



GENESIS

Structural Analysis and Optimization

New Features and Enhancements

Version 11.0

November 2009

-
- **Introduction**
 - **Execution Enhancements**
 - **Analysis Enhancements**
 - **Shape, Sizing, Topometry and Topography Optimization Enhancements**
 - **Topology Optimization Enhancements**
 - **Output Enhancements**
 - **New Input Data**
 - **Enhanced Data**
 - **New and Enhanced Data Relationships**
 - **New Example Problems**
 - **Changes in Version 11.0 with Respect to Version 10.1**

1 Introduction

This document describes the new and enhanced features added to Genesis in version 11.0. Key enhancements include the following:

Freeform optimization. This new type of shape optimization capability adds great variability to the design without requiring a lot of user input. The capability allows the user to easily impose fabrication constraints, and control coarseness and variability.

New mesh smoothing algorithms shape optimization. The mesh smoothing algorithms have been completely redesigned to be more robust.

Anti-distortion constraints for shape optimization. A new procedure has been put in place that will prevent shape optimization from producing a distorted mesh. When combined with the new freeform optimization and the new mesh smoothing, this procedure makes it very easy run robust shape optimization of solid element components.

Visualization support for sizing/topometry optimization. The new SIZING command prints all results of sizing and/or topometry optimization in the OPOST post-processing file. In the past, only the thickness distribution of CQUAD4/CTRIA3 elements was available. Now, spring stiffnesses, rod areas, bar cross section dimensions, etc. can be visualized.

Weld element. A new connector element (CWELD) can connect shell properties even when the grids are not aligned. Also, by virtue of referencing MAT1 data, this element can also be designed with topology optimization, allowing, for example, determination of the optimum locations for welds.

New mass matrix types. Lumped mass and coupled mass matrix formulations have been added to the 10-node CTETRA, 9-21 node CHEXA and the CTRIAX6 elements. Tests have shown that the new default coupled mass matrix produces more accurate eigenvalues.

Relative constraint bounds. New data entries (DCONS2 and TCONS2) use bound values that are scale factors of initial response values.

New user supplied equations for topology. A new data entry (TRESP2) allows creation of synthetic responses for topology by writing equations.

New user supplied subroutines for topology. A new data entry (TRESP3) allows creation of synthetic responses for topology by using built-in functions or external subroutines loaded from a dynamic link library.

New design variable definition for topology. A new data entry (TVAR) adds extra design variables to the topology problem.

Genesis version 11.0 should run any input that was successfully running in version 10.1 with no changes.

2 Execution Enhancements

1. New log file. The `genesis` script on Unix/Linux servers and the `genesis110.exe` program on Windows have been enhanced to create a *datafile.log* file. This log file records system environment details as well as all of the output printed to the terminal console during the run.
2. New command line options. The `genesis` script now accepts several new command line options to set many executive control commands. Additionally, for Windows users, the `genesis110.exe` program has been enhanced to accept the same command line options as the `genesis` script. New command line options include:
 - `-old[=n]` save n old output versions (default is 1)
 - `-diag=d1,d2,dn` add executive control DIAG=d1,d2,dn
 - `-iobuff=n,mK` add executive control IOBUFF=n,mK
 - `-restart=n,m` add executive control RESTART=n,m
 - `-dirall=path` add executive control DIRALL=path
 - `-check` add executive control CHECK
 - `-analysis` add executive control ANALYSIS
 - `-sensitivity` add executive control SENSITIVITY

3 Analysis Enhancements

1. Improved mass matrices. New mass matrix formulations available for the second order elements (9-21 node CHEXA, 10 node CTETRA and CTRIAX6). A lumped mass formulation has been added for these elements. Previously, only the consistent formulation was available. Additionally, a coupled formulation (constructed from a linear combination of the lumped mass matrix and the consistent mass matrix) is now available. Tests have shown that using the coupled mass matrix gives more accurate eigenvalues for coarse meshes. The YES value of the analysis parameter COUPMASS (the default) has been changed to select the coupled mass matrix. A new value, FULL, has been added to select the full consistent mass matrix.

Bulk Data Statement - PARAM,COUPMASS

2. New weld element. This new element (CWELD,PWELD) is intended to simulate a spot-weld connection of shell parts and can use one of three different methods: point-to-point, point-to-patch and patch-to-patch. The patch-to-patch methods are particularly useful for connecting components with different mesh densities, as the element will automatically find the appropriate grids to connect.

3. Improved RBE3 robustness. Degrees of freedom that are set as independent on RBE3 elements, but are not connected to any other elements can lead to numerical problems in the solver and spurious solutions. Such degrees of freedom can now optionally be automatically constrained with SPC. The new analysis parameter RBE3SPC controls this function.

