8th AIAA/USAF/NASA/ISSMO
Symposium on Multidisciplinary
Analysis and Optimization
6-8 Sept. 2000 Long Beach, CA

A00-40168

AIAA-2000-4933

AN INTERFACE BETWEEN SDRC I-DEAS AND THE GENESIS STRUCTURAL ANALYSIS AND OPTIMIZATION CODE

Iku Kosaka*, Christophe Charpentier†, and Brian C. Watson*
Vanderplaats Research and Development,
1767 S. 8th St. Suite. 100
Colorado Springs, CO 80906
(ikosaka@vrand.com)

ABSTRACT

An interface between SDRC I-DEAS and the GENESIS structural analysis and optimization code is developed and discussed in the context of design optimization applications. The interface creates almost all design data which GENESIS requires to perform size, shape, and topology optimization in the I-DEAS environment so that designers can easily create optimization design data. An application problem is solved using unique features of these tools. First, the problem is defined and solved as topology optimization to obtain preliminary design. Then the initial design is created using the first solution and CAD tool. Finally, the design is refined with shape optimization.

INTRODUCTION

Although the validity of design optimization has been widely acknowledged in the engineering community, it is not widely used as a standard design tool for various reasons. Non-technical reasons include difficulty in creating design data and lack of time to educate engineers to perform optimization. Those engineers spend most of their time modifying and analyzing their designs and making iterative changes, rather than modifying designs using optimization technique in one iteration. However, since there is enormous pressure to shorten the "time to market," while enhancing the quality of product at the same time, those attitudes are gradually changing and/or are under pressure to be changed. It is the researchers' and software developers' responsibility to keep refining the state of the art of optimization technologies and to provide easy to use, user friendly optimization tools, so that as many engineers as possible turn their attention and curiosity to this technology.

In recent years, efforts to provide a pre- and post-processing design optimization capability, either as an add-on interface or as a built-in for CAE tools, have been expended by several vendors. Altair computing has expanded Hypermesh [1] to supply design optimization pre- and post-processing capability to their product, OptiStruct. MSC/Patran [2] has been integrated with MSC/NASTRAN [3] and MSC/CONSTRUCT, and also with GENESIS [4] using an add-on interface [5]. Femb/Genesis [6], a special version of FEMB [7] by Engineering Technology Associates, has such capability for GENESIS.

The main objective of this development is to provide unified design tool, including drafting, pre- and post- processing for both analysis and design, and optimization, using SDRC I-DEAS [8] and GENESIS, so that not only CAE designers but also CAD designers can easily modify their designs in systematic fashion.

The purpose of this paper is to demonstrate a design process for realistic design application problems using the tool developed here. We will first briefly introduce the GENESIS and SDRC I-DEAS programs, then discuss how the interface program is integrated into the SDRC I-DEAS environment. Then the capabilities of the interface are described along with design application problem. The paper is concluded with a brief summary of this development and the design applications presented.

GENESIS

GENESIS is a fully integrated analysis and optimization program instead of a simple coupling of the two. The entire program was designed to use second generation approximation concepts and well-established optimizers so that faster convergence and efficient and robust performance can be obtained in optimization process.

* Senior R&D Engineer, Member AIAA
† Mechanical Engineer

GENESIS can simultaneously solve both member dimension (sizing) and coordinate location (shape) optimization problems and a wide range of options are available to define the design problem. For example, design variables may be individual member dimensions and/or grid locations, or may be linear or nonlinear combination of these. To create smooth variations in structural dimensions and shapes, a semi-automatic creation of basis vectors, DOMAIN, can be used. To simplify the task for designing frame or plate structures, a library of common elements is available. In addition to that, users can provide their own subroutine to calculate bar section properties as functions of the design variables or they can use "synthetic" functions to define the section properties at run time.

Any response that is calculated, including mass, volume, stress, strain, displacement, temperature, vibrational frequency, system inertia, and strain energy can be used as the design objective and constraint. Additionally, users can create their own "synthetic" function of the design variables and other responses to specify the design objective or constraints. Geometric quantities such as length, distance, angle, area, and volume can be designable responses. In this way, users can impose a wide range of design specifications to their design problems and can perform very detailed and flexible design optimizations.

In addition to the shape and size optimization, GENESIS also solves the material layout (topology) optimization as a separate problem. In a typical design process, it is used to identify the preliminary or conceptual design for further refinement using size and shape optimization. With topology optimization, any property that references one isotropic material (MAT1) data can be designed and displacement, strain energy, natural frequency, and mass fraction can be used as responses. GENESIS will control the "checkerboarding" solutions by default and a topologically symmetric solution can be enforced as an option.

