Application of Optimization Methods to the Flexseal Design Process

Alexander J. Kaczkowski[1] and Alex T. Hidalgo[2]
Northrop Grumman Corporation, Brigham City, Utah, 84307, U.S.A.

Various methods are used to guide trajectories of solid rocket motors by redirecting thrust. A common approach with an extensive history of success is the flexseal. The flexseal allows for the redirection of thrust while having the requisite strength to contain the extreme environments that exist within a solid rocket motor. Design and analysis of flexseals has typically been a lengthy and complicated process. Several key geometric dimensions directly influence load transfer through the flexseal and require iteration to obtain an optimal design. Analysis of flexseal performance is challenging and convergence is often difficult to obtain due to the nonlinearity of materials and the nonlinear solutions required to model their behavior. Historically, simplified analytical tools have been developed to guide design engineers toward an acceptable design, which is then verified with detailed analysis by structural analysts. It has been observed that the simplified analyses are often poor approximations of the results from the detailed analysis; thus, significant design iterations are often required to converge on an acceptable design. This paper summarizes an improved process for flexseal design that leverages the optimization tools of Altair's HyperStudy™ software, allowing analysis to drive design modifications in an automated process. Improvements in design time and levels of effort will be shown, along with limitations and the applicability of this process to a wider range of applications.

I. Introduction

The flexseal is a common nozzle component in a rocket motor that allows for vectoring of the rocket thrust. This change in the direction of the motor thrust is achieved through alternating layers of flexseal reinforcements and elastomeric pads known as the flexseal core. These reinforcements and pads are sandwiched between end rings. The combination of pads, reinforcements, and end rings comprise the flexseal assembly (Figure 1).

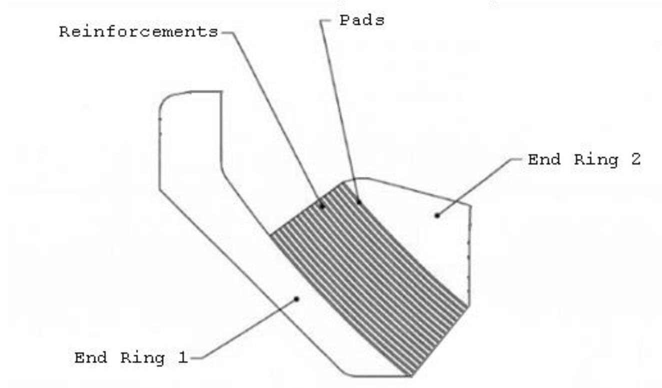

Figure 1 Flexseal assembly example.

The flexseal assembly design is one of the most important features of a rocket motor nozzle due to two primary loads on the nozzle that are transmitted through the flexseal. The first, internal motor pressure, loads the movable part of the nozzle. This pressure load is transmitted through the flexseal into the stationary part of the nozzle. The second

[1] Principal Structural Engineer, Space Systems.
[2] Principal Structural Engineer, Space Systems.

primary load on the nozzle is the vectoring load. This load redirects the motor thrust. These two primary loads cause high stresses in the flexseal assembly and make it difficult to design.

Numerous variables drive the design of the flexseal core and assembly. The most important are the magnitude of the internal motor pressure and the rotation or vector angle of the movable part of the nozzle. Several geometric design variables that control the flexseal size and shape directly affect the influence of pressure and vectoring on a flexseal's structural performance. Their modification is always necessary to obtain a design that meets structural and vector requirements and balances the torque needed to vector the nozzle and the amount of stress in the reinforcements.

A. Preliminary Design Code and Analysis

Design and analysis of the flexseal are two of the main components in the development of a rocket motor nozzle. Historically, simplified analysis tools have been used for the initial design of flexseals by design engineers without a structural analysis background. These simplified tools allow for an increase in flexseal design iterations that can be analyzed quickly at the expense of the accuracy of the stress results in the flexseal assembly components.

This rapid iteration is done by taking the design drivers of a flexseal as inputs and creating a simplified finite element model (FEM) of the flexseal to be analyzed (Figure 2). The pads and reinforcements are coarsely meshed based on the flexseal core geometry and the number of reinforcements and pads in the core. In addition, a representative forward end ring is meshed and added to the FEM. Hand calculations determine the pressure load to be applied to end ring 1 to represent the motor's internal pressure on the flexseal. Finally, boundary conditions are applied to represent end ring 2 and to rotate the flexseal core to the desired vector angle.

