An Integrated FEA and Design Optimization System for Composite Structures

Craig S. Collier, P.E.
Collier Research & Development Corporation
45 Diamond Hill Road
Hampton, Virginia 23666-6016
e-mail: ccollier@widomaker.com

Abstract

The benefits of a software package which couples with the MSC/NASTRAN^{TM} FEA program is presented. The software, called HyperSizer^{TM}, performs panel and beam structural sizing optimization with metallic and composite materials. Applications in the aerospace and transportation industries are presented. The aerospace application, an X-34x experimental vehicle, emphasizes capabilities and accuracy required for a high speed flight, thermomechanical environment. The transportation application emphasizes practical and inexpensive capabilities suitable for a cost driven, manufacturing oriented environment.

Published with permission by The MacNeal-Schwendler Corporation for the MSC 1996 World Users' Conference Proceedings.

1 Introduction

This is an application paper that is a follow-up to a previous MSC World User's Conference paper. That paper "Thermomechanical Finite Element Analysis of Stiffened, Unsymmetric Composite Panels With Two Dimensional Models", [1], along with papers [2,3] laid the theoretical ground work for a software program which is coupled with the MSC/NASTRAN FEA package. This paper describes the application of the software and theory to two diverse industries:

- Aerospace: Needs include powerful and accurate capabilities for a complex thermomechanical environment
- Transportation: Needs include practical and inexpensive capabilities suitable for a cost driven market

The software, called HyperSizer, performs strength and stability analyses of panels and beams, automatically sizes structure for thermomechanical environments, and provides analytical weight predictions. HyperSizer is significant due to its generality and ability to be linked accurately with planar finite element analysis (FEA). Non-linear, temperature and load dependent constitutive material data of each composite material's laminate are used to "build-up" the stiffened panel membrane, bending, and membrane-bending coupling stiffness terms and thermal coefficients. These panel data are input into the MSC/NASTRAN FEA program to accurately perform analysis with coarse meshed models. Resulting FEA solved thermomechanical forces and moments are used to calculate strains at any location in the panel permitting orthotropic strength analysis of the laminates and orthotropic instability checks, such as local buckling, of each laminated segment of the cross section.

The method is robust enough to handle panels with general cross sectional shapes, including those which are unsymmetric or unbalanced. New thermal coefficients are introduced to quantify panel response from through-the-thickness temperature gradients.

Formulations of [1,2,3] enable the solution to any applied thermomechanical load combination. A major benefit of being able to accurately formulate stiffened panels with smeared equivalent plate properties is that a coarsely meshed 2-D FEM with a single plane of shell finite elements can be used to analyze complex thermomechanically loaded structures. Traditional methods of formulating equivalent plate panel stiffness and thermal coefficients, though intuitive, are difficult to use for a wide possibility of applications. More importantly they give incorrect results as reported in reference 2 and reference 4. 2-D FEA that uses this formulation correlates very well with 3-D FEA [2,3].

Application of the software to a hypersonic, airbreathing X-34x vehicle [5] is highlighted. The vehicle is an integrated airframe/engine design that has hat stiffened, polymer composite panels. The airframe and engine has over 200 defined structural components and is analyzed and sized for the critical trajectory timepoint load cases. In general, a structural component is identified for each panel bay that spans from ringframe to ringframe.

Application of the software to a sidewall of an ISO cargo container is illustrated. Structural components on the large (20'x8') panel are identified by optimization zones which represent convenient, manufacturable areas of stiffener tapers and fabric layups. MSC buckling solution 105 is used to validate the stiffened panel shear buckling performed with explicit analyses of the sizing optimization software.

2 Theory

References 2 and 3 describe the general thermoelastic theory of any general panel. The basic approach is to extend classical lamination theory to the stiffened cross section, Figure 1. Equation (1) defines the general stiffness terms and equation (2) defines the general thermal coefficients.

$$\left(A_{ij}^{p},\ -2B_{ij}^{p},\ 3D_{ij}^{p}\right) = \sum_{k=1}^{\eta} \frac{(h_{k-1}^{m} - h_{k}^{m})}{S} (\bar{Q}_{ij}^{*})_{k} \left[w_{k} \text{ or } \frac{t_{k}}{\sin\theta}\right] (m=1, m=2, m=3) \tag{1}$$

$$\left(A_{i}^{p\alpha},\ -2B_{i}^{p\alpha},\ 3D_{i}^{p\alpha}\right) = \sum_{k=1}^{\eta} \frac{(h_{k-1}^{m} - h_{k}^{m})}{S} (\bar{\Phi}_{i}^{*})_{k} \left[w_{k} \text{ or } \frac{t_{k}}{\sin\theta}\right] (m=1, m=2, m=3) \tag{2}$$

where S is the distance of the repeating pattern of corrugation and w is the width, t is the thickness, and θ is the angle of a stiffening segment (θ=90° for perpendicular stiffeners). Each stiffness term and thermal coefficient is the summation of all laminate/metallic-sheet segments. In this way, each segment and its shape can be accounted for in any panel concept.

Reference 3 supplements reference 2 by including specific equations that define all of the characteristics of a hat stiffened panel. For instance, twelve stiffness terms are formulated for a corrugated, hat-shaped composite stiffened panel by including the unique characteristics of the closed cells that effects bending-twisting, in-plane shear, and their associated stiffness couplings. Reference 3 compares the formulations with 3-D FEA. All terms had differences less than 1%. Implication of this agreement is that the equivalent plate formulations of stiffness and thermal coefficients implemented in the software, and used in the following applications, provide the capability to quantify the thermoelastic response of any structure with coarsely meshed 2-D models.

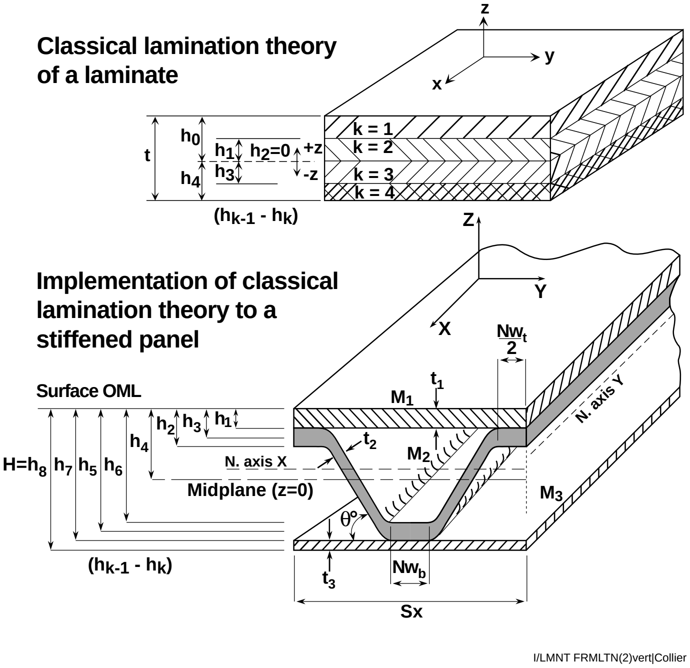


Figure 1 HyperSizer extends laminate formulation to stiffened panels.