Bulk Data Statement - PARAM,RBE3SPC

4. New user viscous damping matrix entry. Terms can be added to the viscous damping matrix using DMIG bulk data together with the new B2GG solution control command.

Solution Control Statement - B2GG = matrix name

5. New user structural damping matrix entry. Terms can be added to the structural damping matrix using DMIG bulk data together with the new K42GG solution control command.

Solution Control Statement - K42GG = matrix name

6. Updated SMS version. The latest version of SMS includes tuning that further improves performance when large numbers of modes are calculated.

Bulk Data Statement - EIGR, SMS option

7. Updated sparse matrix ordering methods. The sparse matrix ordering methods have been updated to fix performance issues that may have occurred when using the automatic support inertia relief option. Now there should be little difference in solver time between an ordinary static analysis and an automatic support inertia relief analysis.

Solution Control Statement - SUPORT = AUTO

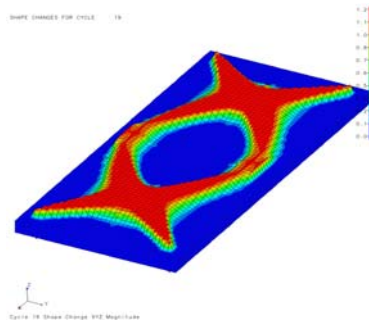
4 Shape, Sizing, Topometry and Topography Optimization Enhancements

1. Freeform Optimization. This type of optimization is a special type of shape optimization that splits an existing perturbation to obtain more design freedom. This capability has three important features: fabrications controls, coarseness controls and variability controls.

Bulk Data Statement - DSHAPE

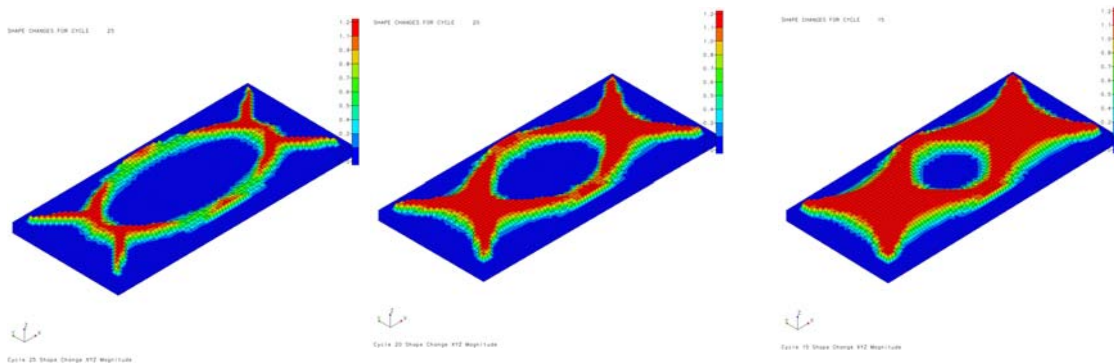
Example 1:

The following example shows freeform optimization in a solid block assembled with hexa elements. The structure is subjected to torsional loads. The result demonstrates the ability of freeform to find an optimal rib pattern.



Example 2:

The following example shows three answers to demonstrate one of the features of freeform: grid fraction controls that affect the variability of the final design.



2. Distortion Constraints. This new feature automatically creates distortion constraints that can dramatically reduce distortion that can occur in shape, topography and freeform optimization.

Bulk Data Statement - DOPT,SHAPECN

3. **Relative Constraint Bounds.** This new feature automatically generates lower and upper bounds that are scale factors of the initial response value. This new capability can be used with DRESP1, DRESP2 and/or DRESP3.
Bulk Data Statement - DCONS2
4. **New Mesh Smoothing Algorithms.** Completely redesigned mesh smoothing procedures are now extremely robust.
Bulk Data Statement - PARAM,MSMOOTH
5. **Visualization Support for Sizing/Topometry.** This new feature allows to post-process sizing and topometry optimization results as well as visualize any existing property dimension and/or property value such areas in rods, masses in mass element and stiffness in elastic or bush elements.
Solution Control Command- SIZING
6. **Faster Shape Optimization Preprocessing.** Shape optimization of second-order elements often requires the midside nodes to move as slaves to the corner nodes. While Genesis provided this capability previously, the calculation of the midside perturbations is now much faster.

