

# Development of an Optimization Plug-In for GENESIS<sup>®</sup> for Use with Femap<sup>®</sup> Software

Dipankar Ghosh\*, Ph.D  
*Vanderplaats Research & Development (VR&D), Inc.*  
1767 S. 8<sup>th</sup> Street, Colorado Springs, CO 80906  
Phone: (719) 473-4611 x104  
Email: dg@vrand.com

**The ease-of-use is an important consideration in making structural optimization tools successful in virtual product development. Femap software from UGS PLM, Inc. is a widely used pre- and post processing tool for finite element analysis, but it does not offer any design optimization capability yet. VR&D has recently developed an interface to Femap, called Optimization Plug-In for GENESIS (OPG), to facilitate the creation of design optimization data along with the finite element data so that Femap users can perform structural optimization using the GENESIS structural analysis and optimization software, also available from VR&D. The current development supports topology, shape, sizing, topography and topometry optimization. The details of the interface along with some examples are provided in this paper.**

## I. Introduction

This paper will discuss the details of the development of an optimization interface for FEMAP Software. This interface will allow FEMAP users to create optimization data, and then solve it using GENESIS solver. This development was done using FEMAP OLE/COM API. Topology, shape, sizing, topography and topometry optimizations can be performed using this interface. The finite element modeling of the problem can be done using the current capabilities of the FEMAP software. Typically, it involves creating the Nastran analysis data deck using the FEMAP software, and then this interface is used to create the optimization-related data. Once the optimization data is created, it should be included in the bulk data of the analysis, and then run it using GENESIS solver. Once the GENESIS solver finishes the optimization run, the post-processing files could be read in FEMAP for post-processing. While shape, sizing, topography and topometry optimization could be combined in a single run, the topology optimization forms a separate run. Typically, users will perform the topology optimization to find the optimal distribution of material in a given package space. Once the topology optimization creates a preliminary design, it could be further optimized using shape (including topography and topometry) and sizing optimization. The development of this plug-in extends the capabilities of FEMAP for optimization applications. The details of the plug-in with the help of a number of examples will be discussed in the presentation.

## II. GENESIS Solver

GENESIS [1] is a fully integrated finite element analysis and design optimization software package, written by leading experts in structural optimization. Analysis is based on the finite element method for static, normal modes, direct and modal frequency analysis, heat transfer, and system buckling calculations. The well-established DOT<sup>®</sup> and BIGDOT optimizers perform optimization within GENESIS. This process typically requires less than ten detailed finite element analyses, even for large and complex design tasks. No special knowledge of optimization technology is required. Shape, sizing, topography and topology optimizations are the design options available to the user. Shape, sizing, topography and topology optimization typically requires less than ten detailed finite element analyses, even for large and complex design tasks. The topology optimization typically takes twenty or less detailed finite element analyses.

---

\* Product Manager and a Lead Developer, Vanderplaats Research & Development; and AIAA Member.

### III. FEMAP Software

FEMAP [2] is finite modeling and post-processing software that allows you to perform engineering analyses both quickly and confidently. It provides an easy access to CAD and office automation tools. FEMAP can be used to create geometry, and you can also import CAD geometry created using another CAD modeler. FEMAP also provides powerful tools to mesh geometry as well as applying load and boundary conditions. The model created in FEMAP can be exported to a number of finite element analysis solvers, including GENESIS and VR/Nastran, and then the results generated by the solver can be read in the FEMAP for post-processing. Once the results are obtained in FEMAP, a number of tools are available for visualizing and reporting results.

### IV. Optimization Plug-In for GENESIS (OPG)

While the stand-alone FEMAP software can create GENESIS analysis data, it cannot create the design optimization data as required by GENESIS to perform optimization. In order to perform design optimization using GENESIS and FEMAP, FEMAP software should be able to process not only the analysis data but also the design data required by shape, sizing, topography and topometry optimization. VR&D has developed this interface that could be called in from within the FEMAP software to create design optimization data for GENESIS. This will allow users to perform design optimization using FEMAP and GENESIS solver. Using this interface, one can create all the necessary optimization data for solving it with GENESIS. The Figure 1 shows some details of the capabilities of this optimization plug-in.