Available analysis options in GENESIS version 6.0 are the static, normal modes, heat transfer, direct and modal dynamic responses, and buckling, all of which can be used in shape and sizing problems. For topology optimization, static and normal mode analyses are currently usable. Optimization will be performed using the well-established DOT optimizer for moderate sized optimization problems and the BIGDOT optimizer can be used for large scale problems.

SDRC I-DEAS

SDRC I-DEAS is a very comprehensive computer aided design tool, including drafting, analysis (simulation), and optimization capabilities. Its drafting capability is especially powerful, allowing users to design complicated models in a flexible and easy way. Available analysis options are linear statics, normal mode, constrained mode dynamics, buckling, heat transfer, potential flow, and nonlinear statics, yet only linear statics and normal mode analyses can be used in optimization. Although several design optimization options are built into I-DEAS, applicable applications are limited to entry level optimization problems. One attractive optimization capability in I-DEAS is that users can directly optimize CAD parameters as design variables. However, since optimization size, usable analysis options and available responses are limited, a wide range of optimization problem cannot be performed.

The main goal of this development is to provide an interface to link powerful CAD and robust optimization packages so that from entry to expert level design optimization problems can be solved in unified fashion.

INTERFACE ARCHTECTURE

Since both GENESIS and I-DEAS are stand alone programs and are not developed in the same environment, a well organized program architecture should be made in advance. One of the key issues is how the interface program can access the FEM data and preprocessing capabilities of I-DEAS. Fortunately, these essential functions can be provided by I-DEAS' open architecture, also known as Open I-DEAS, which allows a third party program to access its database and to use its graphical tools. Through Open I-DEAS, the program collects information from graphics, puts information into graphics, accesses the database, and creates GENESIS design data as if these functions are just like other I-DEAS' functions.

Another important issue worth mentioning here is how the data being created by the interface will be stored. Unlike other CAE tools such as PATRAN, I-DEAS does not have an option to store third party data unless the information is stored as an attribute of a part or of a geometrical entity. Therefore, all of the necessary data is stored as an attribute of the coordinate system in the I-DEAS database.

The interface's main dialog box is launched from the User Application Launcher, which allows a third party program to run, and each interface function is executed from the icon in the dialog box that is independent of the I-DEAS icon panel. The main dialog box for the interface is shown in the Figure 1. The common functionality such as creating design variables, tables, and equations are accessed from relevant functions on the main dialog.

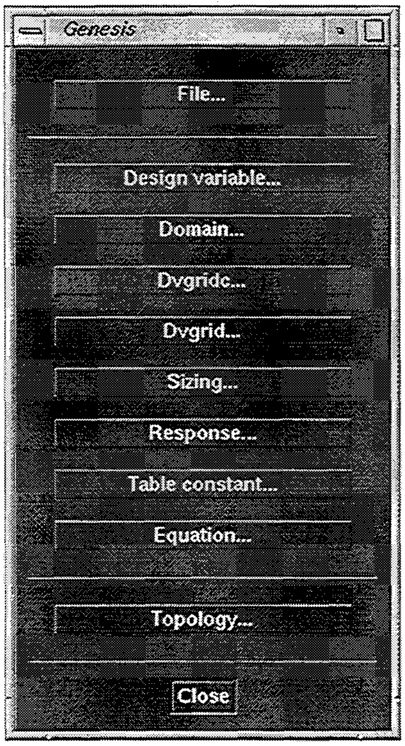


Figure 1. The main dialog box for the interface.

DESIGN OF BRACKET USING TOPOLOGY AND SHAPE OPTIMIZATION

Problem description

A bracket used to attach a strut to a rail is designed using the tools described above. To solve this problem, two independent design optimization problems were defined and solved in sequence. First, we specified the potentially designable space and used topology optimization with a specified amount of material. Then the obtained results were interpreted and CAD (lines and surfaces) data as well as an initial FE model for shape design were created. Finally, shape optimization problem was used with detailed design specifications.

Topology design description

Figure 2 shows the designable space for the topology problem. The solution will be obtained inside of the entire space and certain material must be removed from this space. In this first phase, we focused on obtaining an efficient structure to carry the primary load which is also shown in the Figure 2. Figure 3 shows all of the topology optimization dialogs to specify the GENESIS topology data. The main topology dialog (titled Topology) appears after pressing the "Topology…" button on the main dialog. This has a snapshot of currently defined topology design data, and allows access dialogs for data creation or modification. As one can see, the design goal was placed on minimizing global strain energy subjected to material constraint. This problem was solved for three different values for the material constraint, namely 30%, 50%, and 70%.