5 Topology Optimization Enhancements

1. Synthetic Responses. A new data entry allows creation of synthetic responses for topology by writing equations that can combine fundamental responses (TRESP1) with tabled constants and extra design variables.
Bulk Data Statement - TRESP2 .
2. User-supplied Subroutine Responses. A new data entry allows creation of synthetic responses for topology by writing external subroutines that can combine fundamental responses (TRESP1) with tabled constants and extra design variables.
Bulk Data Statement - TRESP3 .
3. Built-in Equation Responses. A new data entry allows creation of synthetic responses for topology by using built-in equations that can combine fundamental responses (TRESP1) with tabled constants and extra design variables.
Bulk Data Statement - TRESP3 .
4. Extra Design Variables. Design variables can be added to topology for inclusion in synthetic responses.
Bulk Data Statement - TVAR.
5. New Designable Property. The property of the new weld element, PWELD, can be referenced by TPROP. This allows solving problems such as selecting the most important weld elements to keep from a pool of weld elements
Bulk Data Statement - TPROP .
6. Relative Constraint Bounds. This new feature automatically generates lower and upper bounds that are scale factors of the initial response value. This new capability can be used with TRESP1, TRESP2 and/or TRESP3.
Bulk Data Statement - TCONS2

6 Output Enhancements

1. The optimization post processing file (pnameOPOST??.xxx) has been enhanced. Previously it contained the results for THICKNESS =POST. Now it also contains the results for the new command SIZING=POST.

7 New Input Data

7.1 Solution Control

B2GG	Select a DMIG bulk data matrix with terms to be added to the viscous damping matrix.
K42GG	Select a DMIG bulk data matrix with terms to be added to the structural damping matrix.
SIZING	A generalization of the command THICKNESS. Requests output of a post-processing file containing values of all size-designed properties.

7.2 Bulk Data

DSHAPE	Sets properties of perturbation vectors, including the option to use freeform and set freeform parameters.
CWELD	Connectivity information for the weld element.
PWELD	Property and material information for the weld element.
DCONS2	Define constraint bounds that are functions of initial analysis.
TCONS2	Define constraint bounds that are functions of initial analysis.
TRESP2	Selects an existing equation (DEQATN) to create a user response which is a function of standard responses (TRESP1), extra design variables (TVAR), and tabled constants (DTABLE). TRESP2 can be use as: 1) A single objective function (TOBJ can now reference TRESP2) 2) A part of a multiobjective function (TINDEX can now reference TRESP2) 3) A constraint (TCONS can now reference TRESP2)
TRESP3	Selects an existing subroutine (DRESP3) or a built-in equation to create a user response which is a function of standard responses (TRESP1), extra design variables (TVAR), and tabled constants (DTABLE). TRESP3 can be use as: 1) A single objective function (TOBJ can now reference TRESP3) 2) A part of a multiobjective function (TINDEX can now reference TRESP3) 3) A constraint (TCONS can now reference TRESP3)
TVAR	Creates extra design variables for use in TRESP2/TRESP3 responses.

7.3 New Analysis Parameters

CB2	Scale factor for viscous damping terms added with a B2GG/DMIG matrix
CK42	Scale factor for structural damping terms added with a K42GG/DMIG matrix
PRGPST	Controls the printing of the autospc table. If PRGPST = YES, the program will print the autospc table. If PRGPST = NO, the program will not print the autospc table.
RANDOM	Sets the seed for the pseudo-random number generator. Runs with the same seed value will always produce the same sequence of pseudo-random numbers (for a given operating system — different operating systems may produce different sequences for the same seed value).
RBE3SPC	Controls the automatic creation of SPCs for independent degrees of freedoms of RBE3 that are not connected to any other elements.
SEMPC	Sets the MPC id for super-element recovery MPCs generated in a REDUCE run. Recovery MPCs are generated for omitted degrees of freedom of GRIDs that are selected in an eigenvector output request.

7.4 New Design Parameters

OPOST	Controls which property fields are printed to the OPOST sizing post-processing file. OPOST = 0 will only print records for designed property fields. OPOST = 1 will print records for all property fields.
SHAPECN	Distortion Constraint Controls SHAPECN = 0 will cause the program to not use distortion constraints. SHAPECN = 1 will cause the program to impose internal constraints to avoid that the jacobian of any designable element be negative. SHAPECN = 2 is similar to 1, but in addition it will impose constraints so that any distortion measure, as defined in DISTORT analysis entry, does not produce errors. If the user does not define DISTORT values, the program will use the DISTORT default values. SHAPECN = 3 is similar to SHAPECN = 2, but instead of using ERROR limits the program will use WARNING limits.

8 Enhanced Data

8.1 Executive Control

DRESP3 Can now be used to load a shared library for TRESP3 responses.

8.2 Solution Control

TVAR Can now control the printing of design variables created by freeform shape optimization as well as the new user created topology design variables.