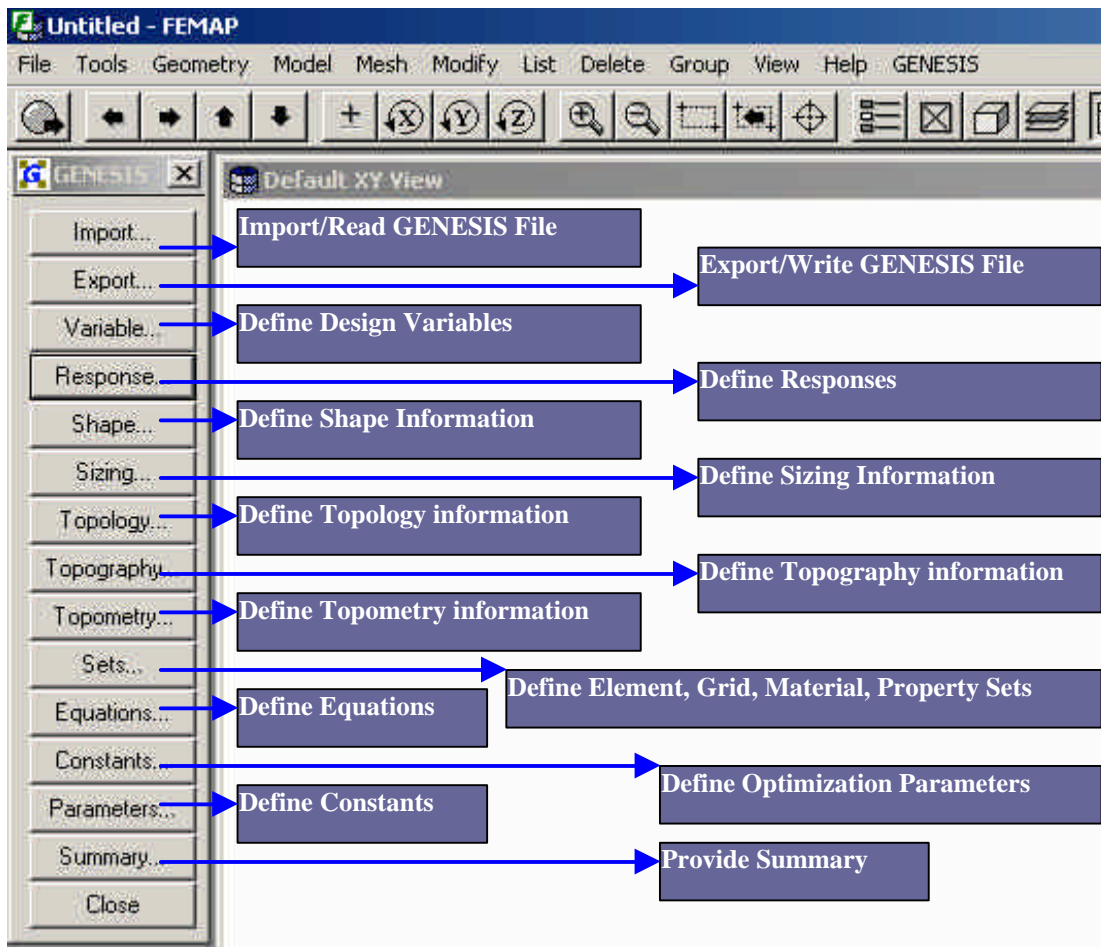
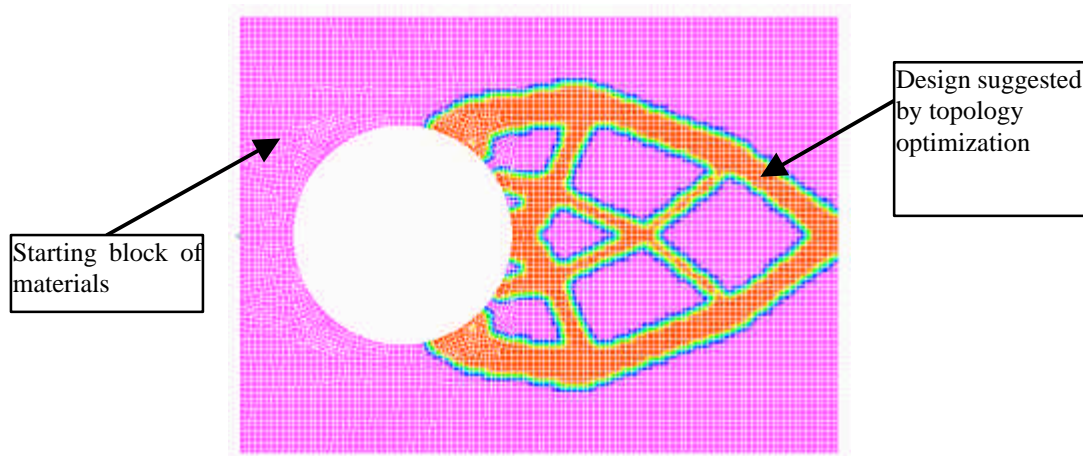


Figure 1: Optimization Plug-In for GENESIS

## V. Optimization Plug-In Capabilities

A. **Topology Optimization** -Topology optimization is used to find the optimal distribution of material in a given package space. Unlike shape and sizing optimization, topology optimization does not require an initial design. Typically, the design starts with a block of material formed by a large number of finite elements and the topology optimization will “take out” from the block the unnecessary elements. The Figure 2 shows an example of topology optimization of the Mitchell truss.



**Figure 2: Mitchell Truss Topology Example**

Topology optimization has a limited number of responses associated with it. These responses are primarily used to create a stiff or light structure. Topology optimization is normally used by design engineers to perform conceptual designs. After the topology optimization is finished, shape and/or sizing optimization can be performed to refine the solution. Topology optimization works by creating design variables associated with the Young’s modulus and density of each element in the package space. The value of the design variable ranges between 0.0 and 1.0, where 1.0 indicates that the element has its normal stiffness and mass, and 0.0 indicates that the element has no stiffness or mass. Topology optimization can only be used with static and eigenvalue load cases. The relevant results are the displacements, strain energy and natural frequency responses. Fabrication requirements such as castability, extrusion and symmetries can be imposed, if necessary. This optimization plug-in provides a number of additional windows to create the topology data from within FEMAP. One should follow the following steps to perform topology optimization:

1. Create Analysis data. This could be achieved using FEMAP software, or any other software that can create Nastran or GENESIS data deck.
2. Run FEMAP, along with this interface Plug-in. Use this interface to create topology optimization data.
3. Run GENESIS to solve the topology problem.
4. Use FEMAP to view topology optimization results.

Figure 3 shows main windows required to create the topology data.

