EFFORTS TOWARDS AN EFFECTIVE STRUCTURAL DESIGN IN ARIANE 5 STRUCTURES DEVELOPMENT

J. Martín[1]

Construcciones Aeronauticas, S.A.
Space Division, Madrid, Spain

ABSTRACT

The continuous striving for designing quick and efficiently optimum structures obliges to define analyses processes guaranteeing that all the structure details are sized to resist the most critical conditions they may encounter. In response to this need, specific tools have been developed at CASA Space Division to design structures of the Upper Stage of the ARIANE 5 Launch Vehicle. The use of these tools guarantees that none of all possible loading conditions and structure details is left unattended.

[1] Structures Engineer. Launch Vehicles Structures Analysis Unit

BACKGROUND

Any design of a structure is supported by a set of analytical investigations that provide its basic foundations. These analytical investigations strongly rely on the use of the finite element technique to predict the response of the structure when subjected to loading environments of diverse nature. The quality of the analysis is clearly linked to the quality of the model generated to represent the structure. Strong emphasis is to be put in the correct modeling of the different elements of the structure. Structural model (built using finite element theory) parameters are to be adjusted to accurately simulate the real structure.

The increasing complexity in the loading environment and in the structure definition, obliges to devise analysis approaches departing from those that were common practice up to that moment. This particular is even more evident when the objective is the definition of an “optimum” structure. The need to consider all the possible load conditions for each part of the structure emerges when none of the structure details is to be left unattended. To tackle that is perhaps an impossible task without assistance from an automated analysis tool.

In response to this need, CASA·Space Division endeavored to develop such an automatic analysis tool since the initiation (back in 1989) of the involvement in the development of Ariane 5 structural elements (EPS- and VEB-Structures, and Payload Adapters). Table 1 gives a short description of these structures. The approach being taken in this ongoing (it has been in continuous development since the beginning) automatization project is to blend proven engineering tools and existing knowledge database (rules for modeling, structure parts analysis procedures,...) into an efficient analysis system. This analysis system is presently oriented towards the static/strength analysis.

Through all these years, in parallel with the system development, important benefits have been obtained from its use. Some of these benefits reverted in enhancing the capabilities of the system. The system has demonstrated to be a rapid, efficient, accurate, and cost effective tool. The flexibility of the system allowing for quick evaluation of the consequences of potential changes in the structure definition (material change, ...) and/or the loading environment (payload mass increase, ...).

The MSC/NASTRAN system has served as an important resource in this development. In the model preparation phase, it has proven to be an invaluable assistant (by means of its optimization solution) in the preparation of simplified and representative models of different structure details (interface rings, connection fittings, ...). In the analysis phase, it has allowed the creation of the structure response database, key ingredient in the subsequent data processing activities.

PROBLEM STATEMENT

The loading conditions definition for the Ariane 5 launcher structures located in the upper part is characterized by its multiple origin. Static and dynamic considerations lead to the definition of a variable loading environment along the different flight phases. In essence, the primary loads are axial loads created by the engines (EAPs and EPC) thrust loads and lateral loads (shear force and bending moment) created by the aerodynamic loads. To these primary loads, the quasi-static conditions associated to the launch vehicle response to dynamic loading conditions (boosters pressure oscillation, ...) are added. When the complete set of loading components is considered, taking into account all the possible combinations in a time consistent manner, a total number of more than 1,000,000. load cases can be the result.

The specific architecture of the launch vehicle (EAPs attachments lay-out, ...) makes the load distribution along the different structures forming the vehicle strongly dependent on the retained structural configuration. Design modifications in one structure could affect the sizing of an adjacent one. CASA · Space Division has had the opportunity to evidence this statement due to its involvement in the development of three structures with common interfaces. This has obliged to consider for the different investigations a model including all the adjacent structures (see Figure 4), and correctly representing the different loading components (EAPs reaction forces, EPS·Tanks inertia acceleration, ...).

In the other hand, the complexities in the definition of the structure, associated to the high demanding requirements (need to balance conflicting requirements for strength and stiffness, ...), the widespread use of composite materials, and the continuous search for the 'optimum' structure are to be combined with the global objective of producing structural designs in less time with no impairment in the quality. All the details of the structure are to be interrogated to guarantee that none of the parts has been left unattended. To manage all this would be an impossible task without the assistance of an appropriate analysis system. Figure 2 presents an outline of this analysis system, indicating with the MSC logo the contribution of MSC/NASTRAN to the system.

Such an analysis system is to be supported by four strong pillars (see Figure 1): a representative analysis model, a powerful database management system, a solid set of analysis procedures, and a set of reporting capabilities. At the same time, the system design requires of the necessary flexibility for being adapted to scenarios differing from the original one and to the specific requests from the different engineers using the system.

ANALYSIS MODEL

Finite element analysis is normally used to predict the response of a structural design to the application of a set of loads. The first step in this type of analyses is the generation of the finite element model from the physical model. The real structure is idealized and converted to a finite element model. This includes selection of elements and nodes, and mesh generation. To accomplish this task, the engineer relies on his judgment and past experience. Automated mesh generators are used to reduce the labor in this task (SDRC I-DEAS™ is being used for this purpose). But modeling goes beyond mesh generation. A detailed knowledge of finite element modeling and the capability of each one of the element types is necessary to avoid mismodeling. Through the years, different rules for modeling and its verification have been established aimed to guarantee the quality and representativeness of the model. The compliance of these rules is specially enforced when there is no test data that could be used to validate the modeling. This finite element model will be the key piece in the overall effort to quick and efficiently design the structure.

In order to accurately simulate the real structure, the finite element model needs to be tuned, or adjusted. Tuning is necessary since the model may not respond to loading conditions in the same manner as the real structure. The differences occur because the structure's parts are modeled by less accurate finite elements which can closely approximate, but not exactly duplicate, all the part's physical characteristics.