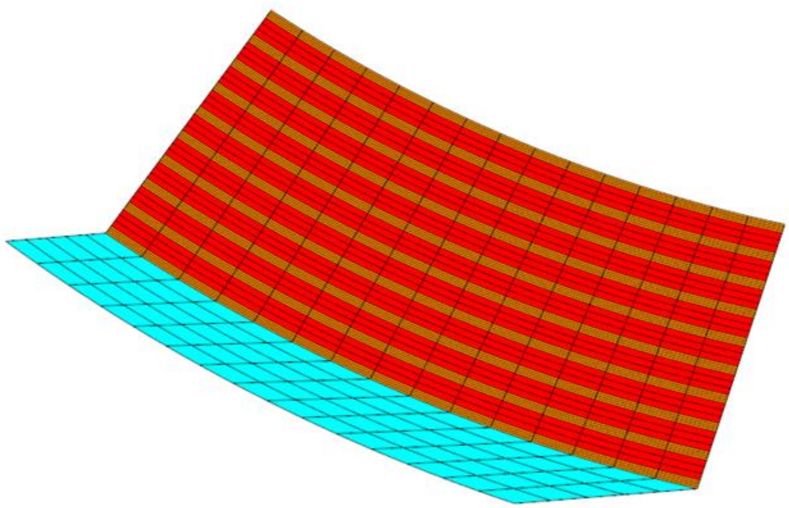

Figure 2 Rapid iteration finite element model.

This simplified FEM is then run and failure modes of the flexseal core components are assessed. The first is the stress state in the reinforcement material where material directional stresses are compared to design allowables. The second is the shear strain in the pads, which is compared to design standards and is driven by the vector angle. In addition, the torque needed to vector or rotate the flexseal core to the desired angle can be determined and compared to system requirements.

This rapid iteration cycle where design engineers input basic parameters of a flexseal core design and an automated analysis process is run to determine predetermined outputs has its advantages and disadvantages. The major advantage is the rapid iteration time. Design inputs can be put into a spreadsheet and outputs can be determined within a matter of minutes for a single design. It also allows for minimal design inputs as just the flexseal core is meshed to the exact geometry, with the end rings either not modeled or modeled as a representative design. The tool allows the design engineer to explore a representative design space with relatively little experience in common structural analysis areas such as material modeling, meshing, and finite element analysis (FEA).

With the advantage of rapid iteration time, the main drawback to the tool is the stress accuracy of the FEM. The stress state in the reinforcement material does not match the results from a full structural analysis of the flexseal core. This is due to the assumptions made to create a rapid analysis tool that can affect the stress state in the reinforcements. In addition, the flexseal assembly end rings are not modeled accurately and thus stress states in these components are not accurate. Another issue is model convergence. Often the FEM does not converge to a solution due to the high loads on the soft pad material.

B. Structural Analysis Tools

In addition to the design engineer tool, flexseals are analyzed by structural analysts in a full FEM of the rocket motor nozzle. In this analysis, the entire nozzle is modeled from where it attaches to the motor case to the nozzle exit plane. A fine mesh is used in the flexseal assembly to model the stress state in the reinforcements, pads, and fully modeled end rings. By modeling the entire nozzle, the actuator, which is used to vector the nozzle, can be modeled at the correct attach location. The full nozzle model has an accurate internal pressure load and boundary conditions applied. This results in stress predictions for the reinforcements and design end rings along with torque and pad shear stress predictions.

As with the design engineer tool, the structural analysis analytical tools for the flexseal have advantages and disadvantages. The main advantage is the increase in stress accuracy of the FEM prediction. Modeling of the entire nozzle allows for accurate representation of the localized deformations in the flexseal assembly in addition to the rest of the nozzle, which can affect stress results. Along with accurate localized deformations, modeling the entire nozzle allows for an accurate compressive load on the flexseal core. These localized deformations can also affect the torque and pad shear stress prediction. The full nozzle analysis also provides vector clearances of nozzle parts as the movable part of the nozzle is rotated to a desired vector angle. Finally, this analysis has been correlated to test data and accurately models the stress state of the flexseal assembly.