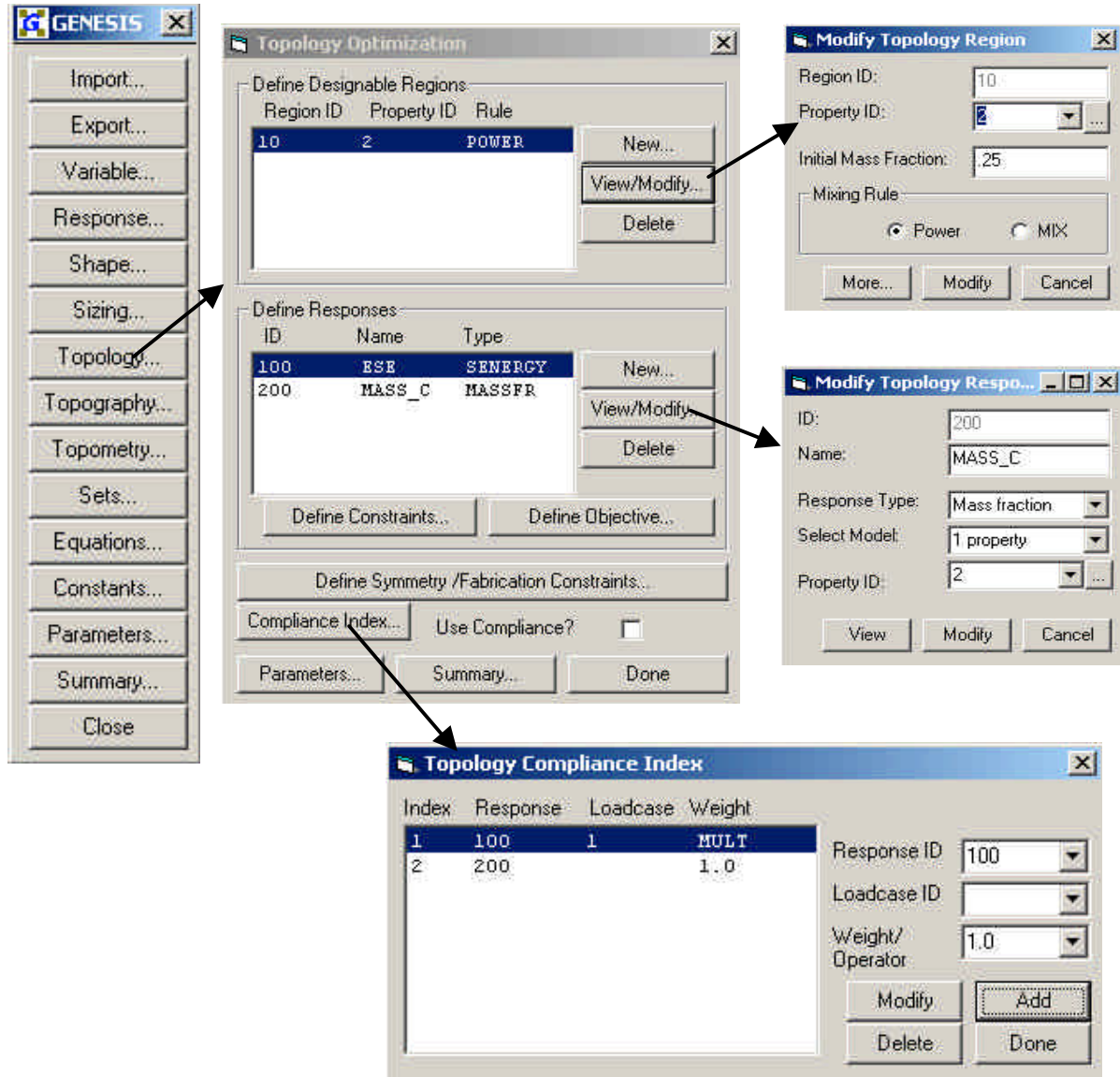
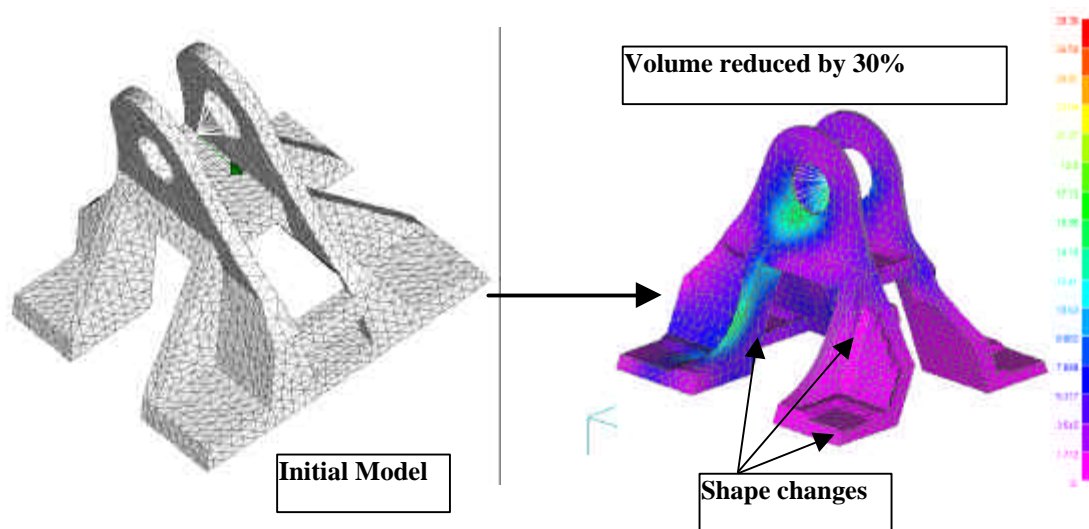


