st Frequency	683 Hz	554.48 Hz = - 18,8 %

Tab. 3: Results of the optimization in optiSLang

order to build the response surfaces. Also, the most appropriate optimization algorithm is suggested. In Fig. 11, the equivalent stress distribution for the worst LC, before and after the optimization, is compared.

Validation

The validation aims to demonstrate the possibility for an optimization of other cast components using the same workflow. Therefore, a much more complex geometry with a large amount of load cases was tested. The complex geometry examined is the pressurization and degassing plate for the hydrogen tank (PPDRH). The optimization goals and constraints are the same as previously described. The geometry presents 5 external mechanical interfaces and the simulation model consists of 320 load case combinations plus the modal analysis in clamped configuration.

By using the LHS method and 4 optimization parameters, 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 using the MOP. The NLPQL is again suggested as the most appropriate algorithm for the optimization because the number of inputs is low, the variables are continuous and the optimization problem presents one objective function. The algorithm generates 154 designs

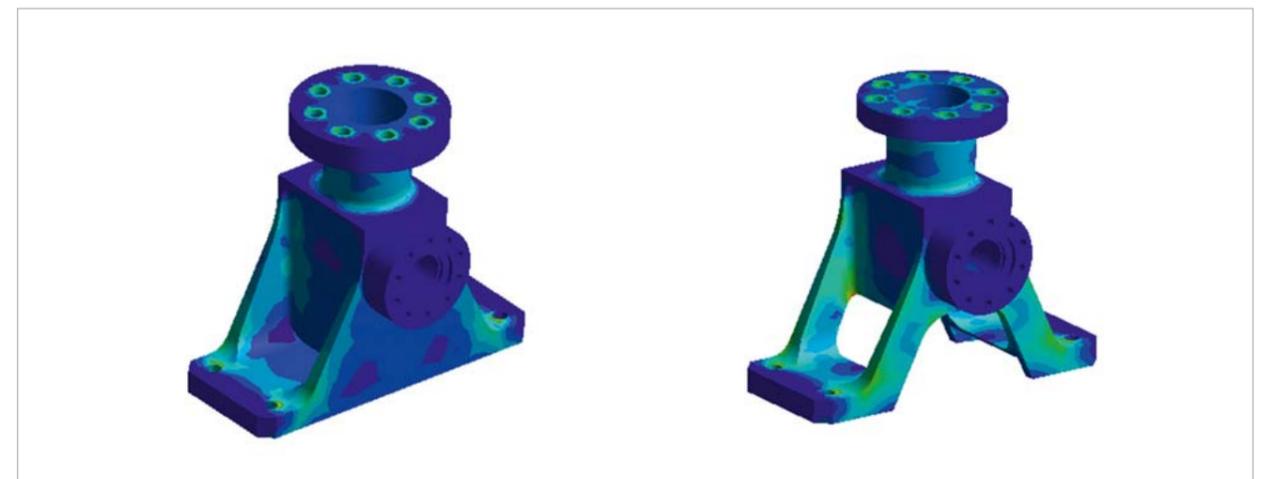


Fig. 11: Comparison of the equivalent stress distribution, initial (left) vs. optimum design (right)

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.

		Basic 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	6.580 kg = - 6,75 %
	Maximum stress	239 MPa	238 MPa = - 0,4 %
	1st Frequency	1150.7 Hz	1064.1 Hz = - 7.52 %

Tab. 4: Results of the optimization in optiSLang

By looking at Table 4, it is possible to see that all the constraints are considered and the mass value has been reduced by 6.75%. In Fig. 12, the initial geometry and the final design are compared.

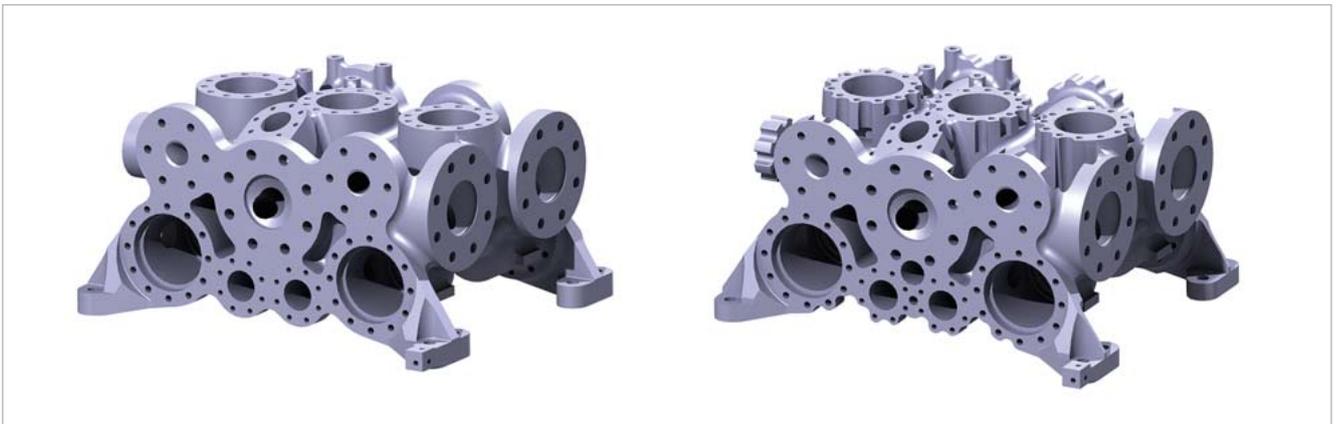


Fig. 12: Comparison of the initial (left) and the optimum design (right)

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

Conclusions

The aim of the presented work was to develop a highly automatic and efficient design optimization process to optimize different structural cast components. A parametric approach based on the Design of Experiments and the Response Surface method was chosen to perform the optimization. The process was developed, implemented and validated successfully. The design optimization was applied for the redesign of

a valve geometry with the objective to reduce the structural weight as much as possible. The initial design was optimized using the three most importance input design variables and the mass was also 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 large numbers of optimization parameter, such as 50. Thus, the same design optimization process can be applied in order to optimize more complex geometries with a large amount of geometric parameter. In conclusion, the optimization process provided an efficient, flexible, suitable approach and allowed to explore possibilities of improvement in order to satisfy goals and constraints.

Outlook

The parametrical values of the design can be improved by using the parametric interface ANSYS space claim direct modeler which allows to automatically parametrize any kind of basic geometry STEP file. This is really suitable when the basic geometry becomes more complex. The optimization allows the user to perform a multi-objective optimization by using 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 the improvement of the structural components in a multidisciplinary context in order to obtain a product with a high performance quality in several fields of application. A further step of an important improvement could be the performance of a robustness evaluation of the final design.

Author // D. Corbo (Airbus Defence and Space)

Source // www.dynardo.de/en/library