DRESP2 Can now control the printing of responses generated by the new TRESP2 and TRESP3 entries.

ESE Element strain energies (ESE) from statics and normal mode analysis associated to superelements (K2UU) and the new CWELD element can now be printed to the output files. ESE can also be printed in PUNCH and OUTPUT2 formats for the CWELD element.

8.3 Bulk Data

DMIG Can now be referenced by the B2GG and K42GG solution control commands.

DOPT The parameter that control number of design cycles has be changed to allow 0. DOPT,0 is equivalent to use the solution control command ANALYSIS

DSELECT Can now be used to reference design variables that are generated in freeform.

DTGRID Can now use a new way to generate initial random values of internal design variables.

DVGRID Can now be split using the new DSHAPE entry.

DVGRIDC Can now be split using the new DSHAPE entry.

DEQATN Can now be used with the new TRESP2 entry.

DTABLE Can now be used with the new TRESP2/TRESP3 entries.

TCONS Can now be used with the new TRESP2/TRESP3 entries.

TINDEX Can now be used with the new TRESP2/TRESP3 entries.

TOBJ Can now be used with the new TRESP2/TRESP3 entries.

TRESP1 Can now be used with the new TRESP2/TRESP3 entries.

TPROP Can now be used to design the new PWELD entry.

8.4 Analysis Parameters

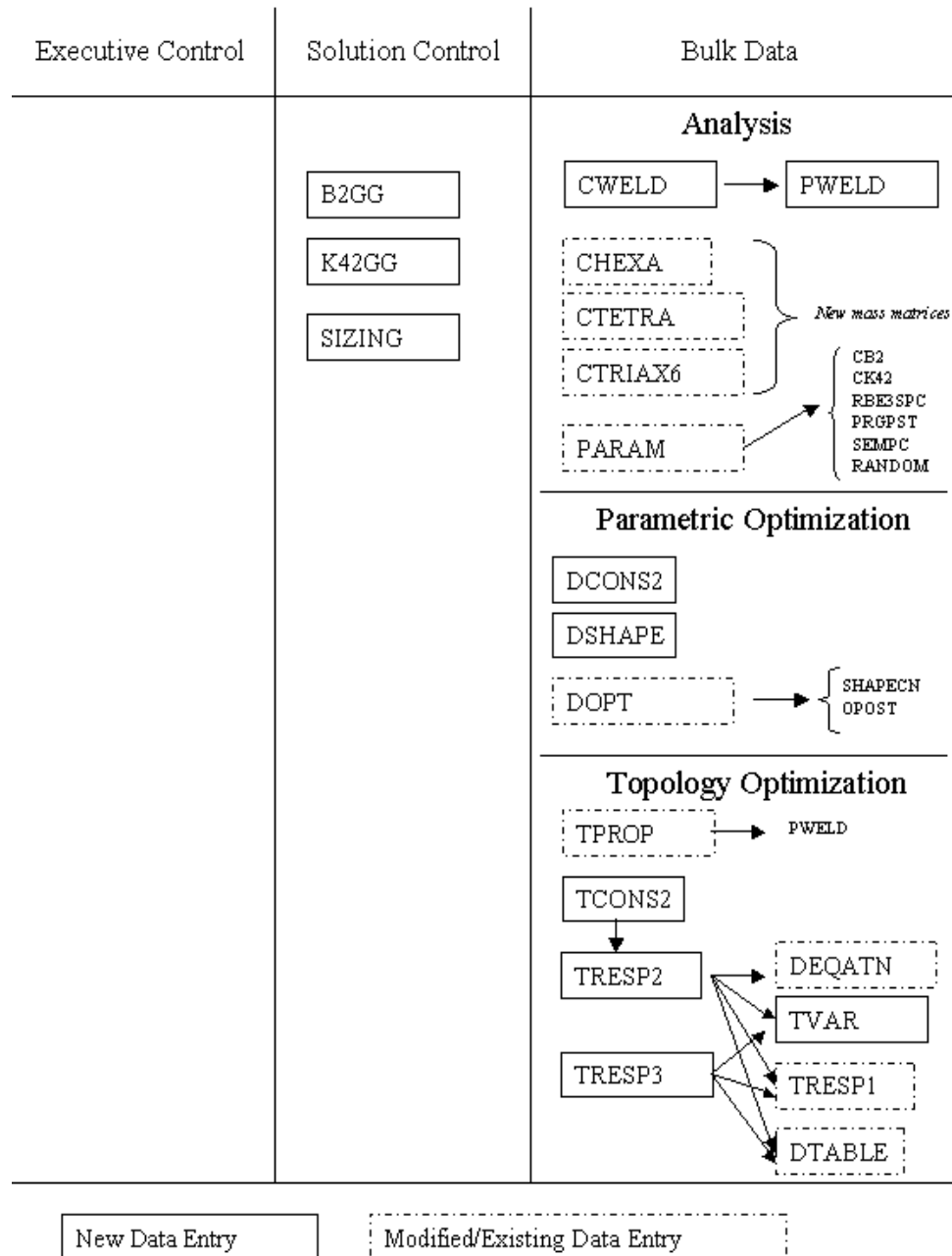
COUPMASS Allows for new option FULL. With this option the user can now use coupled mass matrices. In the past the options were limited to consistent and lumped.