The finite element model can sometimes be improved by increasing the number of nodes or adjusting the model's parameters. The size of the model can be limited by different type of constraints. In the case of the Ariane 5 structures under discussion, the limitation is due to the need to make compatible the model definition of the different structures composing the upper part of the launcher (a global model is needed to correctly simulate the load path through the different structures), structures that are designed by different companies.

The elements used to represent the structural components are simple idealizations that probably do not exactly duplicate the behavior of the component in bending and/or torsion if loaded identically. The element properties are to be adjusted to obtain the best simulation of the component of interest. Those elements do not need to have the same physical and material properties than the component.

This comment is specially applicable for the case of joints (connection between panels, ...). To accurately represent the structure displacements (and the correct internal loading distribution), it is important to include the true behavior of those joints. Springs can be used for this purpose adjusting the K-value.

The selected parameters (design variables) are adjusted to force the model to duplicate a set of structural performances (objective). This set of structural performance includes displacement of different nodes due to different loading conditions derived from a detailed finite element model of the component. Detailed model that would be used in subsequent investigations to determine the stresses distribution in the component (*Structure Part Analyzer*).

Various material (Young's Modulus, ...) and physical properties (thickness, inertia, 'offset', ...) are adjusted to make the desired deformation(s) difference a minimum. Certain physical and material constraints are also observed during the tuning to maintain the credibility of the model.

The MSC/NASTRAN optimization solution (SOL 200) has been effectively used for those tuning processes. Prior to the use of the optimization techniques, the model was manually tuned to match the desired static displacements. The parameters were varied based upon engineering judgment, which improved as more insight was gained in the effect of the parameter modification on the model performance. This approach proved to be a time-consuming and, sometimes, difficult technique. The procedure to derive these simplified models duplicating the response of very detailed and representative models of components is presently of common use ([4], [5]).

Optimization Process Particularities

To tune the model, different loading conditions are to be considered. In some cases ([4]) a total of nine loading/boundary conditions combinations have been contemplated. The objective is to minimize the distance between the detailed and the simplified models displacements. This would oblige to formulate a second-level response (DRESP2) by combining responses from different subcases. Unfortunately, this is one of the few restrictions that apply to the formulation of second-level response ([6], p. 79).

To overpass this limitation, a simple technique has been used. Several equal models are simultaneously analyzed. The only differences between the models (apart from grids and elements numbering) are the loading/boundary conditions. In that case, all the responses of interest correspond to a single subcase and can be combined in the DRESP2 formulation.

Importance of Representative Modeling

The finite element model should correctly represent the structure stiffness to guarantee the "quality" of the load path derived from the analysis. This is specially important for complex structures (different materials, branched shells, ...) where redundant load paths are possible and when an optimum structure is the main objective. Incorrect assumptions modeling one part of the structure affect to the totality, questioning the validity of the obtained results.

To illustrate the influence of the modeling in the response and design of the structure, we have selected the case of the EPS-Structure/Propellant Tanks interface ring (Figure 5). Two plate elements are used to simulate the stiffness of this part in the global analysis model (one of the elements represents the flange interfacing with the tank and the other, the connection between the ring and the sandwich platform).

The initial modeling, basis for the optimization, was established based on normal engineering assumptions trying to represent the physical characteristics of the interface ring. The updated model was the result of the model parameters tuning to reproduce the response of a detailed local model. Subsequent local and 'full-scale" testing validated the accuracy of the modeling approach.

Obtained results (see Figure 5) clearly show the influence of the modeling in the internal loading distribution. This influence is evident not only at the location directly affected by the modification

(❶), but also at adjacent structures (❷) that could be being designed by a different company. Figure 5 shows a reduction (about 20.%) in local bending moment (critical load component) obtained after modifying the interface ring modeling.

DATABASE MANAGEMENT

The analysis system inherently needs of a flexible management of all the data involved. Different databases (*Loading Environment*, *Structure Response*,...) are the repositories of all this data. The content of these databases is to be continuously monitored to guarantee the use of up-to-date information. To efficiently manage all this data, the availability of good designed databases is of primary importance. An important effort has been expended in the design of databases which structure can be adapted to new and/or changing requirements (addition of new loading components, ...).

The *Loading Environment Database* is created from the loading data appearing in the different applicable specifications. These specifications range from the system level data (inertia accelerations, ...) to the structure level data (AESTUS engine thrust, ...).

The *Structure Response Database* is created by extracting the information from the MSC/NASTRAN .F06 files. Investigations are under way to use other MSC/NASTRAN files that would accelerate the process augmenting the information extracted from the performed analysis.

The different database management activities are embedded in the analysis system, facilitating the interface between the different modules. The information flow is strictly controlled, providing to each analysis module the needed data and recovering from it the appropriate data.

ANALYSIS PROCEDURES

Elementary Loading Conditions

Once the finite element model is built, the external loading conditions incurring during the vehicle operations are to be applied to that finite element model. To be able to deal with the huge loading environment (more than one million cases could be theoretically possible) is clearly necessary to extract a set of elementary load cases which linear combination would allow to study anyone of the potentially sizing load cases.