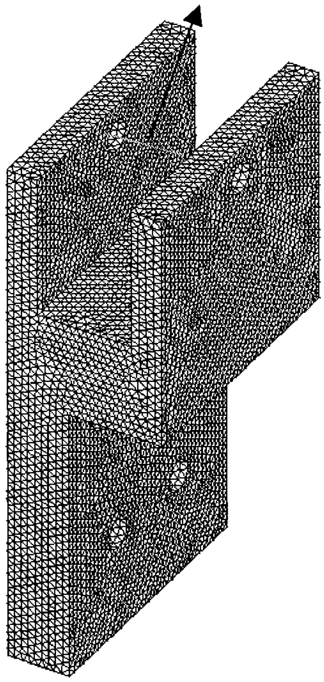


Figure 2. Designable space for the topology problem.

Topology results and design interpretation

Topology optimization results can be notoriously difficult to interpret since they often contain zigzag borders and/or some degree of gray zone. To overcome these problems, and to help designer interpret the solution easier, GENESIS version 6.0 provides the iso-density surface, which will visualize the 3-D structures as FE surfaces. Figures 4, 5, and 6 show the topology design solutions using iso-density surface for each material constraint case. Although the first two solutions (30% and 50% material constraint cases) and the third solution (70% material constraint case) are topologically different, tendencies are very similar. The main difference is the size of the cut out hole, thus the first two cases developed an open hole and the third case developed closed hole. Therefore, the conservative design (the third case solution) was chosen as an initial candidate design for shape optimization, and CAD (lines and surfaces) data was created based on this solution. Since we can incorporate the potentially removable material location from the topology solutions, relatively simple initial design was created and meshed.

Shape design description

In the first phase of design problem, we considered preliminary design solutions using simplified criteria. Here, we will focus on more a detailed and complete design problem. Two important loading conditions are considered to design this bracket. In the first loadcase, 15000N of the point load (same as the topology design problem) was applied to simulate force through an object attached to the bracket. The second loadcase calculates natural frequencies to control vibration or vibrational noise.

To improve the initial design, the following optimization problem is formulated:

Minimize Mass

Subject to von Mises stress < 200MPa

Fundamental frequency > 850Hz

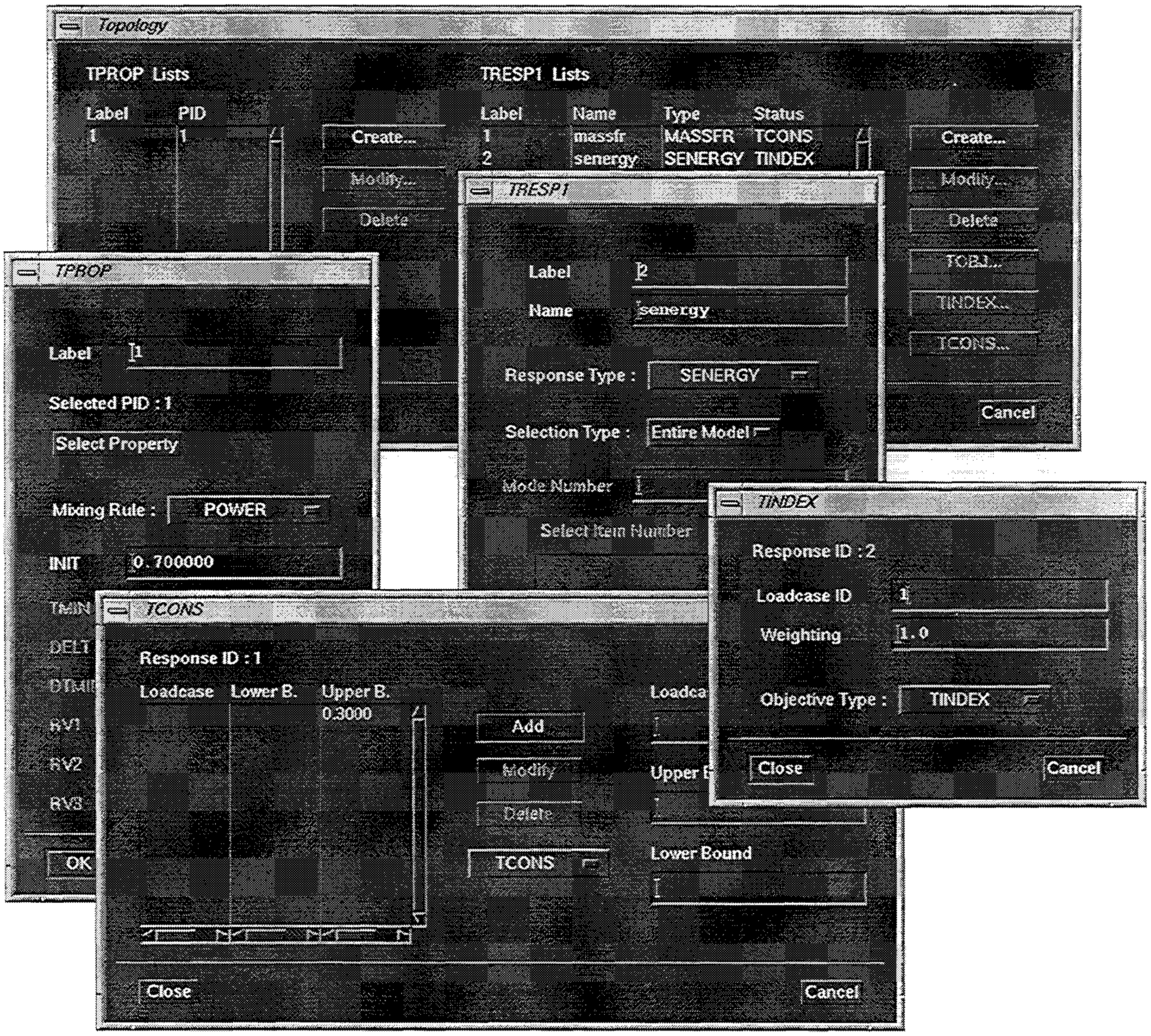


Figure 3. Topology main, TPROP, TRESP1, TINDEX, TCONS dialogs.

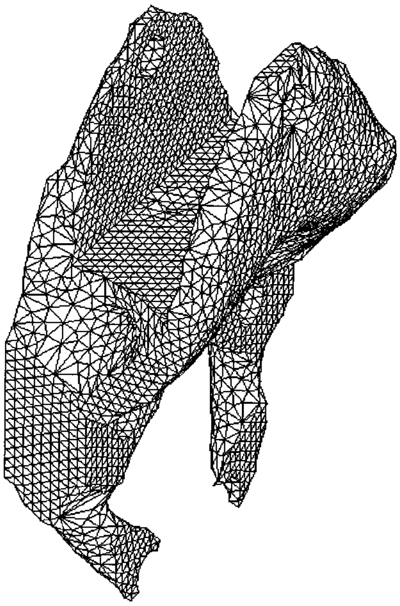


Figure 4. Topology optimization result for 30% material constraint case.

