



# ***GENESIS***

## ***2022***

Structural Analysis and Optimization

**New Features and Enhancements**

**Version 21.0**

September 2021

- 
- **Executive Summary**
  - **Analysis Enhancements**
  - **Structural Optimization Enhancements**
  - **Post Processing File Enhancements**
  - **General Enhancements**
  - **New Input Data**
  - **Enhanced Input Data**
  - **GENESIS Manual Updates**
  - **Changes in GENESIS 2022 with Respect to Version 19.0**

---

# 1 Executive Summary

This document describes new and enhanced features included in *GENESIS 2022*.

The following is a summary of some of the key new features and enhancements:

**Fatigue Optimization:** Fatigue damage and life can now be used in optimization. Fatigue damage and life can be calculated at any solid or shell element and can be used as an objective or constraint using the new FATIGUE response. In addition, the new GFATIGUE response is available for grids associated to solid elements. These new responses can be used in any type of optimization.

**Responses for Resonance Frequency:** Existing DRESP1 and TRESP1 response types MDISPR, MVELOR, MACCER, DDISPR, DVELOR and DACCER used to calculate the loading frequency corresponding to the highest peak of the corresponding displacement/velocity/acceleration magnitude within the calculation range have been updated to also calculate: the loading frequency of the first peak, the lowest peak, and the first antipeak.

**Subsystem Inertia Responses:** For system inertia responses (includes system inertia and center of gravity), now the user can define a subsystem by specifying a list of properties and only those properties will be included in that response calculation.

**Mode Shape Response:** Eigenvector component responses (EVECT) are now available for topology optimization.

**Analysis Performance Improvement:** The SMS and sparse matrix solver are now faster due to improved parallelization of certain internal calculations.

**Kinetic Strain Energy Output:** Element kinetic energy results can be output to the punch/output2 format post-processing files for eigenvalue and frequency response loadcases. For frequency response, this result is a real value comprising one of an average, amplitude or peak value of the dynamic function.

**Topology Optimization with Extrusion along a Curved Path:** Extrusion fabrication constraints can now follow along a curved path.

**Topology Optimization with Extrusion/Filling along Surface Normal:** Extrusion or filling fabrication constraints can now grow structures normal to a user-defined surface.

**Topology Optimization with New Frozen Region:** A new frozen region entry (TFROZEN) allows easy definition of areas where material is forced to be kept.. This can be used to make cleaner results near loads/boundary conditions, or can be combined with fabrication constraints for special situations, for example, to create an obstruction blocking a filling constraint. Frozen region can be defined by surfaces or element sets.

**Optimization Performance Improvement:** Some key sections of the approximate problem has been parallelized. Problems with a large number of retained constraints are much faster when multiple processors are used.

---

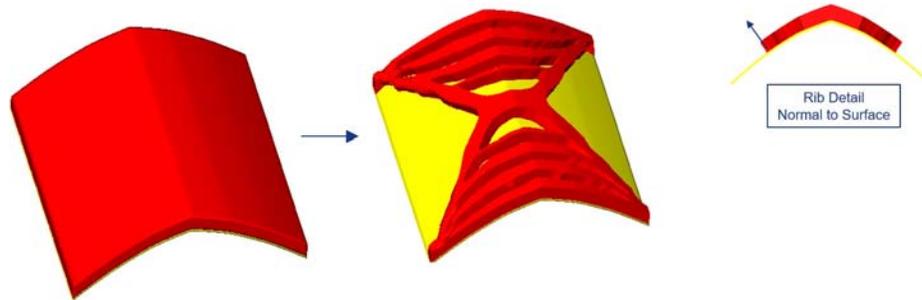
## 2 Analysis Enhancements

1. Disabled loadcases: *GENESIS* now has the ability to disable loadcases. Disabled loadcases will be will not be calculated. Design data referencing disabled loadcases will also be ignored.  
Solution Control Command - ENABLE=NO.
2. Analysis performance improvement: The sparse matrix solver and the SMS eigensolver are now faster due to some improved parallelization.
3. New mass element. CONM1 defines a 6x6 mass matrix at one grid.  
Bulk Data Entry - CONM1
4. New modal effective mass output: This is a new command to control the printing of a modal effective mass output.  
Solution Control Command - MEFFMASS
5. New mesh smoothing controls: Mesh smoothing can now be performed on user-selected regions. In the previous versions, enabling mesh smoothing always affected the entire model.  
Bulk Data Entry - MSMOOTH
6. Parameter to change second order elements to first order: With this new parameter, the all second order elements will be converted to a first order automatically. This allows for faster calculations at the expense of a decrease in accuracy.  
Bulk Data Entry - PARAM,E2TOE1
7. Contact surface separation definition: BCPAIR includes a new option to define the initial surface separation as a fixed specified value. BCPAIR field GPAD now allows a new option, "SETU0", which will assign the value from the U0 field to the initial surface separation. This option makes it easier to convert contact data from some third-party analysis programs.  
Bulk Data Entry - BCPAIR