3 Implementation of the theory with MSC/NASTRAN

In-plane and through-the-thickness temperature gradients can be correctly applied and solved for anisotropic/orthotropic, unsymmetric, and unbalanced laminates or stiffened panels with a single plane of shell elements with the MSC/NASTRAN FEA program. This is accomplished by including the full complement of smeared equivalent plate stiffness matrices and thermal expansion and bending coefficient vectors in the FEM data deck. Stiffness matrices for membrane, bending, and membrane-bending coupling are entered directly into MSC/NASTRAN with only minor adjustments. Thermal expansion and bending coefficient vectors for membrane, bending, and membrane-bending coupling cannot be entered into MSC/NASTRAN without major adjustments to their formulation, [1].

MSC/NASTRAN Stiffness Terms

The full complement of either laminate or panel membrane [A], bending [D], and membrane-bending coupling [B] stiffness terms can be entered on MSC/NASTRAN MAT2 material bulk data cards. MSC/NASTRAN refers to all of the [A], [D], and [B] 3x3 stiffness terms as Gij [4]. Gij are the 11, 12, 13, 22, 23, and 33 fields of the MAT2 card. A MAT2 card is used for each stiffness behavior. Therefore, to model panel membrane, bending, and membrane-bending coupling stiffness requires three MAT2 cards. The MAT2 cards can represent laminates or smeared equivalent anisotropic plates.

MSC/NASTRAN Thermal Coefficients

MSC/NASTRAN calculates thermal behavior with the general equation

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_{12} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{21} & G_{22} & G_{23} \\ G_{31} & G_{32} & G_{33} \end{bmatrix} \left(\begin{Bmatrix} \epsilon_1 \\ \epsilon_2 \\ \gamma_{12} \end{Bmatrix} - (t - t_0) \begin{Bmatrix} A_1 \\ A_2 \\ A_{12} \end{Bmatrix} \right) \tag{3}$$

which relates stresses (σ_i) to strains (ϵ_i) for an anisotropic homogenous layer. G_{ij} are the Q_{ij} reduced stiffness terms. The A_1, A_2, and A_{12} are the α_i expansion coefficients of the material. The term ($t - t_0$) is its change in temperature. The A_1, A_2, and A_{12}, coefficients also get entered for each MAT2. Likewise, when on separate MAT2 cards, they represent membrane, membrane-bending coupling, and bending thermal response. Unlike the stiffness terms, however, smeared equivalent plate thermal coefficients cannot be entered directly into MSC/NASTRAN. They must be formulated to account for MSC/NASTRAN's particular formulation of thermal forces and moments. MSC/NASTRAN smeared equivalent plate NA_i^α, NB_i^α, and ND_i^α thermal coefficients are:

$$NA_i^\alpha = A_{ij}^{-1} A_i^\alpha \tag{4}$$

$$NB_i^\alpha = B_{ij}^{-1} B_i^\alpha$$

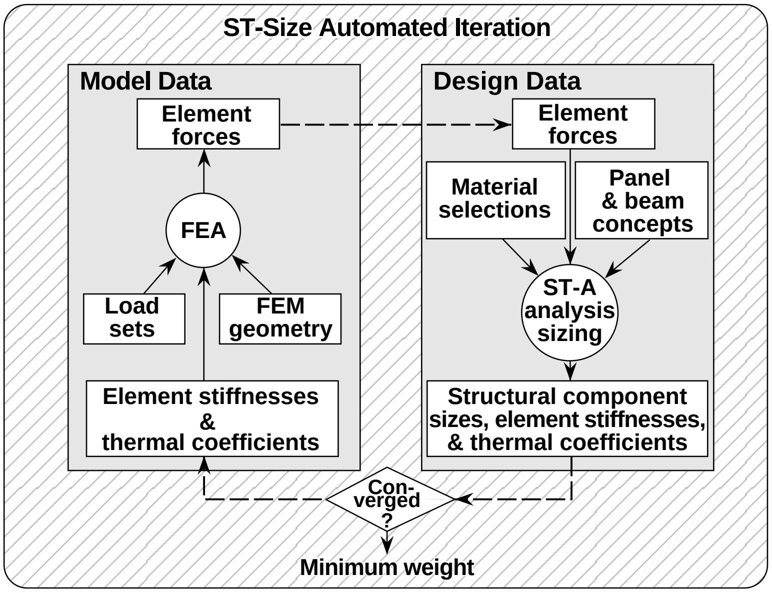
$$ND_i^\alpha = D_{ij}^{-1} D_i^\alpha$$

Their derivation is reported in reference 1.

A change in the panel's bulk temperature is entered in the FEA by supplying the reference temperature on the MAT2 record and the loadcase dependent temperature on the TEMPP1 record. The effect of in-plane temperature gradients is then captured with the model's discretization. Loadcase dependent through-the-thickness gradients are entered on an element basis with the TEMPP1 record. When the MSC/NASTRAN thickness and inertia factor are set to 1.0 and 12.0 respectively, then the load case dependent temperature gradients can be entered directly.

4 Implementation of the process with FEA

The HyperSizer computer code operates in the process depicted in Figure 2. It involves both the analysis and design environments. In the analysis environment, it supplies the NASTRAN FEA with the thermoelastic stiffnesses and thermal coefficients and reads the resulting computed element forces. In the design environment, HyperSizer designs the sections represented by each finite element to the lightest weight that will withstand the computed forces. In doing so, it locates all appropriate data such as material properties and performs numerous failure mode checks.



F/STRCTRL SZNG|Collier

Figure 2 The HyperSizer structural sizing process.

Other major inputs to the design area are element temperatures, material properties, sizing data, and the linking table. Element temperatures are needed to retrieve temperature dependent properties from the materials file. For the most part, sizing data are factors for load, non-optimum weight, strength, and buckling data. More importantly though, they provide a way to "size" the sections as representative structural hardware. For instance, the sizing data contain information such as panel effective buckling lengths that neither the FEM nor any other input contains. The last major input is the linking table which is required to associate the elements of the FEM to their corresponding sections.

Panel and beam sections

Panels and beams have different variables that may be adjusted to give different sizes. Shown in Figure 3 is a trusscore sandwich. By omitting the bottom facesheet, the panel becomes hat stiffened. A hat stiffened panel concept is used frequently in many industries.