In the case of the EPS-Structure, a total of twenty-two elementary cases are extracted. In essence, these cases correspond to,

- Inertia acceleration (in three axes) acting separately on the most important internal masses (four propellant tanks, AESTUS engine)

- Payload Inertia Loading

- Propellant Tanks internal pressure (6 to 20. bars depending on the flight phase), and differential pressure between the launcher compartments

- EAPs attachment points reaction forces

Finite Element Analysis

The completed finite element structural model with the adjusted properties and the loading definition (elementary cases) is ready for analysis. MSC/NASTRAN (currently at Version 69) is used for the finite element model analysis. This analysis produces the internal structural loads due to the elementary loading conditions for each element of the structural model (grid points displacements and force balance are also recovered for subsequent analyses). The modeling approach followed for the Propellant Tanks obliges to model modifications to run the different selected loading conditions. These modifications affect to the consideration of the propellant tank masses and do not affect the validity of the subsequent results linear combination. A total of five separate runs are necessary. A simple operating system script manages the runs submittal and combines the recovered results (forces, displacements, ...) to create the *Structure Response Database*.

Data Processing Code

The *Data Processing Code* is the core of the analysis methodology depicted in this paper. It processes the data contained in the *Structure Response* and *Loading Environment Databases*, with the assistance of external *Structure Parts Analyzers*, conveying the obtained results to the *Reporting Processors*.

This module carries out the "number crunching" allowing to compute the response (force, stresses, margin of safety, ...) of each one of the components of the structure for each one of the load cases (the finite element model response is transformed in structure component response). The intensity of this task (more than 2,000. details are analyzed for 1,000,000. load cases) makes this module the most computer time expensive of the complete analysis chain. Efforts are under way to augment the efficiency of this link of the chain (see *Further Improvements/Future Work* section).

The information processed by the code (stress, forces, displacement, ...) is directly controlled by the analyst. In cases where the complete processing of the information is not needed, the analyst can control the code flow deciding parts to investigate, load cases to consider and output to produce. This option is of special interest when performing investigations focused on specific parts of the structure or specific segments of the flight sequence, when the complete processing would produce unnecessary time delays.

Structure Parts Analyzers

Different analyzers are used to study each part of the structure from the data recovered from the finite element analysis. These analyzers provide the necessary information to the Data Processing Code to size each part of the structure. The *Data Processing Code* manages the correspondence between the finite element analysis data and the analyzer to be considered. The requested data is provided to the analyzer, which returns the associated margin of safety.

Different analyzers are defined depending on the structure part of interest. For the shells, the type of construction determines the analyzer. Presently, the existing analyzers can size isotropic and composite honeycomb panels and stiffened metallic panels. The margins of safety associated to local instability criteria, and yield and ultimate strength are computed using close form formulation.

Connections between the different elements by means of tension or shear type joints also have specific analyzers. The capability of each component of the connection (bolts, flanges, ...) to withstand the loading is analyzed.

For some specific structure parts (interface rings, ...) the results of detailed finite element models provide the necessary insight into its capability to resist the different loading conditions.

The modular design of the system allows for an easy integration of additional analysis modules (analyzers) that could be needed for design configurations differing from the ones presently considered.

REPORTING CAPABILITIES

The importance of a precise reporting of the results of the analytical investigations is well known. The engineering community also acknowledges the effort and time necessary for that task. An effective analysis system would be incomplete without the support of reporting tools.

These reporting tools are also important during the analysis process to assist in the quick identification of potentially critical items that could require a modification of the dimensions of some components. These modifications would alter the stiffness properties obliging to re-run the Analysis Process. Two levels of reporting are presently being considered. A simplified level provides all the data to ascertain the validity of the structure configuration, and a detailed level provides all the data to adequately document the results of the investigations (graphic visualization of results, detailed stress analysis reports for specific parts, ...).

Graphic Visualization of Results

Several post-processing programs are available to directly visualize the results of a finite element analysis. CASA · Space Division presently utilizes SDRC I-DEAS™ Master Series™ for those tasks. The data of interest can be visualized if it can be prepared in a format that I-DEAS can understand. The Universal File format is used for that. Specific programs have been prepared through the years to produce the desired data results visualization. For instance, one of these programs produces a developed view of a cylinder shell finite element model allowing to visualize in a single figure the contour map (MS, stresses, ...) of interest.

Figure 6 shows the distribution of minimum margins of safety for the CFRP facings of the internal panels of the EPS-Structure. The margin at each location could correspond to a different ply of the laminate and to a different load case.

The possibilities are enormous. Although several visualization modules are readily available, the analyst can easily create new ones to respond to any specific need. As an example, Figure 7 shows the evolution of the predicted minimum margin of safety in the core of the EPS-Structure tanks support platform along the different phases of the launch.

Numeric Presentation of the Results

Complementing the graphic visualization of the results, data can be produced in numeric format. In a similar manner to the graphic visualization, the possibilities are also numerous. Presently, the following presentation possibilities are implemented,

- Tabular presentation of results (summary of minimum margins of safety, ...) as depicted in Figure 8 – φ.

- Interface with PC based spreadsheet programs to fill in stress analysis forms that can be directly included in the formal documentation. Examples are shown in Figure 8 – κ and λ.

CONCLUSIONS

Through this paper, we have quickly summarized the most important characteristics of the analysis process being employed by CASA · Space Division in the development of structures of the Ariane 5 launch vehicle. The active participation of MSC/NASTRAN in the whole process is highlighted.

The system has been built around tools that were already in use at CASA · Space Division. The system has been evolving in parallel to the evolution of those tools, and by adding new tools (some of them specifically developed). The engineers using the system have been the most important contributors to its continuous enhancement.

Its use along the last seven years has shown this system as a fast, efficient, accurate, and cost effective tool. The modularity of the system has allowed modifications and improvements to be made in an easy manner. The addition of new procedures or computational algorithms is a straightforward process.

All the benefits of such an analysis process would be lost without a representative finite element model of the structure. MSC/NASTRAN has effective tools to assist in the production of an accurate model. The use of the optimization solution to duplicate the response of detailed models has been invaluable to produce global models reproducing local responses of the structure's components.