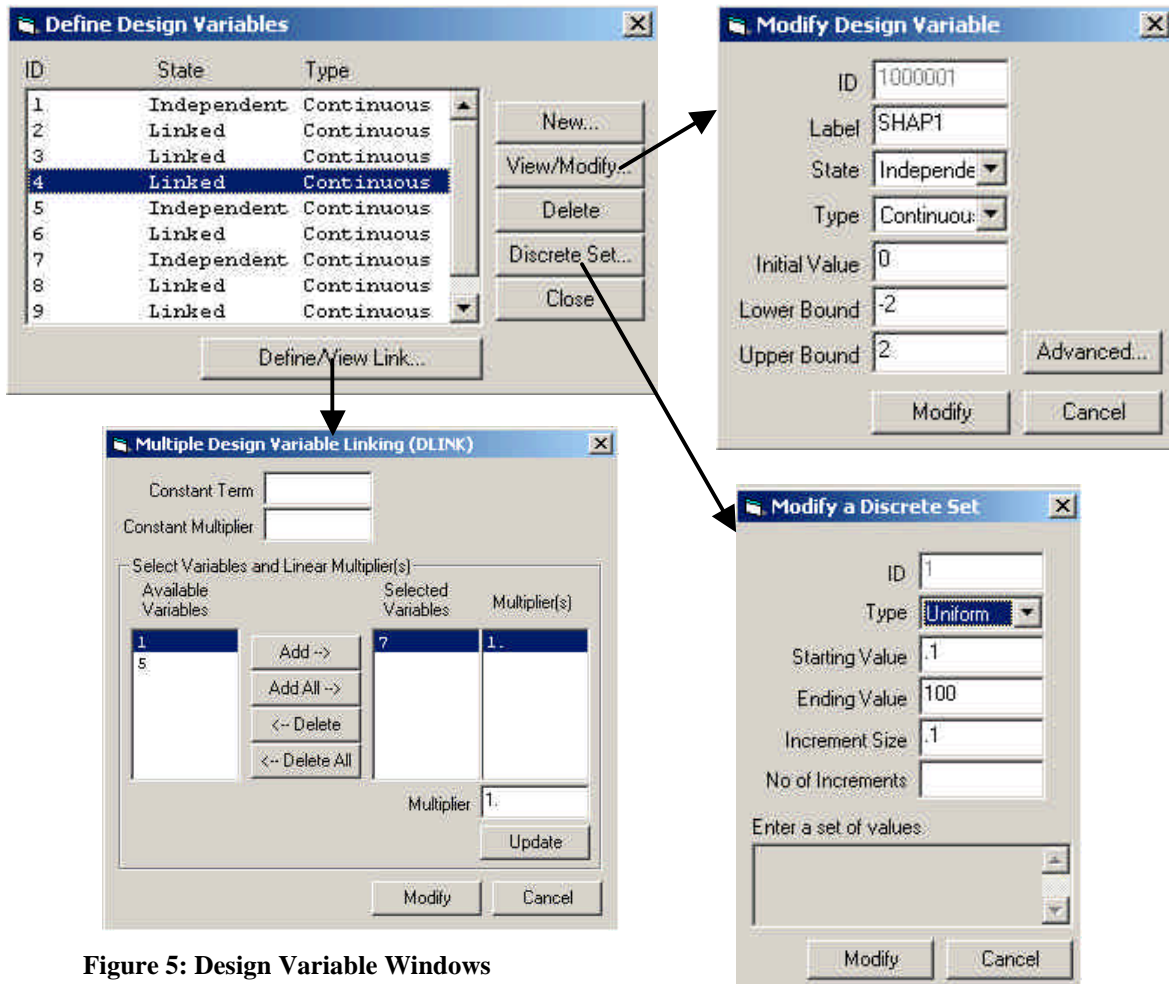
Figure 3: Topology Optimization Windows

**B. Shape and Sizing Optimization** - The cross-sectional dimensions, section properties and grid point locations typically define the analysis model. When cross sectional dimensions such as height, thickness, etc., and section properties such as area, bending moment of inertia, etc are changed (used as design variables) to maximize/minimize one or more responses, it is referred to as sizing optimization. When grid point locations are changed to perform optimization, then the approach is called shape optimization. GENESIS supports both shape and sizing optimization simultaneously. Shape and/or sizing optimization typically involves defining the following optimization data. The Figure 4 shows a hinge used in an aircraft that was optimized using shape optimization.



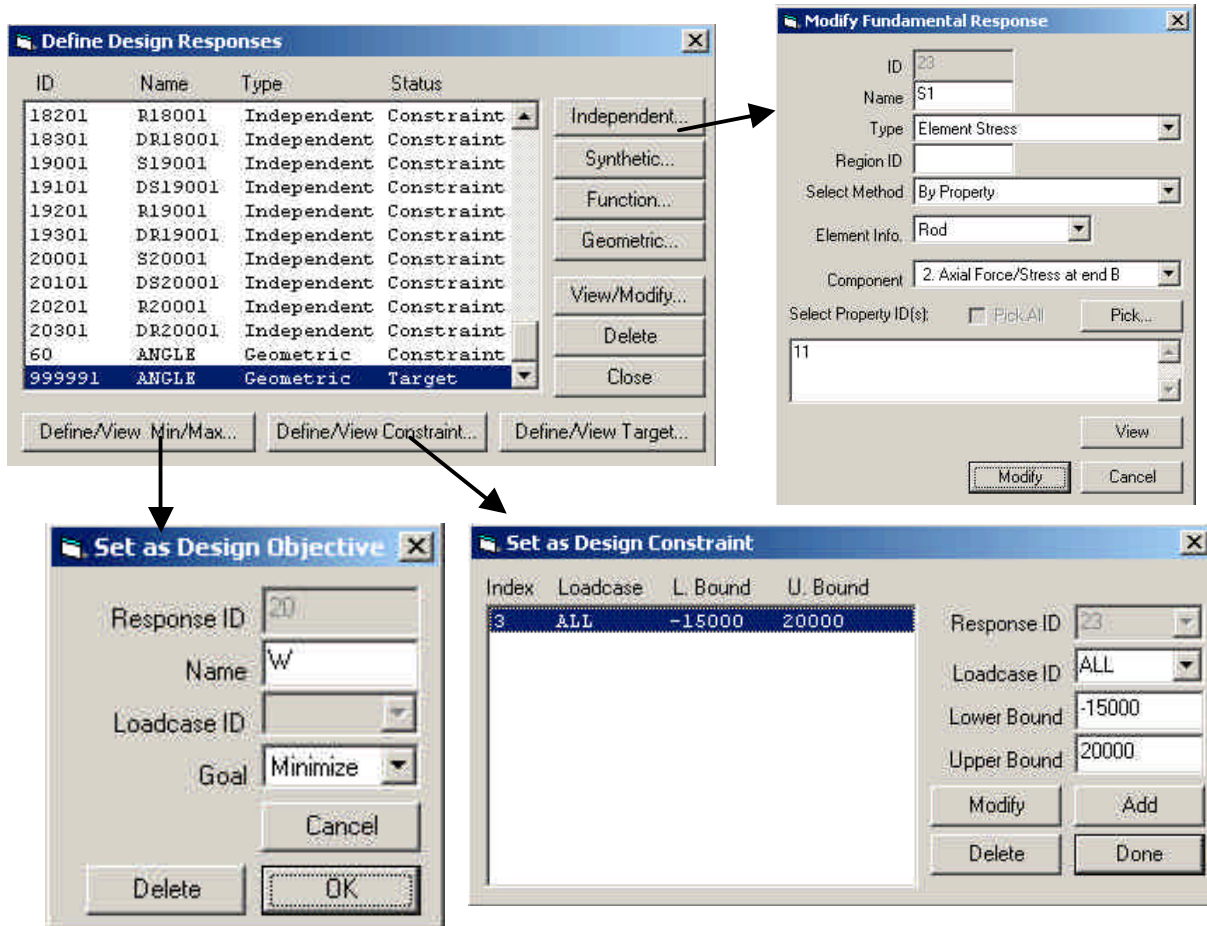
**Figure 4: Shape Optimization Example**