Drawbacks come with the improved stress accuracy of the structural analysis model. The most prevalent is that the model often does not converge to a solution. This is due to extremely nonlinear behavior of the elastomeric pads, which experience high deformations under the pressure load. This problem also occurs in the design engineer models but is more prevalent in the structural analysis models. Because the model encompasses the entire nozzle, the structural analysis model takes longer to converge to a solution. With this long analysis time, design iterations take much longer.

The often-significant disparities between the design code and more detailed structural analysis predictions often result in significant design iteration before an acceptable solution is obtained. A flexseal assembly needs to be designed by the design engineer using the rapid iteration tools, then passed to the structural analysis engineer who creates the FEM and runs it to determine the final stress results. This process can take up to a few days for a single iteration.

II. Explanation of Optimization Codes

To improve the efficiency of the flexseal design process, the disparities between preliminary design code and detailed analysis predictions need to be addressed. Two paths to address these issues are immediately apparent:

1) Improve the existing preliminary design code to address the prominent sources of error in its predictions.
2) Devise a new approach that allows for use of the detailed analysis models in the iterative process.

Uncertainty in the capability of the simplified modeling employed by the preliminary design code to correlate with the more accurate detailed structural analysis models make the first path undesirable. Furthermore, it is likely that any modifications required to improve the correlation would not be generally applicable or would require significant effort to validate their widespread applicability. For those reasons, it is not expected to yield a reliable tool. Alternatively, devising a new approach that employs the detailed analysis model in an iterative process removes those uncertainties. Its primary disadvantage is the significant level of effort required to build, solve, and post-process the models. For the number of iterations typically explored utilizing the preliminary design code, the same level of study utilizing the detailed FEM would be prohibitively expensive with regard to effort and schedule.

Altair's HyperMesh™ software has the capability to easily manipulate the discretized domains of the detailed analysis models and parameterize those modifications as perturbations to nodal locations. Altair's HyperStudy software provides tools to optimize the design in an automated process using the parameterized node locations from HyperMesh. This process, described in greater detail in the following subsections, does not introduce errors due to modeling simplifications because it uses the more detailed analysis models in the solution. Automating the solution and post-processing of the design iterations results in negligible increases in time and effort required of the analyst.

A. Model Construction

HyperStudy is capable of using the analysis input files for many popular finite element solvers. The FEM is built in the same fashion as if the objective were to perform a single analysis to validate the acceptability of the design. The domain is discretized, material properties assigned, boundary conditions and loads defined, and output variables specified. Consideration should be given to the results of interest for the optimization so that proper output variables are specified. HyperStudy utilizes the output variables during the optimization to inform modifications to the geometry and converge on an acceptable design.

During the optimization process within HyperStudy, the specific objectives will be defined for result outputs from the FEM. These results are often considered for a group of nodes or elements specified by the user within HyperStudy by node or element numbers or a range of node or element numbers. In anticipation of this, it is advantageous to consider renumbering the elements or nodes within the components that will be modified to a larger or offset group of numbers during model construction. This greatly simplifies their selection later on.

B. Parametrizing Node Locations for Modification of Key Geometric Dimensions

Modification of the design discretized in the FEM is accomplished using the mesh morphing capabilities of HyperMesh. Mesh morphing is a powerful tool that allows for easy mesh modifications while maintaining element quality. Groups of elements and associated nodes that are desired to move relative to each other are organized into morph domains. Their position is modified within the baseline design to define morph shapes. These groups of elements are typically distinguishable by key design features or dimensions that would be expected to directly affect the performance of the structure (e.g., a component of an assembly, a part's thickness, a radius on a reentrant corner, etc.). The objective of the optimization, the acceptable modifications to the design, and any design constraints must be well understood prior to defining the morph domains and associated shapes.

With the problem reasonably understood, the nodes and elements within their morph domains are morphed to the desired limit of their modification individually (e.g., max/min thickness, max/min radius, etc.). Each modification is saved as its own morph shape with the data stored as node perturbations within HyperMesh. There is no limitation to the number of shapes that can be created. Limiting the number to three or four is optimal to avoid overcomplicating the study and dramatically increasing solution time. More is certainly feasible if required.