A section table for hat or trusscore sandwich panels is generated by varying these dimensions independently within bounds established by design or manufacturing constraints. The resulting combinations of variables as shown in Figure 3 produce sections of varying weight that are ranked in increasing order and stored in section tables. These variables are for the panel's total thickness (H), facesheet and coresheet thicknesses and materials (t_1, t_3, t_2, and M_1, M_3, M_2), corrugation spacing (S_x), coresheet angle (θ), and joining node thickness and widths.

The use of sections provides a powerful capability to the code. By definition, the section is the identical cross-section shape of the structural component it is representing. The detailed dimensional data carried within a section entry allows for examination of all potential failure modes during selection of a workable section. Each section type has its own innate set of equations for strength and stability that are appropriate to it. For instance in addition to the usual strength and stability failure modes, honeycomb panels may fail by intercell dimpling, core crushing, facesheet wrinkling, and crimping. The hat/trusscore panel is analyzed for over 90 potential failure modes. These types of failure mode checks may not be accomplished without the detailed dimensions available from stored section tables.

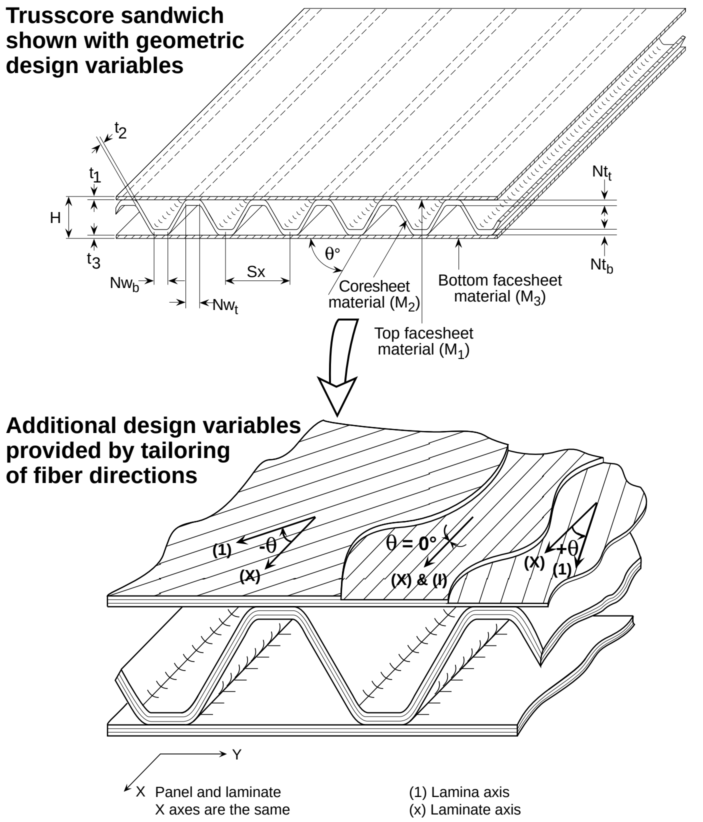


Figure 3 Optimization of composite panel strength and stiffness.

The sizing process begins by selecting the lightest entry in the section table and analyzing it for the computed FEA forces. Each potential failure mode is investigated. Any failure mode that gives a negative margin of safety will cause that section table entry to fail. In that case, the next entry in the table is evaluated until one meets all of the strength and stability requirements. In the sizing optimization process, millions of different sizes, materials, and layups are evaluated for an aerospace application.

When a section size satisfies all load conditions of all the finite elements that comprise a structural component, that component is "sized", and its unit weight is computed from the section geometry and material densities. Each component's unit weight and surface area is computed and summed to determine the lightest vehicle weight.

The dimensions of Figure 3 are not modeled in planar finite element analysis but are instead represented with equivalent plate properties. Both isotropic and orthotropic sections are specified in terms of [A], [D], and [B] stiffness matrices. If another iteration is desired, the dimensions and material properties of each section are converted to stiffness terms and written out to MSC/NASTRAN input data cards. For aerospace applications, different trajectory time points will cause an element to be at different temperatures, therefore MSC/NASTRAN stiffness data is created for every trajectory time point considered. The number of stiffness data sets required per element is referred to as the number of temperature sets.

When material temperature sets are written for all elements, then an FEA solution is executed. Since each material temperature change will necessitate a decomposition of the finite element stiffness matrix, an FEA solution is executed for each temperature set. After all FEA executions are complete, the computed element forces of the multiple load cases are read back into the sizing process. This continues until the user is satisfied that the structural weight has converged to a value.

Use of composite facesheets of sandwich panels can result in different ply counts and lay-ups between upper and lower facesheets when a minimum weight solution is identified. At times the lamina materials might come up different. This may be due to loads, temperature gradients, or a combination of effects. In any of these cases the section is unbalanced and creates the [B] matrix stiffness coefficients and the {B} thermal expansion and bending coefficients for membrane-bending coupling. These unsymmetric stiffness and thermal terms are handled by the HyperSizer code. Note that all stiffened panels are unsymmetric by nature of their shapes.