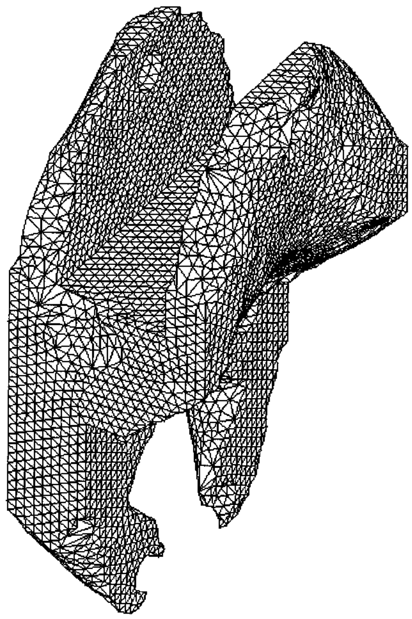
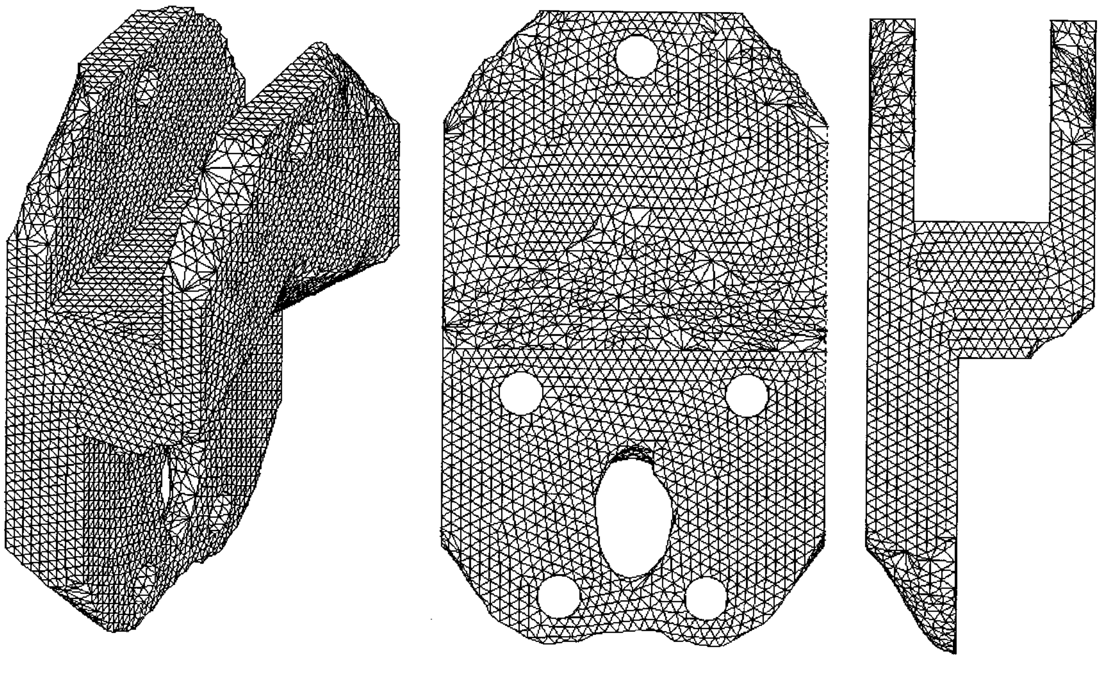


Figure 5. Topology optimization result for 50% material constraint case.



(a) Isometric view

(b) Front view

(c) Side view

Figure 6. Topology optimization result for 70% material constraint case.

Shape changes considered here are to two locations suggested by the topology design results, and are shown in Figures 7 and 8. Figure 7 shows the HEXA DOMAIN and perturbation vectors that are governed by four independent design variables and Figure 8 shows five QUAD4 DOMAINs and four TRIA3 DOMAINs and the perturbation vectors which are also governed by four independent design variables. The description of each design variable and their bounds are summarized in Table 1.

Table 1: Design Variables

(1) E1 : Upper edge	-5 < E1 < 10
(2) E2 : Upper mid-side	-5 < E2 < 5
(3) E3 : Lower edge	-5 < E3 < 10
(4) E4 : Lower mid-side	-5 < E4 < 5
(5) H1 : Hole size	-2 < H1 < 2
(6) H2 : Hole up and down	-2 < H2 < 2
(7) H3 : Hole longitudinal size	-2 < H3 < 2
(8) H4 : Hole lateral size	-2 < H4 < 2

All the shape optimization data can also be created using the interface, same manner as topology optimization. The dialogs to create the DOMAINs and perturbation vectors are shown in the Figures 9 and 10.

Optimization results

All the optimization results including the first and second phase design problems are discussed and summarized in this section. In the first problem, all three topology runs converged between 20 to 25 design cycles and very interpretable solutions are obtained. In Tables 2 and 3, the structural responses of the full solid (100% material) model and of the optimized (with 70% material constraint) model are summarized to compare performance.

Table 2: Full solid model analysis results

Mass	21.537Kg
Max von Mises stress	165.5MPa
First frequency mode	789.3Hz

Table 3: The optimized model analysis results

Mass	15.063Kg
Max von Mises stress	172.8MPa
First frequency mode	842.4Hz

Although the topology optimization took out 30% of the material, the structural performance was not degraded very much. Therefore, we can conclude that most of the unnecessary material can be removed from designable space.

As stated earlier, the above solution was used and a CAD and FE model was created to refine the model. In order to compare the structural performance of each model, the responses of new FE model (Figure 11) are summarized in the Table 4. As one can see, the response values between the final solution of topology model and the initial shape design model are remarkably similar.

Table 4: The initial shape model analysis results

Mass	15.888Kg
Max von Mises stress	170.1MPa
First frequency mode	848.0Hz

Finally, design variable values and the analysis results obtained by the shape optimization are summarized in the Tables 5 and 6.