The results numeric and graphic visualization can not be left in a second plane. Clear results presentation enables the analyst to take decisions related to the validity of the structure configuration being analyzed. The integration of the reporting capabilities in the analysis process guarantees the accuracy of the data being presented in the final documentation.

FURTHER IMPROVEMENTS/FUTURE WORK

As it has been mentioned, this analysis process is in a state of continuous improvement.

At the present time, the process considers only the static response of the structure, and does not have the capability to "decide" a design modification if negative margin of safety is obtained for any of the failure modes interrogated by the analyzers. To overcome this limitation appears with a high priority in the TO DO column.

The increasing demand for faster and better analysis processes obliges to continuous "tune-ups" to accommodate the system to those demands. The *Data Processing* engine has been the first one to show signs of weakness, being necessary its revision. Present requirements (elements and load cases to consider, failure modes being interrogated, ...) exceed in more than a factor of 100. the initial ones, which were the basis to establish the code.

The evolution of the analysis system, deciding and implementing the adequate improvements, is conducted by the engineers that continuously utilize the system. Their needs are transformed into objectives for future versions of the system.

ACKNOWLEDGMENTS

The author wishes to acknowledge his colleagues at CASA · Space Division for their continuous effort in developing, applying, and improving the presented analysis process.

REFERENCES

[1] Martín, J., "Brief History of the Development of the EPS-Structure", Proceedings of International Conference on Spacecraft Structures and Mechanical Testing, ESA SP-321, pp. 203-208 (1991)

[2] Martín, J., and Gómez-Molinero, V., "Some Considerations about the Dimensioning of Ariane 5 Structures", *Ariane 5 Structures and Technologies*, pp. 41-56 (1993)

[3] López, M.A., Martín, J., and Viñé, J., "Development and Use of an Analysis Methodology Accounting for the Complete Loading Environment. Application to the EPS- and VEB-Structures", *Ariane 5 Structures and Technologies*, pp. 75-87 (1993)

[4] San Juan, J., "A Special Finite Element Model Analysis Technique for the VEB-Structure Ring Housing the Ariane 5 Upper Stage Separation Device" Proceedings of Conference on Spacecraft Structures, Materials and Mechanical Testing, ESA SP-386 , pp. 269-276 (1996)

[5] Canay, M., "Correlaciones entre Modelos Matemáticos de una misma Estructura usando las Herramientas de Optimización de MSC/NASTRAN", Temario de la 1ª Conferencia de Usuarios de MSC-Ibérica, S.A., Madrid (España), 1995

[6] *MSC/NASTRAN Design Sensitivity and Optimization User's Guide, Version 68*, The MacNeal-Schwendler Corporation, Los Angeles, CA, May 1994

[7] *I-DEAS™ Master Series™ User's Manual*, Structural Dynamics Research Corporation, Milford, OH, 1997

[8] Mason, P., et al. "Towards a Realistic Structural Analysis/Design System", *Computers & Structures*, Vol. 10, 1979, pp. 285-194.

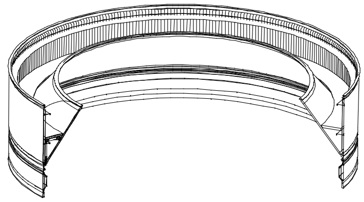
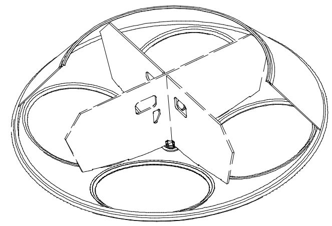
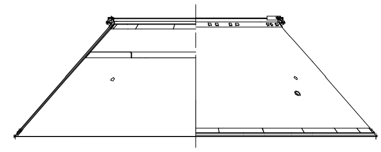
	 VEB-STRUCTURE	 EPS-STRUCTURE	 PAYLOAD ADAPTER
Dimensions (mm)	Lower Interface: ∅ 5405 Upper Interface (Cylinder): ∅ 5405 Upper Interface (Cone): ∅ 3936 Cylinder Height: 1560 Cone Height: 870	Lower Interface: ∅ 3936 Upper Interface: ∅ 2624 Cone Height: 780	Lower Interface: ∅ 2624 Upper Interface: ∅ 1194 (937 or 1666) Adapter Height: 860 (900 or 950)
Structure Description	Aluminum Alloy Stiffened Cylindrical Shell CFRP Skins/Honeycomb Core Sandwich Cone Shell Aluminum Alloy Interface Rings	CFRP Skins/Honeycomb Core Sandwich Cone Shell and Internal Reinforcing Panels Aluminum Skins/Honeycomb Core Sandwich Tanks Support Platform Shell Aluminum Alloy Interface Rings	CFRP Skins/Honeycomb Core Sandwich Cone Shell Aluminum Alloy Interface Rings