**Design Variable** - Continuous, discrete and integer variables are supported. One or more design variable could be linearly linked with other independent variables. The discrete variables are included, the discrete variable set may consist of uniformly distributed values, and/or nonuniformly distributed discrete variable values. The Figure 5 shows windows available for defining the design variables.



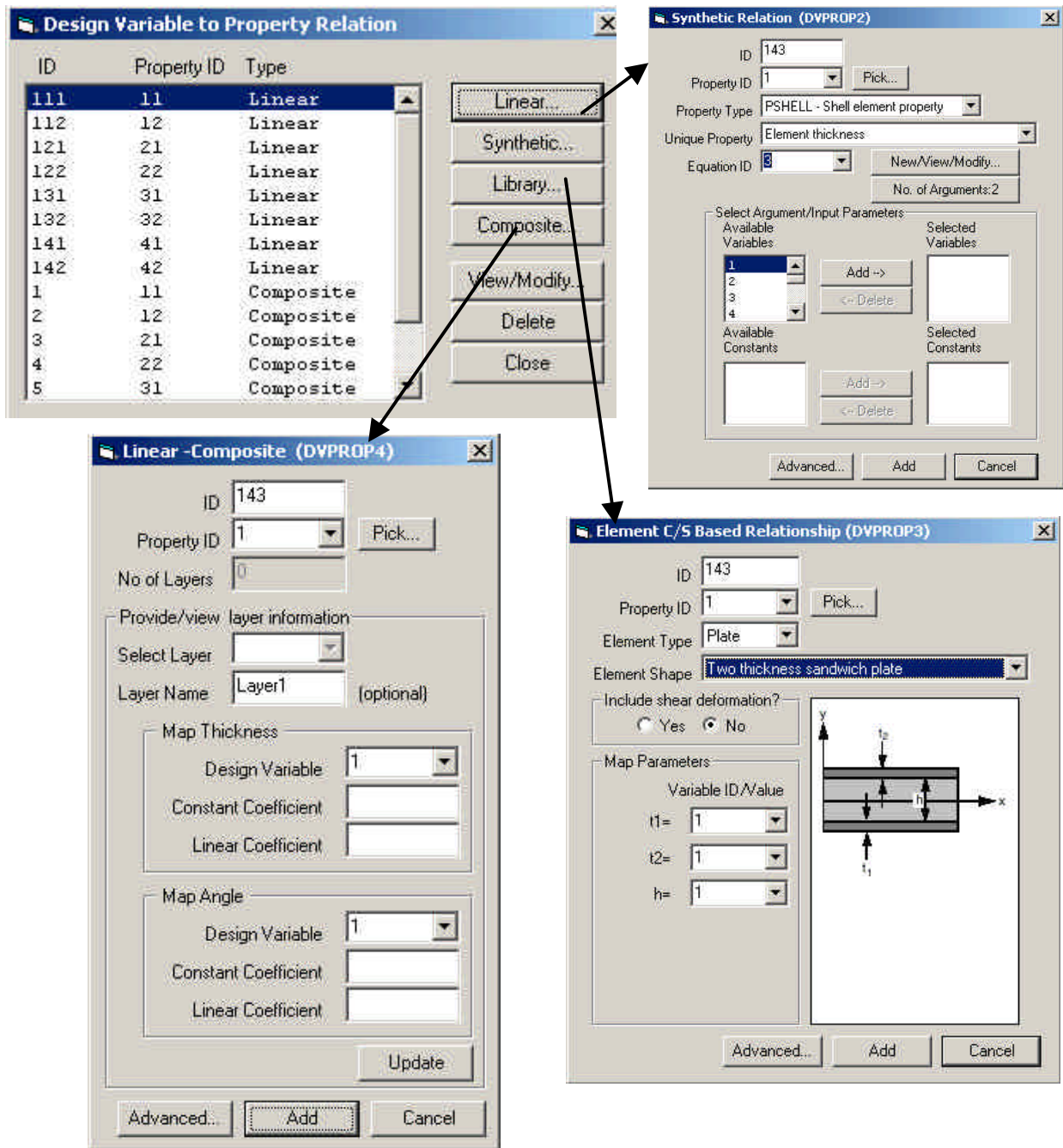
**Figure 5: Design Variable Windows**

**Design Responses** – GENESIS supports a number of design response types. The most frequently used one is the “Independent Response” (DRESP1) type. Volume, mass, grid displacements, element stresses/strains, frequency, mode shapes, etc. are examples of independent responses. Dependent responses such as those calculated based on using equations (DRESP2) and functions (DRESP3) are also supported. In addition to these responses, GENESIS also supports geometric responses, such as length, area, volume etc. Geometric responses are generally calculated using a number of grid locations. Any of these responses could be used as objective and/or constraints. Typically one of the responses will be used as objective function, and others as constraints, if necessary. The Figure 6 shows the Response windows.