Mesh morphing constraints may be used to control the general motion of nodes during morphing operations to maintain critical profiles. Comprehensive discussion of these tools is available within Altair's HyperMesh documentation.

When the desired shapes defining the allowable design modifications have been created, the Altair OptiStruct™ user profile is used to translate these shapes into design variables. A design variable should be defined for each shape that was created. The information defining the node perturbations is then exported out of HyperMesh as a node template file (.node.tpl) in the proper format for the structural solver to be used for the optimization.

C. Optimization Routine Set Up and Execution

Set up and execution of the optimization routine within HyperStudy is only briefly described in this section. Various examples and tutorials are available with Altair's HyperStudy documentation to guide the interested reader in the process.

The node template and structural solver input files are used within HyperStudy to implement the defined shape modifications in the design and study their influence on the design's response. The base model is utilized by replacing the node definitions within the structural solver input file with the parameterized node locations from the node template file. Input variables are defined within HyperStudy from the design variables created within HyperMesh. As shown in Figure 3, each variable is given an upper and lower bound.

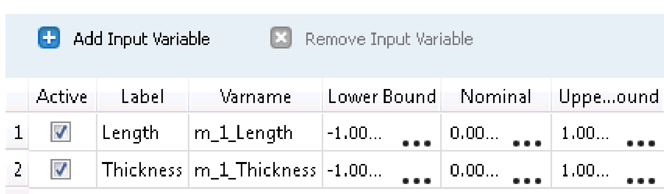

Figure 3 Example of HyperStudy input variables.

These bounds are effectively scale factors that will be applied to the movements defined by the shapes associated with each variable. An upper bound of 1.00 would move the nodes to the limit that was defined during the mesh morphing within HyperMesh. A value above 1.00 may be defined to alter the mesh beyond what was defined within HyperMesh but care should be taken not to define values so large that the mesh becomes highly distorted. The nominal values apply no modification to the baseline mesh. The software searches for feasible solutions by modifying the design within these bounds to define nodal positions, adjust them within the baseline FEM, and generate an updated structural solver input file for execution.

Once the input variables are defined, a nominal condition is evaluated (no modification of the baseline design) within HyperStudy. This allows the output responses (e.g., displacement, max stress, mass, etc.) to be defined for the optimization. After both input variables and output responses have been defined, an optimization study can be created to study the influence of the input variables on the output responses. Objectives or constraints for the output responses (e.g., maximize deformation, minimize mass, do not exceed a specific stress, etc.) are defined by the user and are used to guide the software during the optimization toward acceptable solutions. The various optimization schemes provided within HyperStudy are shown in Figure 4.

Label	Varname
Adaptive Response Surface Method	ARSM
Global Response Search Method	GRSM
Sequential Quadratic Programming	SQP
Method of Feasible Directions	MFD
Genetic Algorithm	GA
Multi - Objective Genetic Algorithm	MOGA
Sequential Optimization and Reliability Ass...	SORA
ARSM based SORA	SORA_ARSM
Single Loop Approach	SLA
System Reliablity Optimization	SRO
Xopt	xopt

Figure 4 Available optimization schemes in HyperStudy.

Discussion on each method is available within the software documentation. It is also worth noting that HyperStudy provides capabilities beyond optimization with other methods for design of experiments (DOE) and stochastic analysis that can also be employed.

D. Review Results

A method is selected and the optimization study evaluated in an automated process by HyperStudy. It iterates on the design by modifying nodal locations within the upper and lower bounds set by the input variables, generating new solver input files, and submitting them for solution. As each analysis solution completes, it looks in the results for the defined output responses, assesses them against the objectives and constraints imposed on the design, and formulates a new guess. Then the process repeats. The study is completed once an acceptable design is obtained or the limits defined for an unsuccessful termination are met. Several tools are available to review input-response relationships from the results of the study both during and after the solution. The iteration history summarizes all attempts made and the results of each. It highlights acceptable solutions and the solution that best balances the objectives and constraints. An example of this is shown in Figure 5. The two left-most columns show the scale factors applied to the two design variables for this problem and the condition column indicates whether the design constraints and objectives were met.