Table 1 · Brief Description of Ariane 5 Structures Developed by CASA

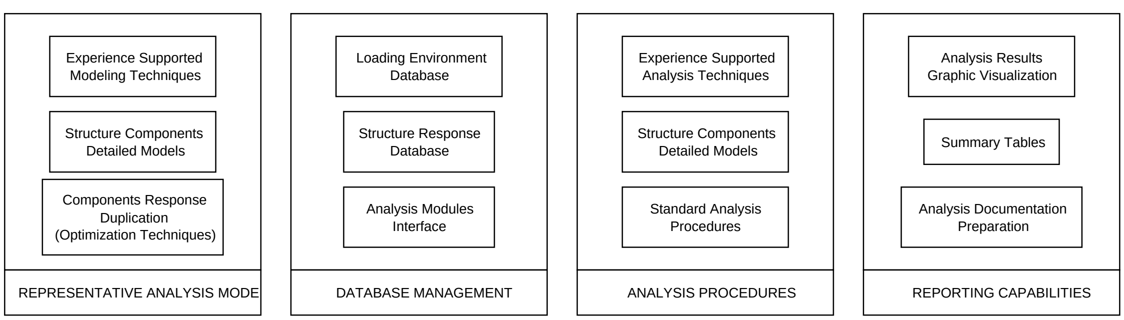


Figure 1· Analysis System Supporting Pillars

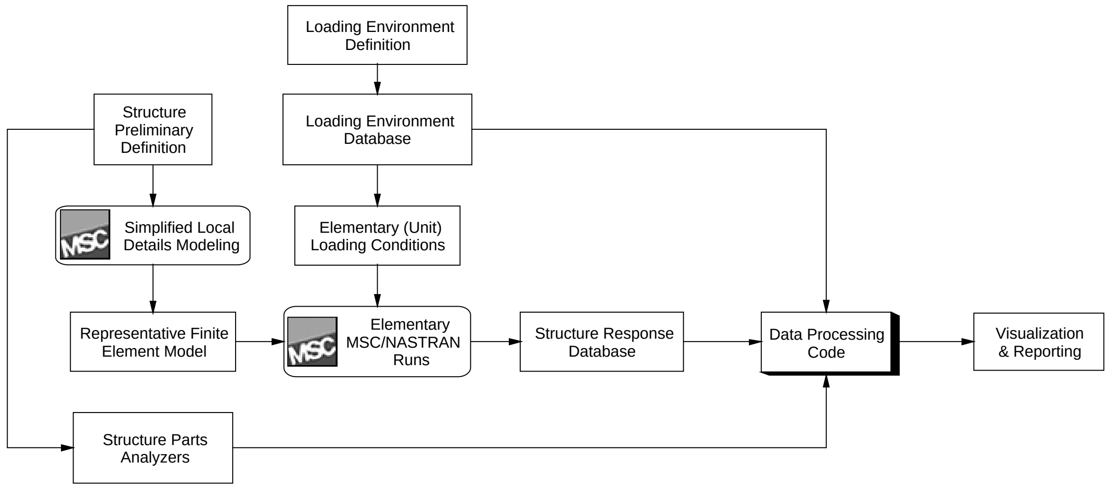


Figure 2 · Structural Sizing Process Diagram