Table 5: Final design variable values

E1	E2	E3	E4	H1	H2	H3	H4
10	5	10	5	2	-2	-2	2

Table 6: The initial shape model analysis results

Mass	14.649Kg
Max von Mises stress	171.3MPa
First frequency mode	850.0Hz

The fact that all the design variables reached to the side constraint bounds (as shown in the Table 5) indicates that further weight reduction may be possible by choosing the larger side constraint bounds. In this shape optimization problem, side constraint bounds are determined by the DOMAIN size to avoid the mesh distortion, thus, the larger DOMAIN and more transitional DOMAINs will be required to be able to expand them. In addition to that, there are potentially designable parts where topology optimization results also suggested. The final shape solution is displayed in the Figure 12.

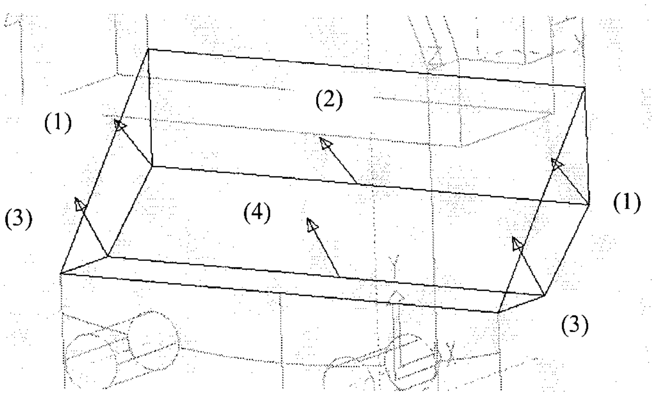


Figure 7. Visual representation of the DOMAIN and DVGRIDC's designing the edge part.

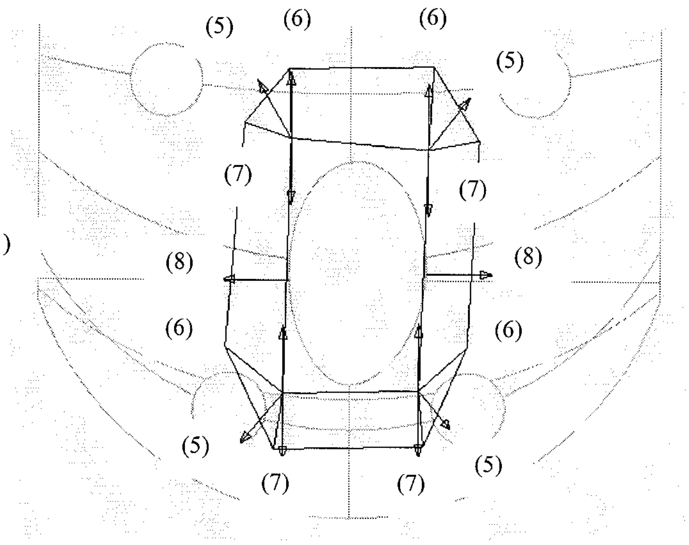
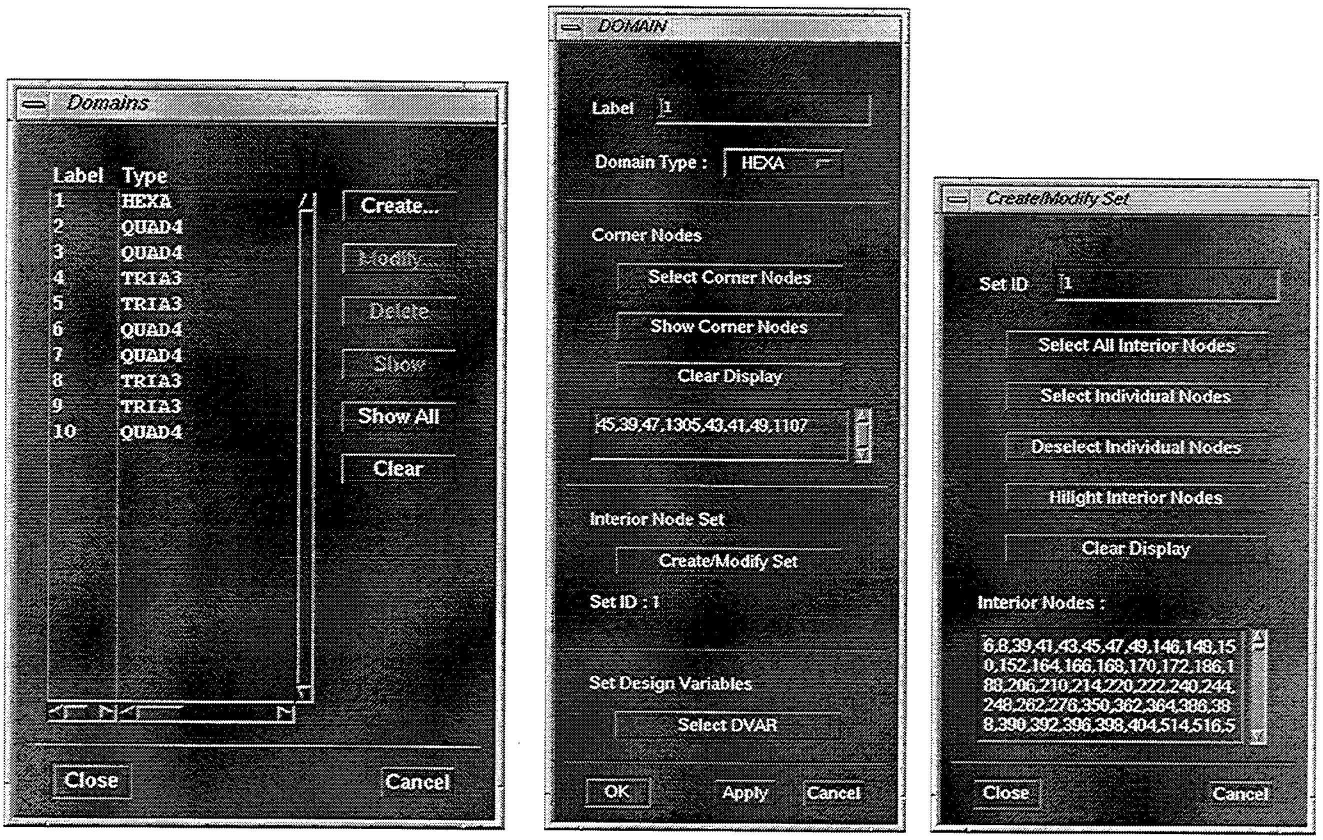
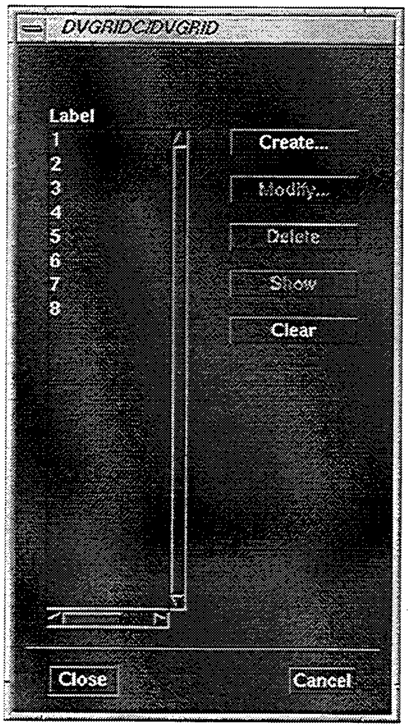