	Length	Thickness	Iteration Index	Evaluation Reference	Iteration Reference	Condition	Best Iteration
1	0.0000000	0.0000000	1	1	1	Violated	
2	0.3300000	0.0000000	2	2	2	Violated	
3	0.0000000	0.3300000	3	3	3	Violated	
4	0.1000000	0.0999999	4	4	4	Violated	
5	0.1000000	0.4148539	5	5	5	Acceptable	7
6	0.2000000	0.4035278	6	6	6	Acceptable	
7	0.2000000	0.4048674	7	7	7	Feasible	

Figure 5 Iteration history summary from HyperStudy.

III. Case Study

The process described above has been used on a number of separate occasions during the flexseal design process and has shown promise. Additional efforts to further understand the use of morph constraints and the general influence of key design dimensions on the flexseal response have yielded an improved approach to designing flexseals over the method employing the preliminary design code tool. A case study is presented to illustrate the time and cost savings available with this approach.

The representative flexseal shown in Figure 6 was created as a baseline design for the subject of the study. An axisymmetric FEM allowing for asymmetric deformation was constructed to assess the response of the flexseal to internal motor pressure and external vector loads (Figure 7). A similar modeling approach would be used in the previously discussed detailed structural analysis models used to verify the design after the preliminary design code is used to generate a baseline design. Key geometric dimensions known to influence the primary responses critical to an acceptable flexseal geometry were modified through mesh morphing in HyperMesh and nodal locations parameterized in the manner previously discussed.

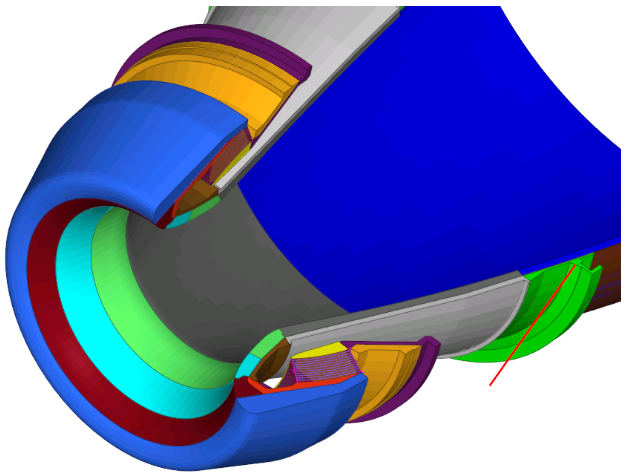

Figure 6 Representative nozzle geometry used for the case study.

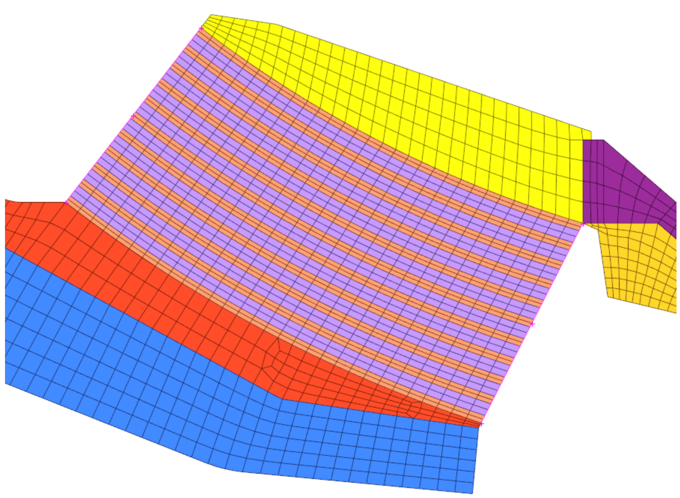

Figure 7 View of the axisymmetric finite element model used in the case study.

Output response variables were defined within HyperStudy for the flexseal reinforcement peak stress and maximum actuator force required to vector the flexseal to the required vector angle. Flexseal reinforcement peak stress was constrained to be below the material's capability to ensure no structural failure. An objective was defined to minimize the actuator force as much as possible. Doing so would provide the most options for selecting an actuator to vector the nozzle. The optimization was executed in HyperStudy utilizing the Adaptive Response Surface Method. Results are shown below in Table 1. The optimization found a feasible solution with seven iterations on the baseline design. Total solution time to complete the study was approximately 1.75 hours and is largely attributed to solution of the FEMs. Iteration #1 was a nominal condition as indicated by the zero magnitudes of the two input variables, and therefore gives an indication of the performance of the baseline design relative to the design constraints and objectives. While the baseline design required low actuator force to vector, the safety factor (SF) for the reinforcement peak stress exceeded requirements making it an unacceptable design.