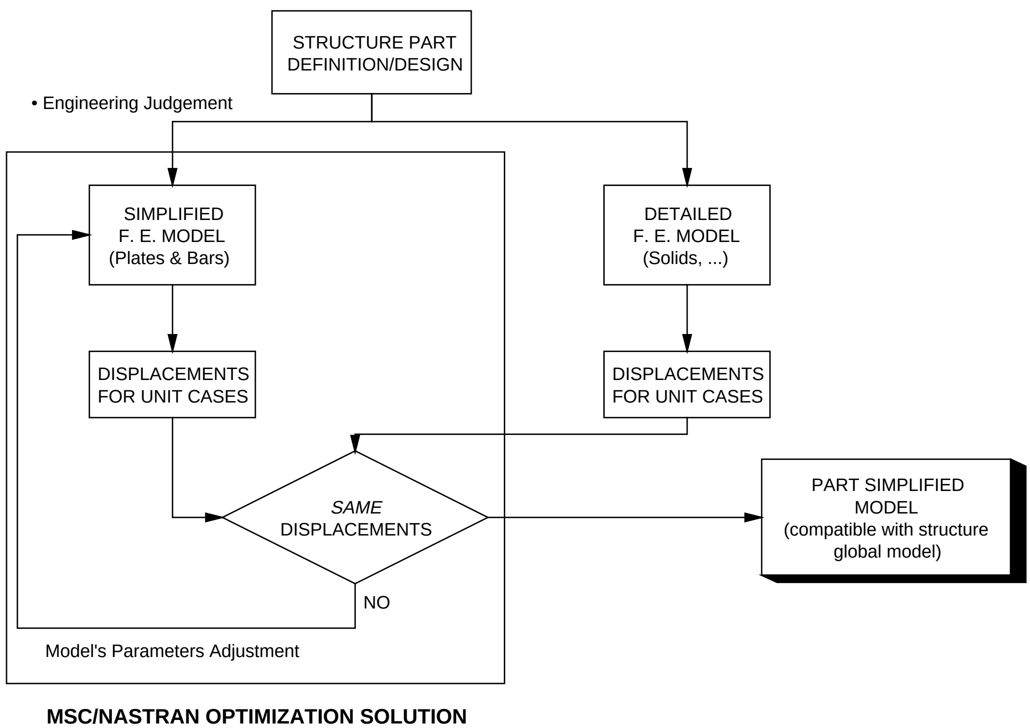


Figure 3 · Structure Details Simplified Modeling Technique

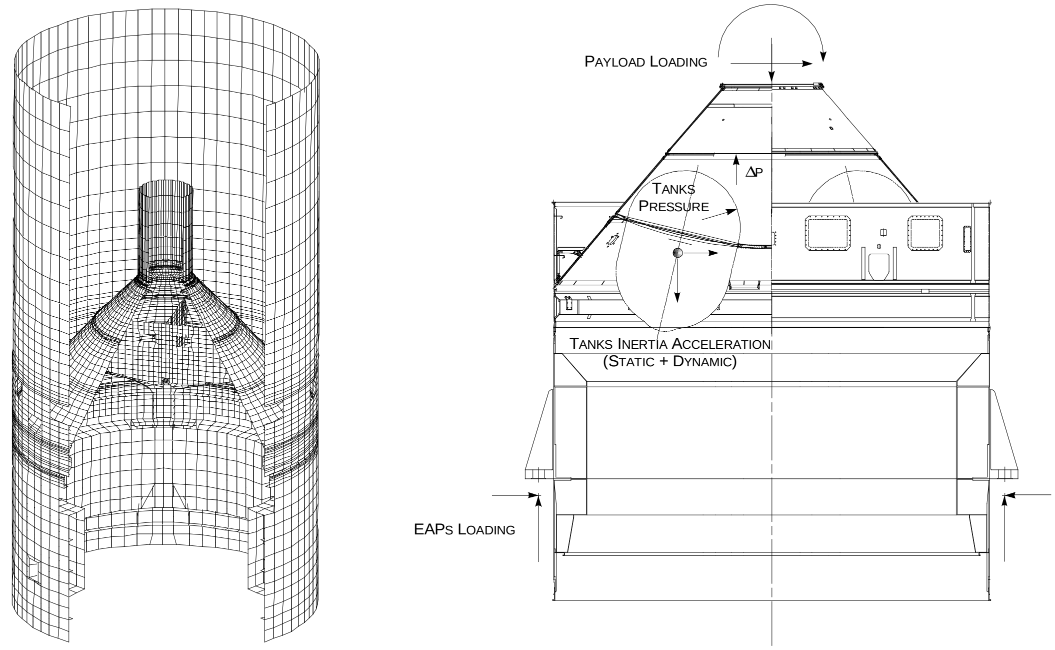


Figure 4 · Ariane 5 Upper Part Structures. Analysis Model and Loading Conditions