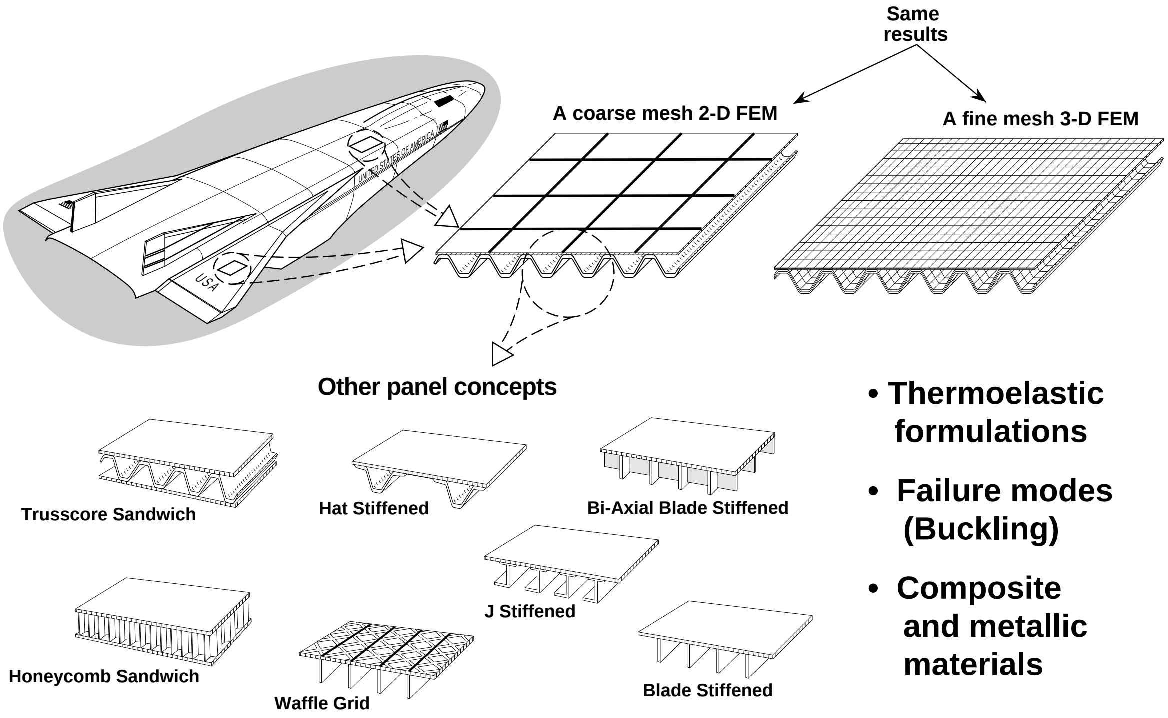
5 Applications

The innovative, accurate, and general 2-D thermoelastic formulations of [2,3] provide a capability to evaluate and simultaneously optimize panel concepts, their actual cross sectional shapes, sizes, thicknesses, material selection, and material layup, see Figure 4. Accurate and efficient smeared equivalent plate stiffness and thermal coefficient formulations of this software are particularly useful for coarsely meshed finite element models of a total structural entity such as an engine or airframe. The formulations and software are useful for analysis and design of lighter weight and higher temperature-capable structure. Two diverse applications are presented below. The first is in aerospace [5] where the needs are for powerful and accurate analyses that handle complex thermomechanical loadings, and the second is for an ISO marine container where the needs are for practical and inexpensive, cost driven commercial design capability.

X-34x Aerospace Vehicle

The process of sizing aircraft structure begins with obtaining mechanical and thermal loads at designated trajectory time points where critical flight events and maneuvers occur. At each time point, the pressure distributions are computed. If the aircraft flies at high speeds, then temperature distributions must also be determined. The pressures and inertial loads (referred to as mechanical loads) have a corresponding set of temperature distributions (referred to as thermal loads) which are used simultaneously in the sizing code for each load case. Each of these load pairs is applied to the FEM and the FEA solution is executed to obtain individual element forces for each load case. These element forces are a major input to the design area.

- ***Material and panel concept*** A hat-shaped stiffened panel concept is used for the fuselage of this X-34x design. A graphite/epoxy (Gr/Ep) fiber-reinforced, polymer composite material is used for the airframe fuselage, wing, and internal structure. Gr/Ep is a high performing, low density material with low coefficient of thermal expansion (CTE). These characteristics make the material particularly beneficial to hypersonic aerospace vehicles which undergo extreme temperature differentials. The in-service, structural temperature of this X-34x design ranged from -400° F to 200° F.



I/ST-SZ ACCRTLY|Collier

Figure 4 HyperSizer accurately analyzes all panel concepts with FEA.

• ***CAD Surface Data*** CAD surfaces were translated to the IGES format and then input into the MSC/P3 FEM modeling system. From there the internal structure was defined and then all surfaces meshed. All details of the structure were included such as the wing spindle, engine support trusses, and the 1" gap between the payload bay door and frame.

•***Finite Element Model*** Finite element analysis (FEA) is used to compute the thermomechanical load paths of the integrated airframe/engine structure. The FEM has 4083 shell elements, 845 beam elements, and 3572 grids. The element normals have a consistent direction for convenient pressure load definitions. All surface shell elements have their material angles aligned with the fore and aft direction of the vehicle. Beam elements have their primary bending strength axes aligned with shape control members, ribs, spars, etc to provide the most structural efficiency. Many details are included such as the payload bay door latches that are modeled with RBE2 elements that allow grid point degrees of freedom to be toggled on or off.

•***Applied Aero and Propulsive Pressures*** Computed data from the full flight trajectory were examined to determine critical events. By scanning trajectory dependent vehicle flight accelerations, dynamic pressures, and fuel mass remaining, it was concluded that Mach 6.3 and 3.0 would clearly be the controlling timepoints. All of the aero, propulsive, rocket, thermal, fuel burnoff, etc. data were then computed and applied to the FEM at these two Mach numbers to provide timepoint consistent load cases.

- ***Loads Mapping*** The computed airframe and engine pressures were converted from the aero and propulsive analytical models to the structural model. Since the mesh densities and grid point locations are different for the aero and structural models, a process called loads mapping was performed to interpolate the data from one to the other. In this case pressures from the aero model were mapped to the structural model. Since the computed trajectory, aero loads, and propulsive loads are computed separately, their superposition onto the structural model will usually cause a load imbalance. That is, the net summation of applied loadings are not in balance. If substantial, another process called loads balancing is performed. For this application, thirteen surfaces were identified on the vehicle, and in an automated fashion, a combination of pressure scale factors, per surface, were computed to balance the loads.

- ***Structural Temperatures*** A large portion of the vehicle is in contact with hydrogen fuel causing the structure to be at -400°F. Other parts of the structure, such as the wing and forward fuselage are at much higher temperatures. At some locations of the integrated airframe/engine, the through-the-thickness temperature gradient reaches of 330°F/in. and was accurately captured with a single plane of finite elements.

- ***Masses*** The MSC/NASTRAN CONM2 data entry format was used to input the mass distribution into the FEM. The mass and mass location for the fuel, tanks, oxidizer, rocket, nose landing gear, main landing gear, pay load, subsystems, active cooling heat exchangers, insulation, and thermal protection systems were input into the FEM either as concentrated masses or distributed over appropriate surfaces. The reduced fuel mass at Mach 3 and 6.3 was accounted. The resulting mass of the FEM was verified to equal the mass projected for performance analyses. On the first pass, the FEA calculated the FEM center of gravity to be within 1.2 % of the CAD calculated CG.

- ***Sizing Input Data*** HyperSizer is tightly coupled with FEA, and in particular with the MSC/NASTRAN FEA package. Most of the HyperSizer input is generated by the FEM building process. The benefit is that little additional input data is required. Original data to be generated is for material files, material layups, structural component specifics such as sizing factors and buckling spans, and ranges of allowed panel shape and size variables.

- ***Structural Components (optimization areas)*** To effectively optimize a large structural system, such as a hypersonic vehicle, the FEM must be categorized into groups of finite elements that form areas for optimization and design The HyperSizer process begins by identifying optimization areas as structural components. The structural components represent the smallest practical manufacturable piece of hardware. That is, they represent areas that should not vary in panel concept, material layup, shape, or size. For the X-34x aeroshell, the smallest optimization areas are the panel bays between fuselage shape control members, as depicted in Figures 5,6,7. As seen in the FEM illustrations, many elements belong to and represent a single component. Over 200 structural components were identified for this X-34x application. Figure 5 shows the structural components on the top, Figure 6 shows the bottom, and Figure 7 shows some of the primary internal components.