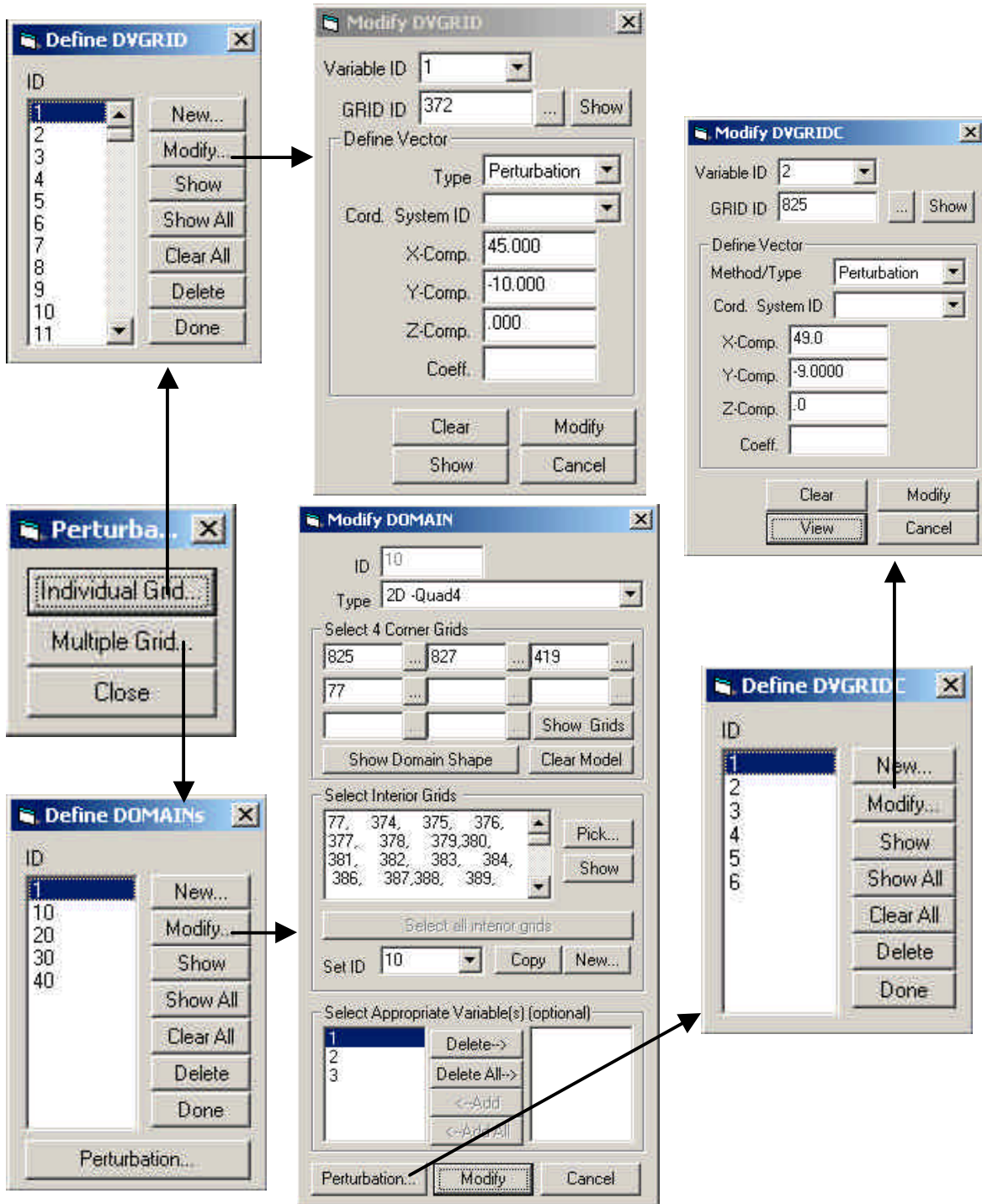
**Figure 6: Design Response Windows**

**Sizing Information** – It involves creating association between the design variables and cross-sectional properties of members. Four types of relationships can be defined. The linear relationship (DVPROP1) involves creating a linear relation between a design variable and a cross-sectional property such as shell thickness, bar/rod cross-sectional area etc. The synthetic relationship (DVPROP2) is similar to linear relationship but the relationship is expressed using an equation, which can provide both linear and non-linear relationships. The library relationship (DVPROP3) allows you to create/map one or more design variables to a set of attributes of cross-sections available in the cross-section library. A number of cross-sections such as square, rectangle, circle, tube, spar, plate thickness etc. are available in the library. The composite relationship (DVPROP4) allows us to linearly map one or more design variables to composite layer/ply angles and thicknesses. The Figure 7 shows some windows commonly used to create the sizing information.