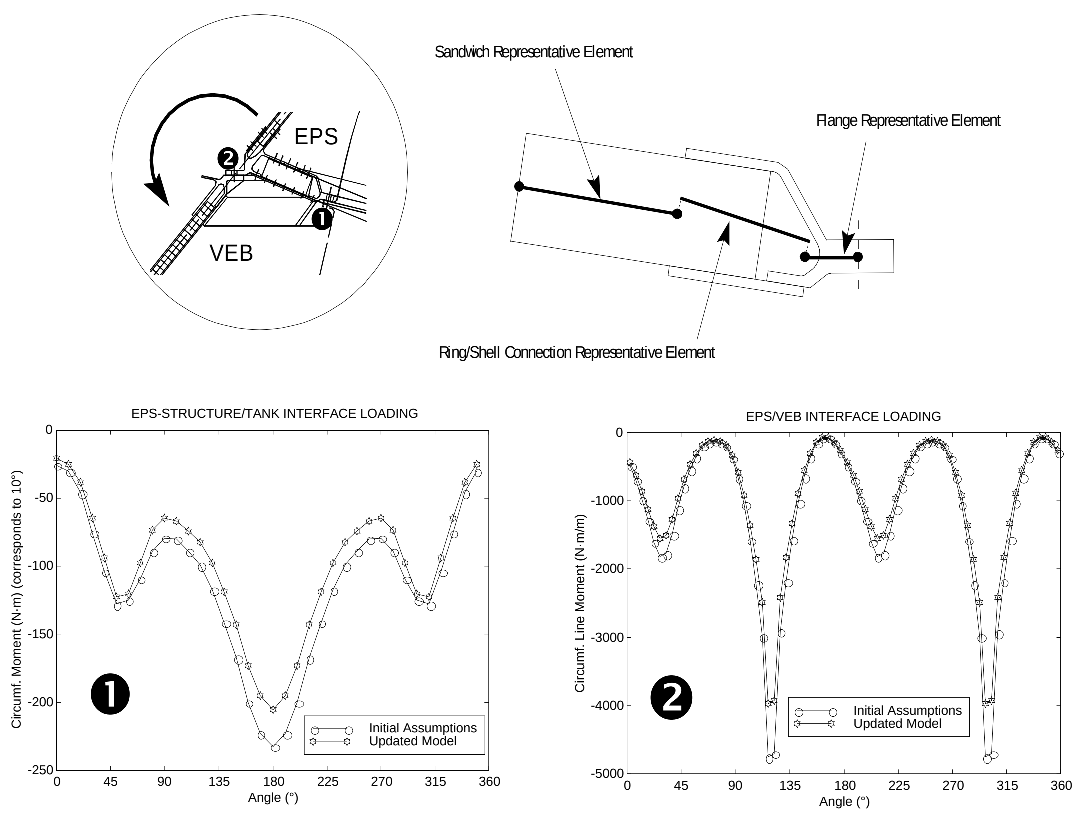


Figure 5 · Importance of Representative Local Details Modeling

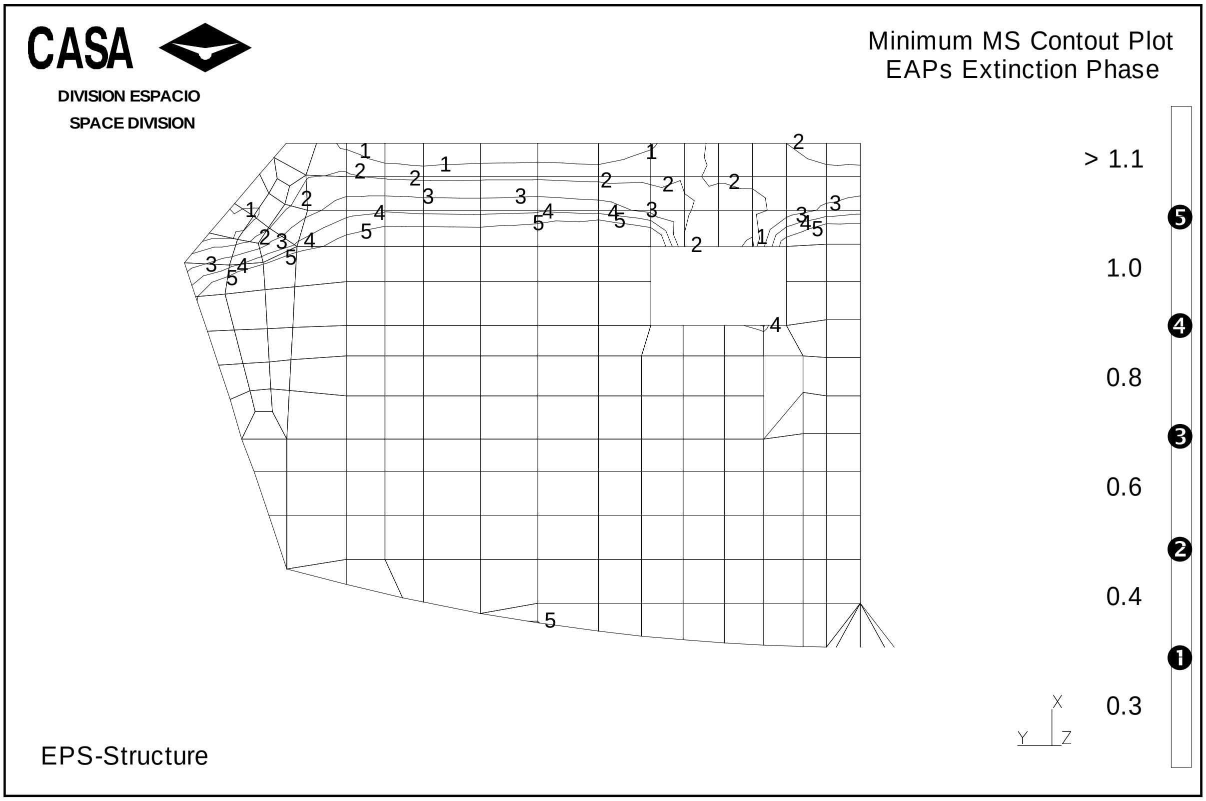
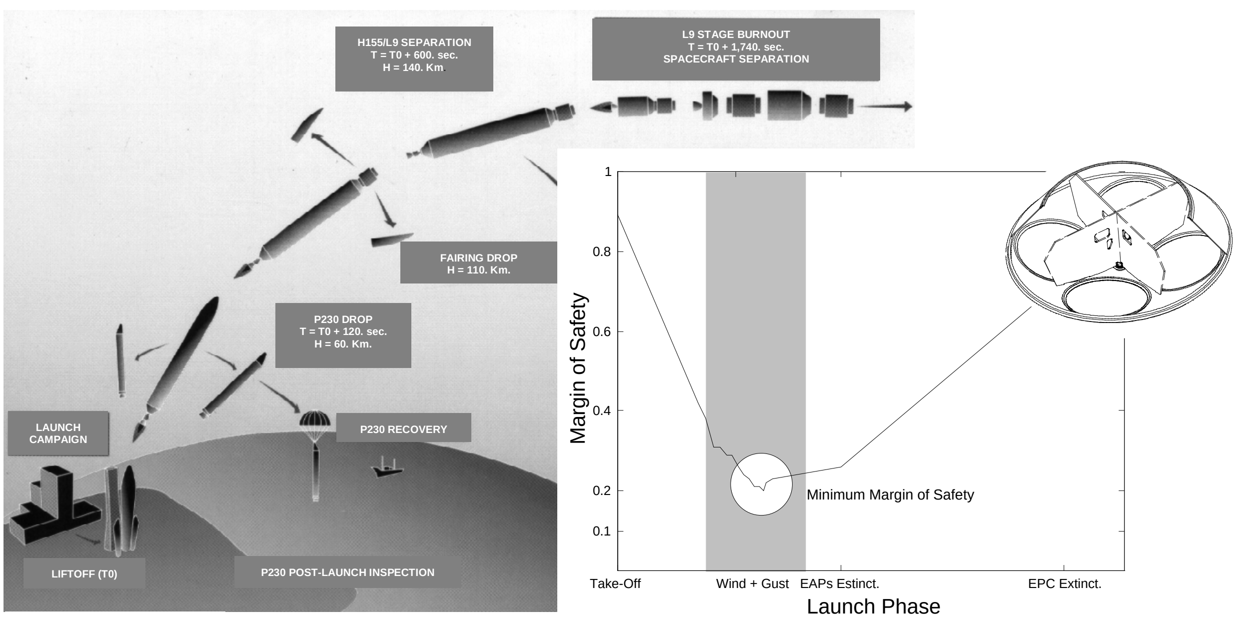
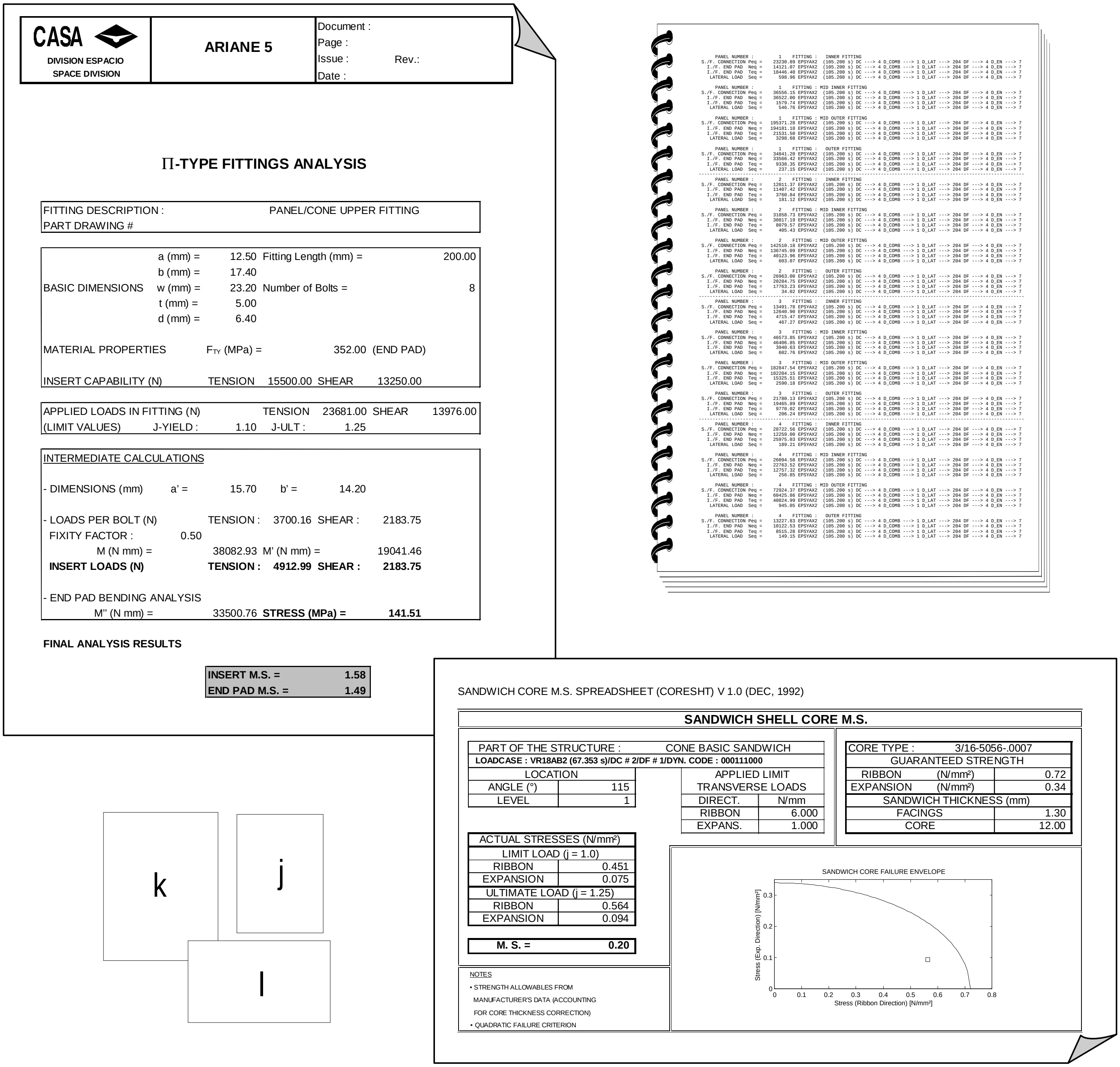