- ***Results*** HyperSizer generates numerous types of output data that enables the graphical representation of analysis and design results. These include, critical failure modes, controlling load cases; and optimum unit weights, panel concepts, materials, layups, section dimensions, thicknesses, and stiffener spacings. The ability to access all of the critical design results is key to the user's understanding of the structural response. X-34x sizing results are not shown in this paper.

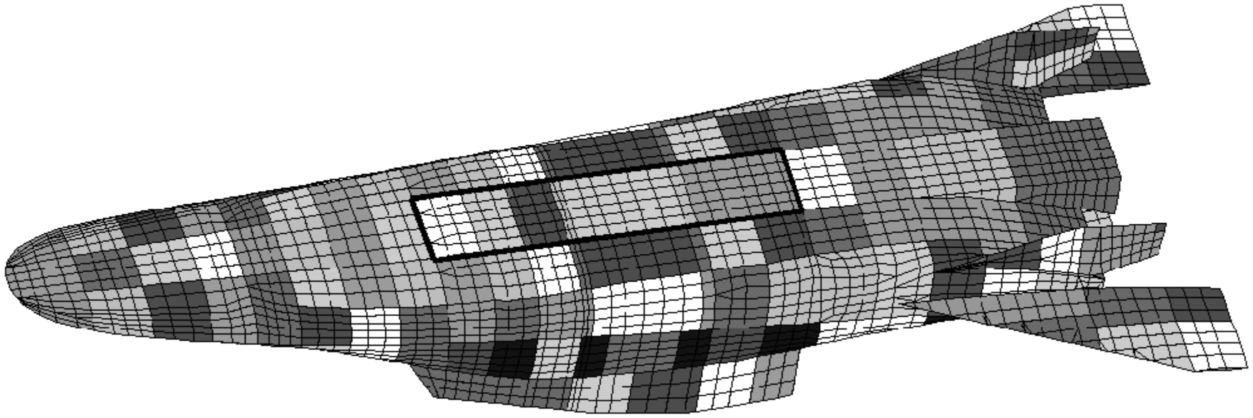


Figure 5 Top view of an X-34x vehicle. The color distribution represents the grouping of elements that belong to structural components. Each color square represents a different component. The components are in effect optimization zones.

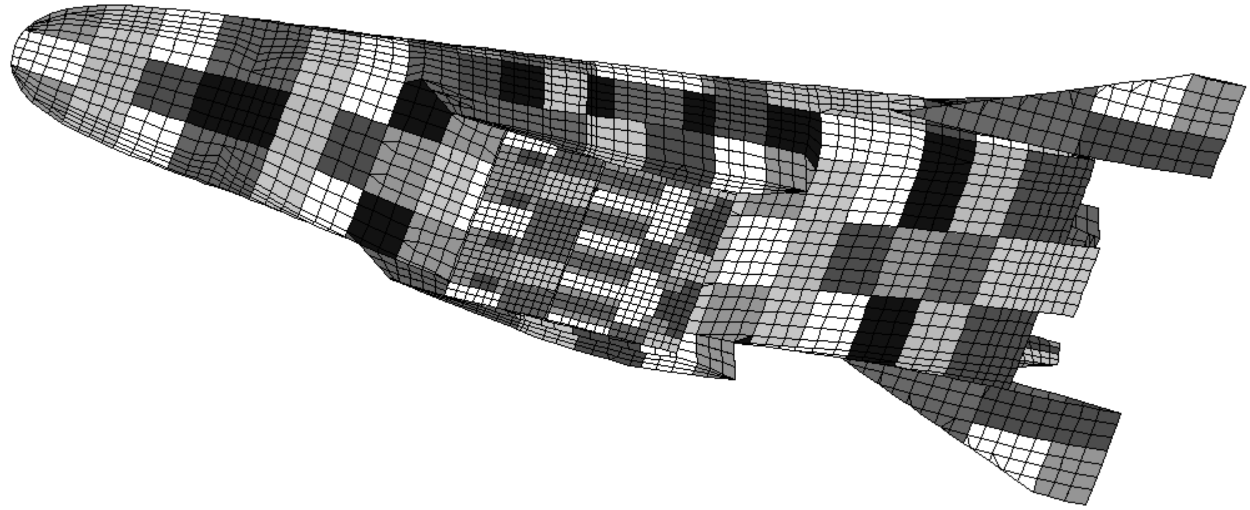


Figure 6 Bottom view of an X-34x vehicle.

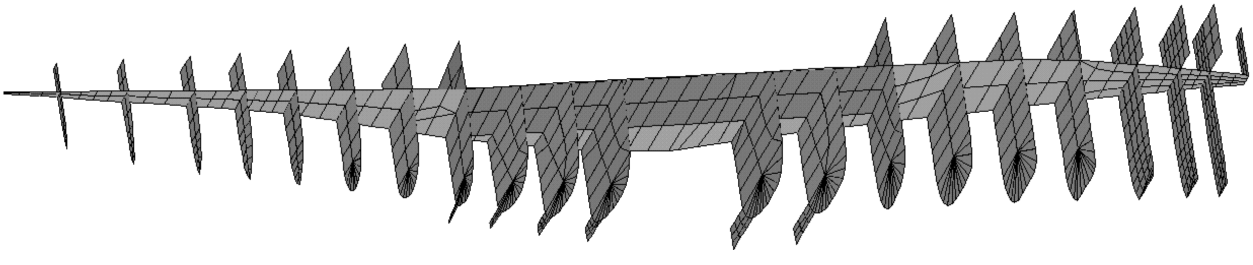


Figure 7 Structural components of the vehicle internal structure. ½ FEM shown.

ISO Cargo Container

The HyperSizer computer program that effectively couples FEA with explicit sizing methods was used to optimize the diverse metallic and composite structural panels of an ISO marine container, see Figure 8. An extensive series of strength and stability analyses were performed to evaluate different panel concepts and material selections in the attempt to find a better and lighter weight structural system.

- ***Loadings*** Structural optimization of a 20'x8' ISO container sidewall was performed for the three loadings it must withstand to meet ISO (International Standards Organization) specification. The first loading is a longitudinal racking load which induces partial picture frame shear. The second loading is a uniform distributed pressure of 0.6Pg which causes panel bending. P is the payload and g is gravity. The third loading is a uniform distributed floor loading of 2Pg, which induces in-plane shear similar to that of a short beam. In depth understanding of sidewall response upon the different loadings with different panel concepts is essential for optimization. All three loadings are analyzed with FEA as part of the automated structural sizing optimization and material tailoring process.