**Figure 7: Sizing Information Windows**

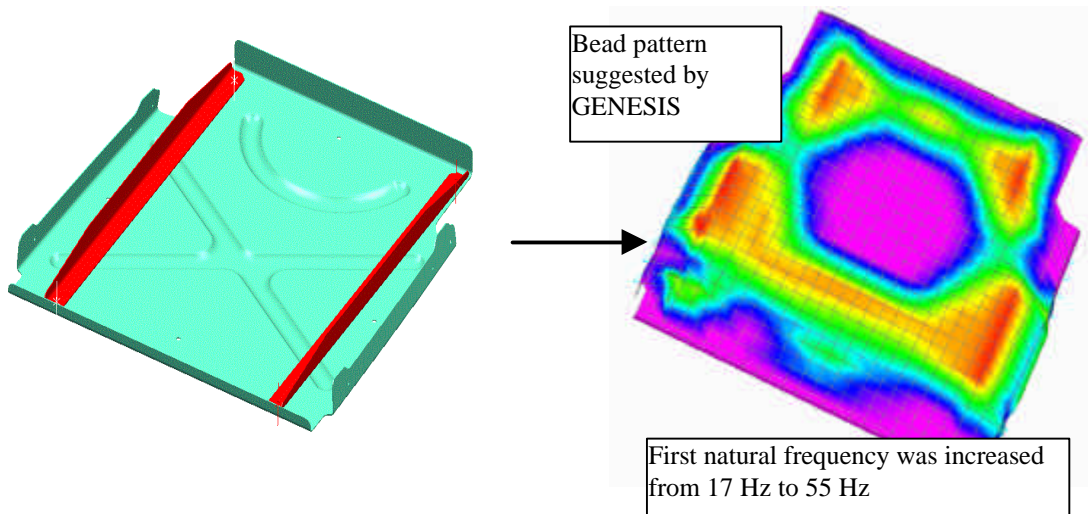
**Shape Information** - Shape optimization consists of finding alternate locations of grids. The information on how to change the grid locations can be provided in two different manners. One way is to provide information to move individual grids (DVGRID); and another way is to provide information for moving a number of grids together (DVGRIDC and DOMAIN). For individual grids, one needs to specify a perturbation vector on the grids to tell the optimizer the direction in which the grid should move when a design variable value is changed. This approach is very complex if a huge number of grids are involved. The alternate approach is to create one or more DOMAIN, which puts together a number of grids, and then provide some controlling perturbation vectors (DVGRIDC) to control the movement of all grids in the DOMAIN. The Figure 8 shows the windows related to shape optimization.



**Figure 8: Shape Information Windows**

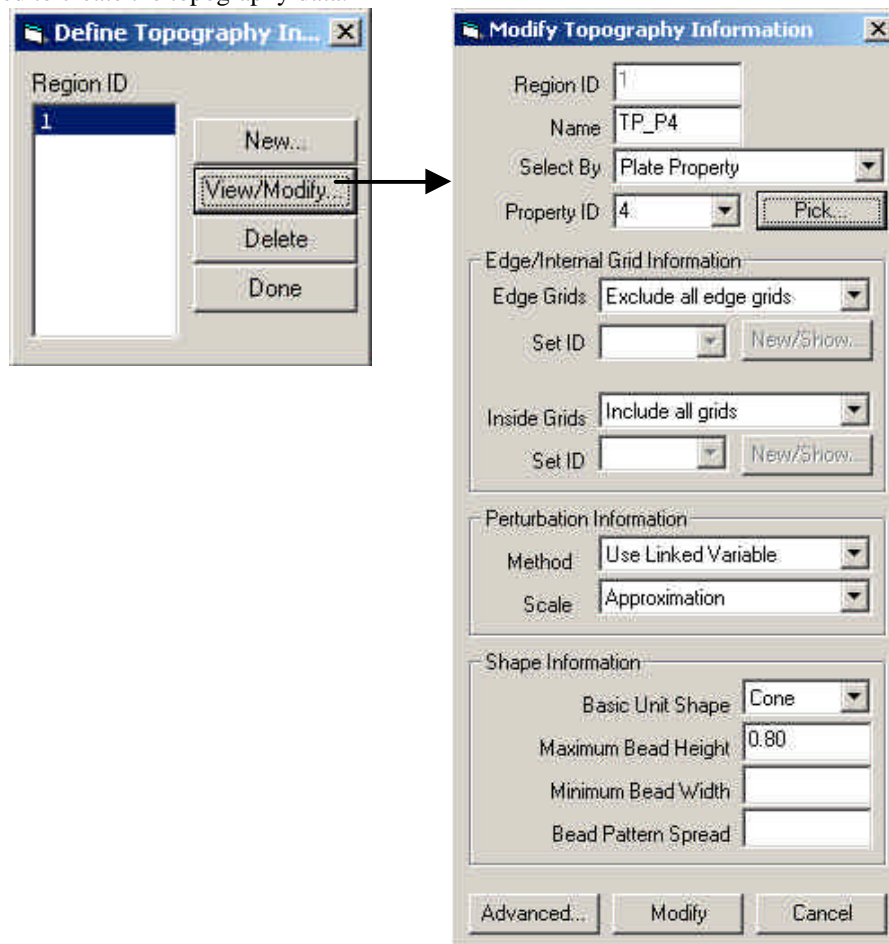
C. **Topography Optimization**– It is a special type of shape optimization wherein the design variables and perturbation vectors are automatically created. In situations, such as stamping, there may be a need to push/pull a number of grids in a region in a particular fashion, topology optimization information (DTGRID) allows us to automate it without creating individual perturbation vectors. The Figure 9 below shows an example where bead patterns were generated using topography optimization.