Figure 8. Visual representation of the 9 DOMAINs and DVGRIDC's designing the bottom part.

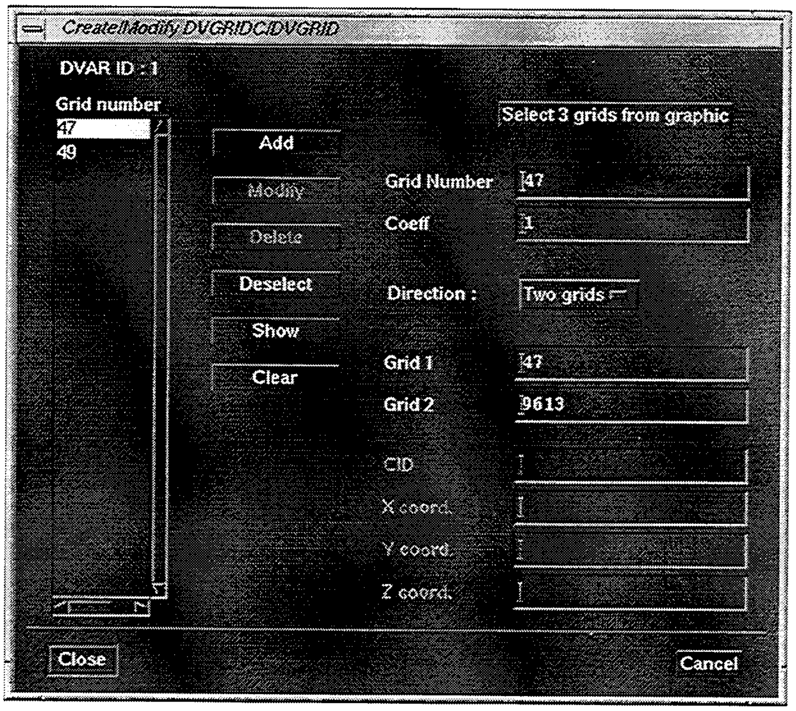


a) Domains main dialog b) DOMAIN subordinate dialog c) Create/Modify Set dialog

Figure 9. Creation of the HEXA type DOMAIN 1.