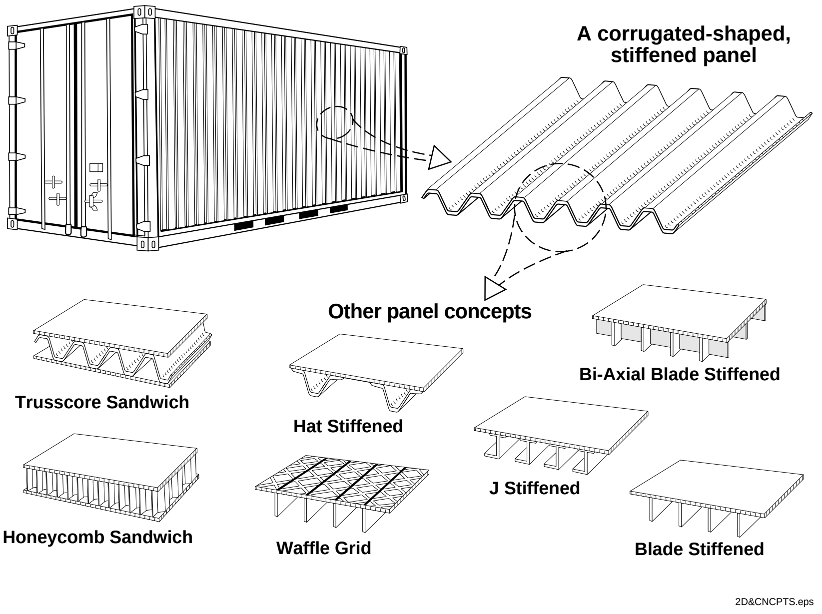


Figure 8 Application to an 8'x8'x20' ISO marine cargo container.

The ISO longitudinal racking test loading shown in Figures 9 and 10 is intended to ensure that container designs have enough shear stiffness in the sidewall to prevent the top translating longitudinally more than 1" from the bottom. External dimensional preciseness is required to guarantee that corner fittings are, and will remain, in proper location during handling and loading. However, this test also requires that a check be done of the material’s shear strength and that the panel does not shear buckle. Shear buckling FEA eigenanalyses were performed, Figures 14 and 15.

- ***Panel and beam sizing optimization*** Figures 9 and 10 show the linear elastic response of the sidewall for the longitudinal racking loading. Figures 9 and 10 were generated to study the effects caused by different panel concepts. The accordion effect of the corrugated sidewall of standard steel ISO containers causes them to be mostly determinate and possible to compute by hand. However, flat facesheets designs, such as sandwich and hat stiffened panels cause the sidewall to be indeterminate as seen in Figure 9. FEA is required to predict this behavior. Membrane stiffness of flat facesheet panel concepts cause load concentration at the corner fittings where the racking load is applied. This concentrated force can be effectively handled by tailoring the composite material, or reduced by providing stiffer corner posts. Figure 9 is a design with actual corner posts. Figure 10 is an analysis done with corner posts that are much stiffer than the corner posts of Figure 9. The stiffer corner posts reduce the concentrated forces. The optimum balance between corner post rigidity and panel type, material, and size are determined with the HyperSizer panels and beam optimization.

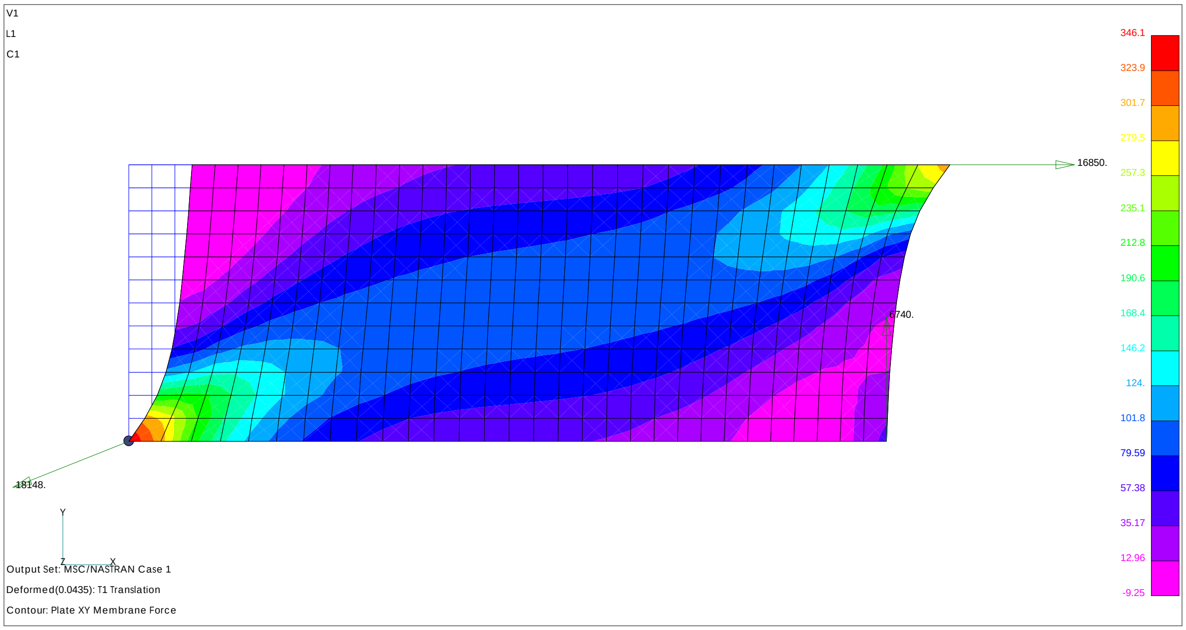


Figure 9 In-plane shear force due to racking of the sidewall. ISO longitudinal rigidity test. Flat facesheet side wall panel with normal corner posts.