Figure 6 · Analysis Results Post-processing. Minimum Margins of Safety Contour Plot



© CNES (France)

Figure 7 · Margin of Safety for the different Launch Phases • Ariane 5 Mission Profile



CASA DIVISION ESPACIO SPACE DIVISION | ARIANE 5 | Document : Page : Issue : Rev.: Date :

Π-TYPE FITTINGS ANALYSIS

FITTING DESCRIPTION :			PANEL/CONE UPPER FITTING		
PART DRAWING #					
	a (mm) =	12.50	Fitting Length (mm) =		200.00
	b (mm) =	17.40			
BASIC DIMENSIONS	w (mm) =	23.20	Number of Bolts =		8
	t (mm) =	5.00			
	d (mm) =	6.40			
MATERIAL PROPERTIES	F_{TY} (MPa) =		352.00 (END PAD)		
INSERT CAPABILITY (N)	TENSION	15500.00	SHEAR	13250.00	
APPLIED LOADS IN FITTING (N)		TENSION	23681.00	SHEAR	13976.00
(LIMIT VALUES)	J-YIELD :	1.10	J-ULT :	1.25	

INTERMEDIATE CALCULATIONS

- DIMENSIONS (mm)	a' =	15.70	b' =	14.20	
- LOADS PER BOLT (N)	TENSION :	3700.16	SHEAR :	2183.75	
FIXITY FACTOR :	0.50				
M (N mm) =		38082.93	M' (N mm) =	19041.46	
INSERT LOADS (N)	**TENSION :**	**4912.99**	**SHEAR :**	**2183.75**	
- END PAD BENDING ANALYSIS					
M'' (N mm) =		33500.76	**STRESS (MPa) =**	**141.51**	

FINAL ANALYSIS RESULTS

INSERT M.S. =	**1.58**
END PAD M.S. =	**1.49**

SANDWICH CORE M.S. SPREADSHEET (CORESHT) V 1.0 (DEC, 1992)

SANDWICH SHELL CORE M.S.

PART OF THE STRUCTURE :	CONE BASIC SANDWICH		
LOADCASE : VR18AB2 (67.353 s)/DC # 2/DF # 1/DYN. CODE : 000111000			
LOCATION		APPLIED LIMIT TRANSVERSE LOADS	
ANGLE (°)	115	DIRECT.	N/mm
LEVEL	1	RIBBON	6.000
		EXPANS.	1.000

CORE TYPE :	3/16-5056-.0007	
GUARANTEED STRENGTH		
RIBBON	(N/mm²)	0.72
EXPANSION	(N/mm²)	0.34
SANDWICH THICKNESS (mm)		
FACINGS		1.30
CORE		12.00

ACTUAL STRESSES (N/mm²)	
LIMIT LOAD (j = 1.0)	
RIBBON	0.451
EXPANSION	0.075
ULTIMATE LOAD (j = 1.25)	
RIBBON	0.564
EXPANSION	0.094
M. S. =	**0.20**

NOTES
- STRENGTH ALLOWABLES FROM MANUFACTURER'S DATA (ACCOUNTING FOR CORE THICKNESS CORRECTION)
- QUADRATIC FAILURE CRITERION

Figure 8 · Analysis Results Reporting Capabilities