a) DVGRIDC main dialog



b) Create DVGRIDC dialog

Figure 10. Creation of the DVGRIDC (Perturbation vectors).

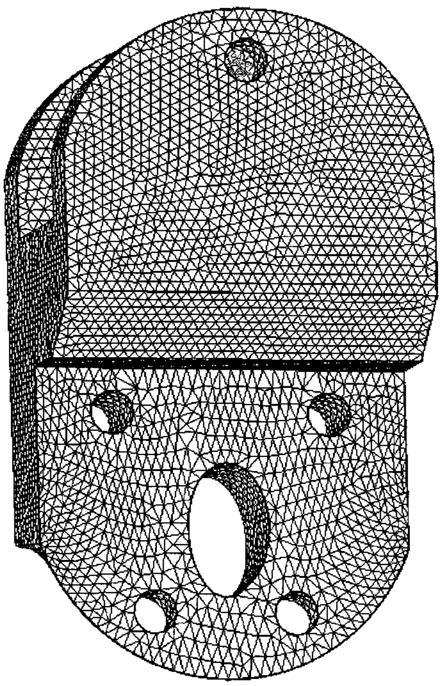


Figure 11. Initial design for shape optimization

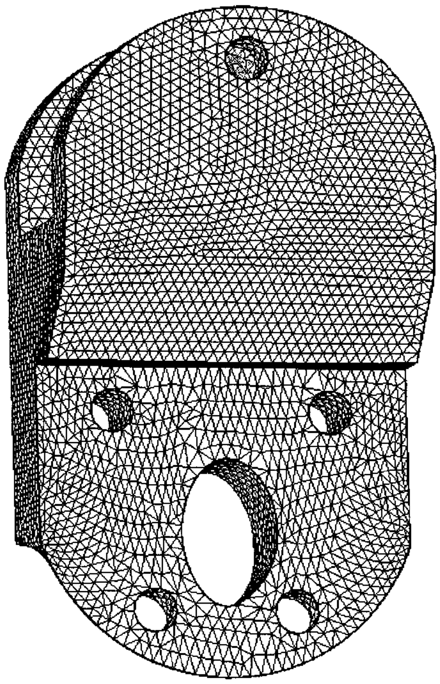


Figure 12. Final solution obtained by shape optimization

CONCLUSIONS

A tool to connect SDRC I-DEAS with the GENESIS structural analysis and optimization code is developed and presented with design application problems. This interface provides the capability to create design optimization data for GENESIS and allows the users to perform a wide range of structural optimizations using these tools.

The results of an application problem shows the advantage of design optimization.

ACKNOWLEDGMENT

The authors thank Mr. Alexandre Azema of IFMA, France for his help on the beginning phase of program development.

REFERENCES

1 *HyperMesh 2.0 Documentation.* Altair Computing, Inc, Troy MI, USA, 1995.

2 *MSC/PATRAN, Version 7.0 User's Manual Vol 1, 2, 3, and 4.* MacNeal-Schwendler Corporation, Los Angeles, California, USA, 1998.

3 *MSC/NASTRAN Version 67 Quick Reference Guide.* MacNeal-Schwendler Corporation, Los Angeles, California, USA, 1992.

4 *GENESIS Structural Optimization Software, Version 6.0 User's Manual.* Vanderplaats Research and Development, Colorado Springs, CO, USA, January 2000.

5 *MSC/PATRAN-GENESIS Interface Version 2.0.* Vanderplaats Research and Development, Colorado Springs, CO, USA, March 2000.

6 *Femb/Genesis Version 2.5 User's Manual.* Vanderplaats Research and Development, Colorado Springs, CO, USA, March 2000.

7 *FEMB Version 26.5 User's Manual.* Engineering Technology Associates, Inc., Madison Heights, MI, USA, 1996.

8 *I-deas Master Series 7, Online Help Bookshelf,* SDRC, Milford, OI, USA 1999.