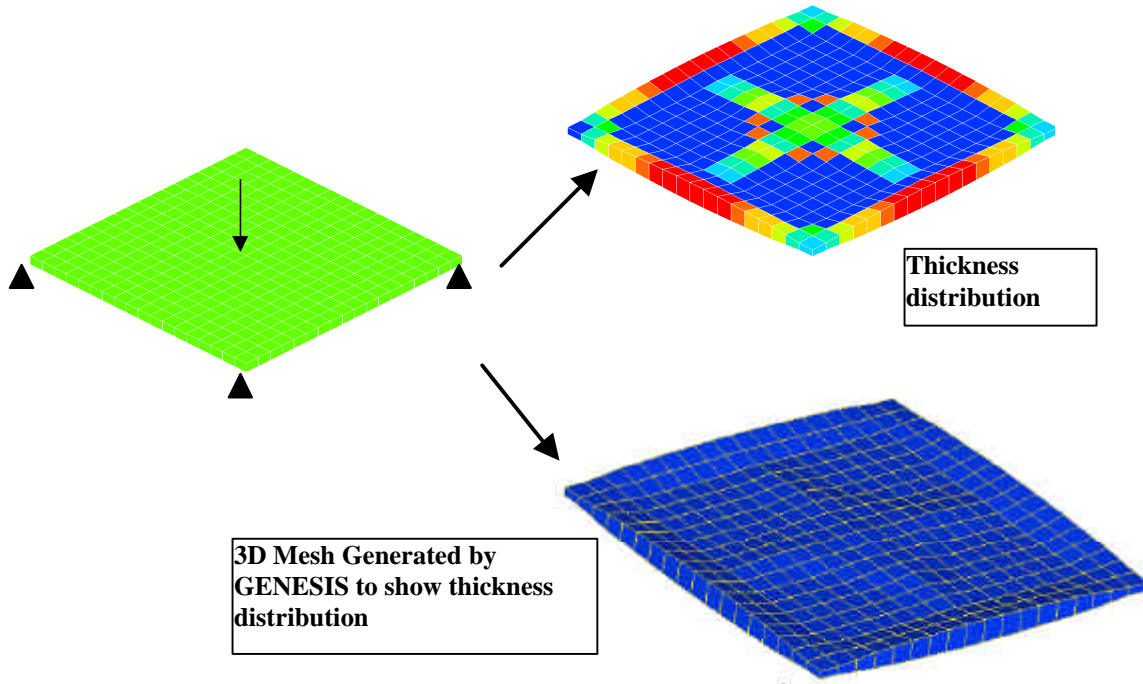
**Figure 9: A Topography Optimization Example**

You will typically specify a region, and the desired shape of the optimized region. The Figure 10 shows the related windows used to create the topography data.

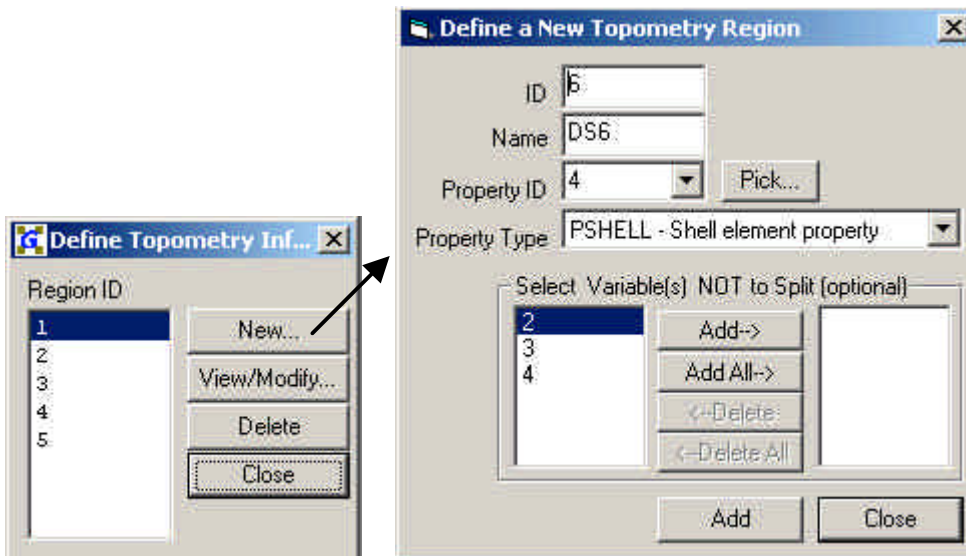


**Figure 10: Topography Information Windows**

**D. Topometry Optimization [5]** – It is a type of sizing optimization, which allows the element-by-element sizing optimization. It can find the optimal distribution of any property, such as thickness in shell elements, that can be size optimized. The Figure 11 shows an example wherein topometry feature was used to design the individual thicknesses of each element.



The Figure 12 shows the related windows to create topometry data.



**Figure 12: Topometry Information Windows**

The interface also allows the creation of number of other related information used in various optimizations. These are briefly described as follows.

**Sets** - It allows you to create sets of entities such as grids, elements, properties and materials. The sets can be used in defining other optimization data.

**Equation** - It allows you to create FORTRAN type equations used in optimization data such as in defining synthetic (DRESP2) response, and synthetic design variable to property relations (DVPROP2).

**Constants** - A number of user-defined constants (DTABLE) can also be defined. These constants can be used in defining synthetic/function responses (DRESP2/3), and in synthetic design variable to property relations (DVPROP2).

**Parameters** – User can also override some default parameters such as default optimizer, optimization algorithms, convergence criteria, etc.

**Summary** – This provides a summary of the optimization information that has been already defined. It is useful for debugging purposes.

## VII. Conclusion

This paper has described the development of an optimization plug-in for Genesis for use with Femap software. It allows the Femap users to create not only the analysis data but also the design optimization data for use with GENESIS structural optimization and analysis code. Femap software along with GENESIS solver now provide an integrated environment that allows engineers to create geometry and/or import it from another CAD modeler, analyze and optimize the model, and view results. VR&D and its customers have solved a number of example problems using this plug-in, and some of them have been briefly mentioned in this paper. More details of these example problems will be described at the presentation.

## VIII. References

1. GENESIS 7.5 Reference Manual (Vol. II), Vanderplaats Research & Development, Inc, March 2004.
2. Femap User Guide, Version 8.3, UGS PLM Solutions, March 2004.
3. Optimization Plug-In for GENESIS for Use with FEMAP – Users Manual, Vanderplaats Research & Development, Inc., June 2004.
4. Optimization Plug-In for GENESIS for Use with FEMAP – Examples Manual, Vanderplaats Research & Development, Inc., June 2004.
5. Leiva, J.P. "Topometry Optimization: A New Capability to Perform Element by Element Sizing Optimization", 10th AIAA/ISSMO Symposium on Multidisciplinary, Analysis and Optimization, Albany, NY, August 30 - September 1, 2004.