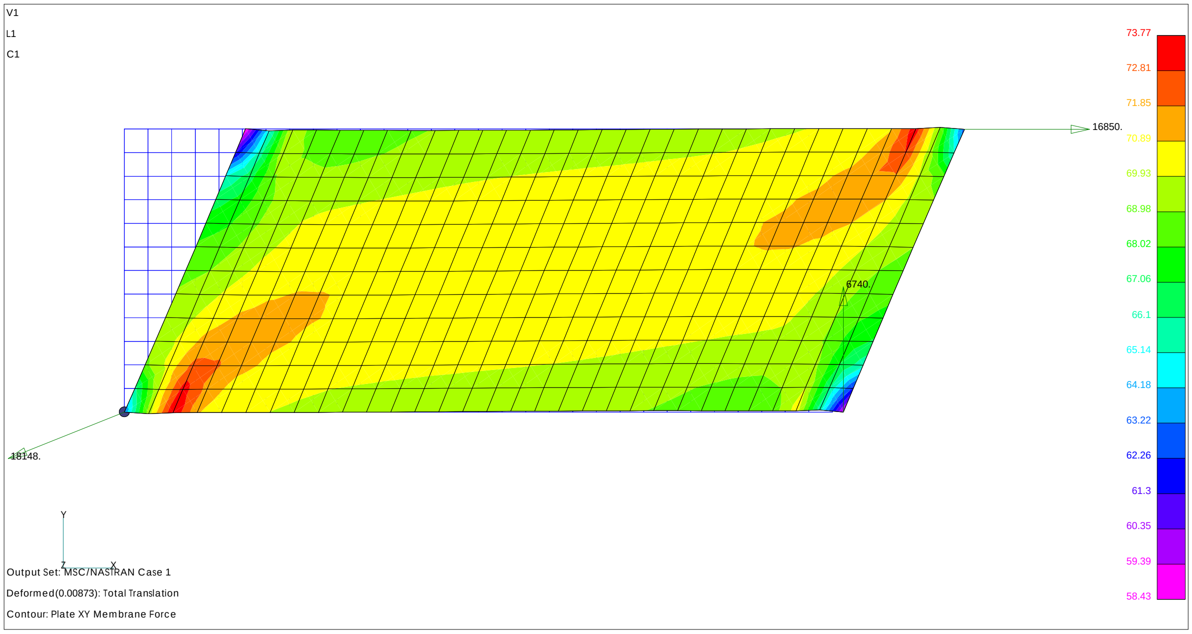


Figure 10 In-plane shear force due to racking of the side wall. ISO longitudinal rigidity test. Flat facesheet panel with very stiff corner posts.

- ***Results*** The sidewall sizing and material optimization clearly shows that an 80% lighter design is possible for the sidewalls when they are made with another panel concept and with an advanced composite material. The new weight is 105# versus the same sidewall made with steel at 545#,Table 1. This considerable weight savings is based on extensive strength and stability optimization. In an automated fashion, HyperSizer was able to identify an optimum design made from a significantly less costly composite material which weighed 200# per sidewall. The less costly material also provides increased damage tolerance.

Table 1. HyperSizer computed weight savings.

Material	**Weight**
Original, corrugated steel design	545 #
Composite with E-glass fiber	200 #
Composite with graphite/carbon fiber	105 #

Figure 11 graphically shows the structural component definition of the ISO container sidewall. These components represent the optimization zones that HyperSizer will size in the attempt to find the entire sidewall lowest weight. These components are regions which can be manufactured to different sizes and composite material layups.

Figure 12 illustrates the margin-of-safety for each structural component. A negative margin indicates a failure of the structure and is unacceptable. For instance the component may buckle, cripple, or be over stressed. A large positive value is also unacceptable because it indicates that it is too conservative and therefore heavy. The ideal margin is just slightly positive. The sidewall margins of Figure 12 go from 0.0 to 0.098 with all components except one with margins less than 0.05 indicating structural efficiency.

Figure 13 depicts the sidewall unit weight distribution. Unit weight (lb/ft^2) plots are useful for determining where on a structure weight is high and low. In the case of the sidewall, the unit weight varies from 1.13 to 1.55. The heaviest area is located at midspan close to the container endwalls. This is an area of panel shear buckling due to floor loading. The next heaviest area is at midspan of the sidewall center. In this area of the container, the dry cargo bears against the sidewall during ship rocking motions. The lightest areas are at the top and bottom of the sidewall center where neither shear buckling nor panel bending is a driver.

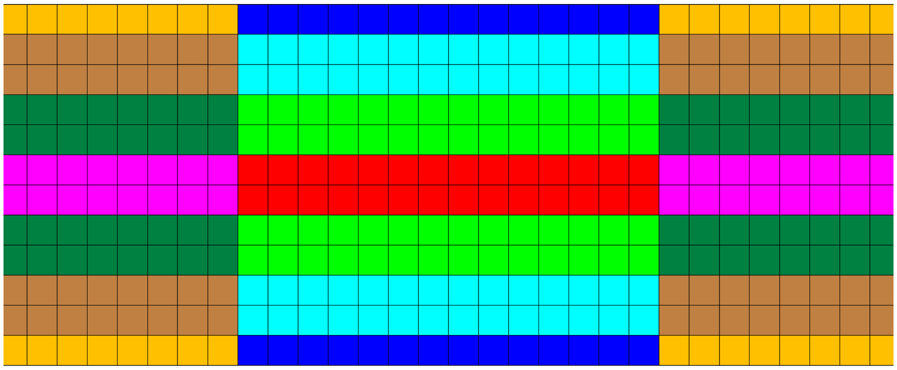


Figure 11 Structural components (optimization zones) of the ISO container sidewall.

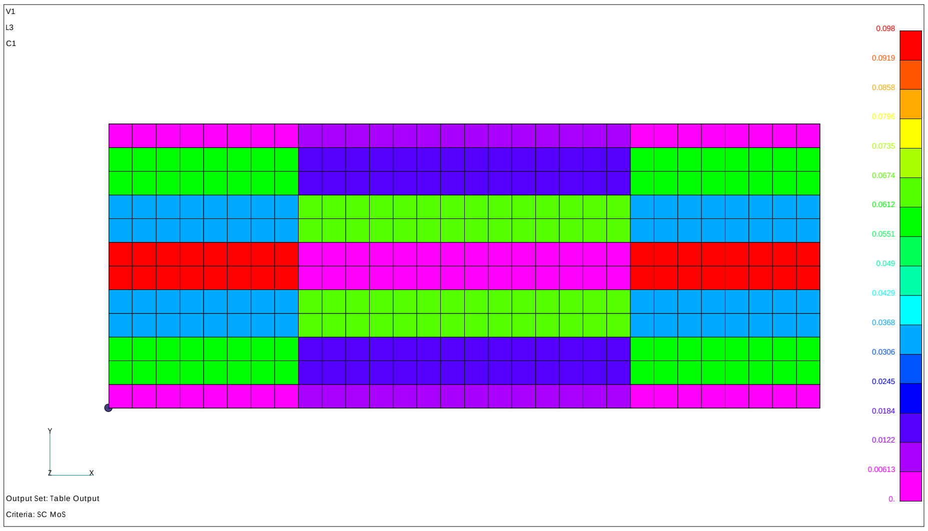


Figure 12 Margin-of-safety of each structural component of the optimized ISO sidewall.