The six remaining designs that were assessed are summarized in the remaining rows of Table 1. A few observations can be made from the results of the study for this particular design:

1) Input variable #1 has less influence on the response of the flexseal than input variable #2.
2) To obtain the required reduction in reinforcement stress, significant increases to actuator force will be required.

Table 1 Summary of optimization results for the case study design.

Evaluation Data					
Iteration	**Input Variable #1**	**Input Variable #2**	**Stress SF**	**Actuator Force (units of Force)**	**Condition**
1	0.00	0.000	0.906	22067	Failed
2	0.33	0.000	0.921	24398	Failed
3	0.00	0.330	0.974	34372	Failed
4	0.10	0.100	0.931	26304	Failed
5	0.10	0.415	0.997	38577	Acceptable
6	0.20	0.404	1.000	38730	Acceptable
7	**0.20**	**0.405**	**1.000**	**38788**	**Feasible**

The parameters that define the geometry of the feasible solution are provided. These can be applied to the shapes previously defined in HyperMesh to obtain the profile of the feasible geometry. Output files for each of the analysis solutions that were solved are also provided within subdirectories created by HyperStudy during execution. These may be used to generate geometry files that may be imported into computer aided design (CAD) software and used to update the baseline design by the design engineers to match the optimized profile.

To quantify the impact of this approach on the process to design a flexseal, the hours required to obtain a feasible solution were estimated based on experience for three approaches:

1) Standard Approach: Utilize the preliminary design code to generate designs and detailed finite element structural analysis to verify the design. Design is refined incrementally by reconciling disparities between design code predictions and detailed FEA results.
2) Manual Iteration Approach: Manually adjust the detailed FEM to characterize the influence of design parameters on flexseal response.
3) Optimization Approach (the approach employed in the case study): Utilize the detailed FEM and HyperStudy to automate the design manipulation and employ optimization methods.

The estimates of the hours required for each approach are summarized in Table 2. They include time to generate a baseline design, build the required detailed FEMs, and post-process results until a feasible design is obtained. These estimates assume a problem of similar complexity to the case study under discussion. For a more complex design with additional input variables and design constraints, the hours to obtain a feasible design will increase. The relative difference in hours between the approaches is expected to scale with the increased complexity.

Table 2 Hour estimates for flexseal design processes.

Estimate of Process Hours		
Standard Approach	**Manual Iteration Approach**	**Optimization Approach**
195	140	40

Improvements to the flexseal design process are dramatic due to the more direct approach. The use of the more detailed FEM within the study removes the effort required in the standard approach to account for the errors introduced by the preliminary design code. Some of this improvement is observed in the manual iteration approach since it also does not utilize the preliminary design code. The effort to manually build, solve, and post-process several detailed FEMs is significant and a large number of hours is still required. While some additional effort is required within the optimization approach to parameterize node locations, the increased efficiency in modifying the detailed FEM and generating analysis input is significant and the hours to obtain a feasible design are greatly reduced.

IV. Other Applications

The optimization approach has numerous other applications within both nozzle analysis and other rocket motor components. In this application, load is optimized with a stress value as the constraint. It can also be applied to minimize weight subject to the same stress constraint. It has been found that the application of the software works best in problems with a clearly defined design space with geometric parameters that need to be optimized to meet an objective subject to a constraint.

V. Conclusions

Overall, the optimization approach is a successful tool in the design and analysis of a flexseal assembly. The approach was successful due to several significant parameters of the optimization problem. First, a deep understanding of the key geometric parameters to vary for an optimal design was known prior to undertaking the study. This comes with a thorough understanding of the design space and problem to be solved. This allows the design and analysis engineers to rapidly come to an optimal design using automated processes. While the above paper is not intended to be a step-by-step guide to using HyperMesh and HyperStudy to solve design and analysis problems, the authors hope it provides some inspiration to design and analysis engineers to use optimization tools to solve the complex engineering problems faced daily.