8.5 Design Parameters

DESMAX The parameter that control number of design cycles has been changed to allow 0. DOPT,0 is equivalent to use the solution control command ANALYSIS

9 New and Enhanced Data Relationships

The following figure shows the relationships of the new Genesis data entries.



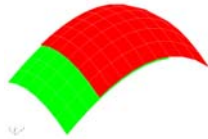
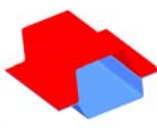
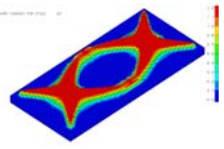
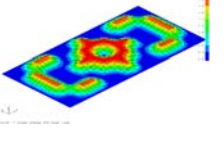
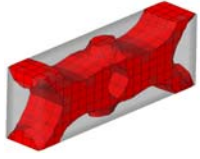
10 Genesis Manual Updates

All Genesis manuals have been updated to reflect the new features, as well as the new and modified data entries.

Manual Title	Filename	Status
GENESIS: Analysis Manual	volume1.pdf	Updated to reflect all improved and new features
GENESIS: Design Manual	volume2.pdf	Updated to reflect all improved and new features
GENESIS: Analysis Examples	volume3.pdf	Updated. Two new examples added
GENESIS: Design Examples	volume4.pdf	Updated. Three new examples added
GENESIS: Quick Reference Manual	quickref.pdf	Updated to reflect all changes and new data entries
GENESIS: New Features and Enhancements	newfeat.pdf	This document

11 New Example Problems

The following table describes new examples and their corresponding input file names. The listed files are provided with the installation

Name	Problem	Special Features	Figure
A039.dat	Rigid body modes of doubly curved shell	Use of new weld element. Weld element types used are PARTPAT, ELPAT, ELEMID, GRIDID and ALIGN.	
A040.dat	Rigid body modes of double hat structure	Use of new weld element. Weld element types used is PARTPAT	
D080.dat	Freeform Optimization on Raw Perturbations (DVGRID) using Grid Fraction and Mirror Symmetry Constraint	SYM and GRIDFR options in DSHAPE data entry to enforce mirror symmetry and also constraint the movement of the grids.	
D081.dat	Freeform Optimization on DOMAIN Perturbations (DVGRIDC) using Grid Fraction and Axisymmetry Constraints	SYM and GRIDFR options in DSHAPE data entry to enforce mirror symmetry and also constraint the movement of the grids.	
T023.dat	Topology Optimization Using the Beta Method and Synthetic Responses (TRESP2)	TRESP2 and DEQATN data entry is used to create synthetic responses. Also TVAR data entry is used to create additional design variables.	

The last three new examples are also provided with the new step-by-step sample manual of Design Studio. This new manual contains over one hundred example problems.

12 Changes in Version 11.0 with Respect to Version 10.1

Genesis version 11.0 should run any input that was successfully running in version 10.1 with no changes.

Genesis version 11.0 might now run successfully problems that were not successfully running in version 10.1 due to mesh distortion. This is because v11.0 includes an improved mesh smooth algorithm and new powerful distortion control parameters that, in effect, impose mesh distortion constraints.

Problems that use mesh smoothing might show some numerical differences due to the new mesh smoothing algorithm. DIAG=901 or DIAG=902 will revert to the old (v10.1 and lower) behavior.

The default mass matrix for second-order elements has been changed from the full consistent mass matrix to a coupled mass matrix. Eigenvalue problems with second-order elements may return slightly different results. PARAM,COUPMASS,FULL can be used to get the v10.1 formulations.

A different pseudo-random number generator is used in version 11.0, so topography problems will start with a different random initial perturbation. This may cause the optimizer to converge to a different local optimum and/or with a different number of design cycles.

The format of the reduced structural damping matrix written by the K4AA command has changed. The command now writes a real matrix, suitable for use with the new K42GG command. To revert to the old behavior of writing the imaginary part of a complex matrix, use PARAM,IPRM4,1.