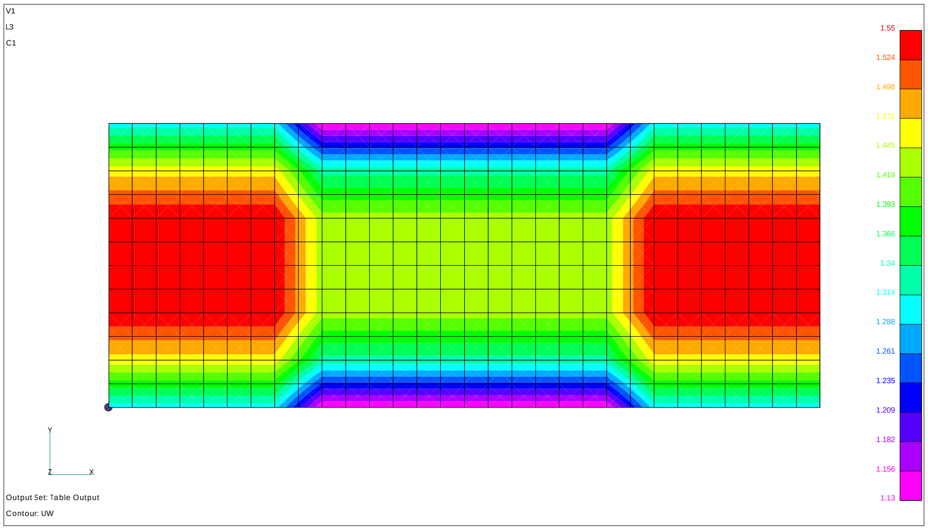


Figure 13 Unit weight of the optimized ISO container sidewall

• ***Verification*** A dominant failure mode of shell structure such as the large panels of ISO marine containers is buckling instability. Accurate analysis and optimization requires the ability to account for these possible failures. They are more difficult to quantify than material stress allowable checks of fully stressed designs. Shear buckling is particularly difficult to quantify by hand methods. After the minimum weight, structurally optimized panel is determined with the optimization software, a verification of the panels structural integrity was performed using the MSC/NASTRAN FEA eigenvalue capabilities. Shown in Figures 14 and 15 are the buckled mode shapes of a steel corrugated panel concept. These analyses correlated well with the closed form methods implemented in the HyperSizer optimization software.

Floor loading is mostly determinate for typical steel corrugated panel designs. However, for the more efficient hat and sandwich panels, the floor loading condition causes compression in the flat panel facesheet. This compression couples with the shear to cause compression - shear buckling interaction. This complex response is also handled with the HyperSizer sizing and material optimization methods.

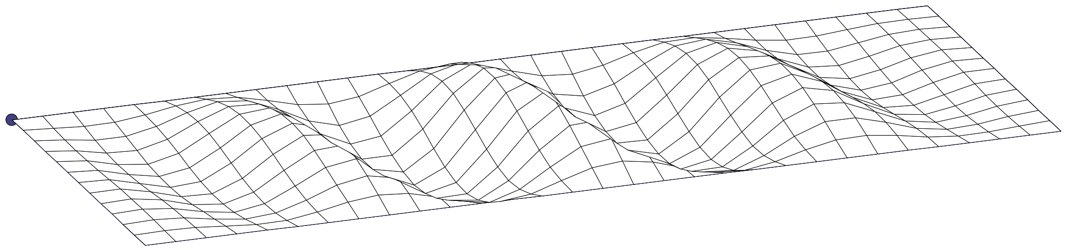


Figure 14 Lowest buckling mode for a corrugated stiffened container sidewall upon racking load. The diagonal tension fields are visible.

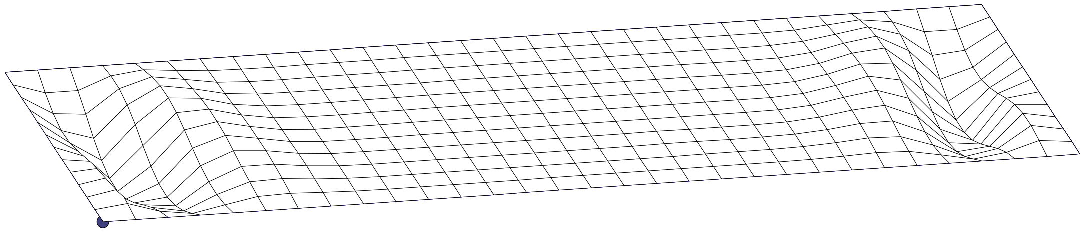


Figure 15 Lowest buckling mode for a corrugated stiffened container sidewall upon uniform floor loading. Note that the buckles are located at the side wall ends where the in-plane shear force is higher.

6 Conclusions

An effective coupling of FEA with explicit sizing methods provides accurate and efficient means of optimizing diverse metallic and composite structures. HyperSizer, the software package that was developed based on this coupling methodology, is presented with examples of its use in two (2) applications: aerospace and transportation. HyperSizer puts the design optimization theory into practice in a way that is powerful, accurate, cost-effective, and fast. It is capable of handling complex thermomechanical structural systems as encountered in the aerospace industry, as well as being able to meet the needs of general commercial products whose designs are highly cost driven. HyperSizer is shown in this paper to accurately and efficiently reduce the weight of an existing ISO shipping container sidewall panel from the original steel weight of 545 lb to the composite material weight of 105-200 lb. In addition to a lighter weight design, the design is less costly in the long term and will not corrode, as current steel containers do, particularly in marine environments.

7 Acknowledgments

‣Portions of this work were performed for the Hypersonic Vehicle Office/Systems Analysis Office under NASA contract NAS1-19000 at Langley Research Center in Hampton, Virginia.

‣Portions of this work were performed for the Naval Facilities Engineering Service Center - located in Port Hueneme, California under NFESC contract N47408-95-C-0224, “Innovative, Collapsible, NFESC Container Technologies”.

8 References

1 Collier, C.S., Spoth,K.A., and Grassi,G.C., "Thermomechanical Finite Element Analysis of Stiffened, Unsymmetric Composite Panels With Two Dimensional Models", The MSC World Users' Conference, Arlington, VA, May 24-28, 1993, Paper No. 31

2 Collier, C.S., "Stiffness, Thermal Expansion, and Thermal Bending Formulation of Stiffened, Fiber-Reinforced Composite Panels", AIAA/ASME/ASCE/ AHS/ASC 34th Structures, Dynamics, & Materials Conference, La Jolla, CA, April 19-22, 1993, Paper No. AIAA-93-1569

3 Collier, C.S., "Thermoelastic Formulation of Stiffened, Unsymmetric Composite Panels for Finite Element Analysis of High Speed Aircraft", 35th AIAA/ASME/ASCE/AHS/ACS Structures, Dynamics, & Materials Conference, Hilton Head, SC, April 18-20, 1994, AIAA paper 94-1579

4 Collier, C.S., "Structural Analysis and Sizing of Stiffened, Metal Matrix Composite Panels for Hypersonic Vehicles", AIAA 4th International Aerospace Planes Conference, Orlando, FL, December 1-4, 1992, Paper No. AIAA-92-5015

5 Lockwood, M.K., Hunt, J.L., Petley, D.H, Collier C.S, et al, "Design and Analysis of a Two-Stage-to-Orbit X-34x Airbreathing Hypersonic Vehicle Concept", 32nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, July 1, 1996, AIAA paper 96-2890

TRADEMARKS