### 3 Structural Optimization Enhancements

#### 1. Normal-to-surface topology fabrication constraints:

Bulk Data Statements - TSYM1,TSYM2,TSYM3, new option "SURF" to allow exiting TYPEs=FBX, FBY, FBZ, FTX, FTY, FTZ, EX, EY, and EZ to grow members (such as ribs) normal to a user supplied surface

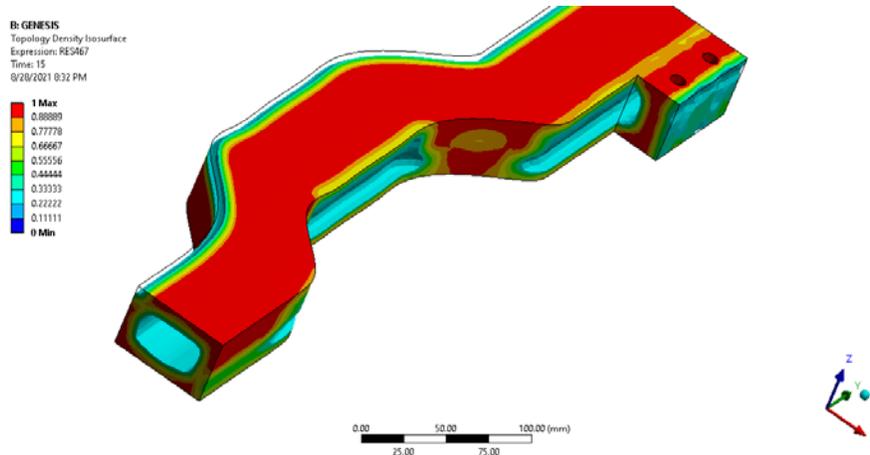


Topology Optimization results with normal to surface constraints

#### 2. Curved extrusion topology fabrication constraints:

Bulk Data Statements - TSYM1,TSYM2,TSYM3, new option "PATH" allows exiting TYPEs=EX, EY, and EZ to grow members using a user-supplied path on a user-supplied curved direction or path.

S



Topology Optimization with Curved Extrusion Constraints on a Curved Structure

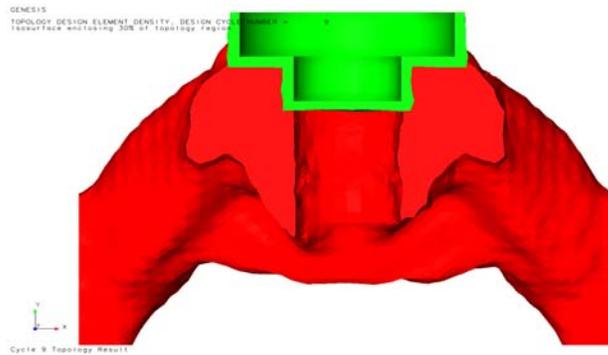
#### 3. Frozen regions for topology optimization:

Bulk Data Statements - TFROZEN.

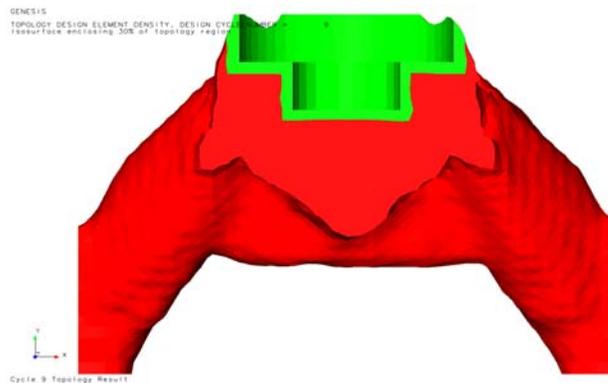
## New Features

With the new frozen region (TFROZEN), the fabrication constraints can be enforced on the design region and frozen region as a whole. Frozen region can be parts or sets of elements of a topology designable region.

The first image below shows a topology result with previous frozen region, which was not a part of the topology design region. The filling constraint is not enforced onto the green non-designable region, and the two bodies are not castable as a whole along Y direction. While the second image below shows the topology results with a new frozen region, which is now part of the topology design region. The fabrication constraints are enforced on both bodies. Thus the entire geometry is castable.



Topology Optimization Without Using a Frozen Region



Topology Optimization Using a Frozen Region

4. New direct/alternative method in topology optimization to interpolate density pole results to elements:

Bulk Data Statements - DOPT,POLEG.

5. Shape optimization improvements: With a new default, now all designable grids associated to midside of second order elements are now moved to their midside location. Non designable grids are not moved. This new option controlled by the PARAM, MIDSIDE, 3 helps to avoid distortions. This new default is now preferred over the previous recommended option (PARAM, MIDSIDE,2). With this option, during optimization, the midside grids are updated so that they are always kept in the middle of its two corresponding corner grids.

Bulk Data Statements - PARAM, MIDSIDE, 3.

6. New responses for fatigue: Fatigue damage and life can now be used in optimization. Fatigue damage and life calculated at any solid or shell element can be used as objective or constraints using the new FATIGUE response. In addition, GFATIGUE is available for grids associated to solid elements. These new responses can be used in all types of optimization.

Bulk Data Statements - DRESP, *new* RTYPE=FATIGUE, GFATIGUE

7. New responses for resonance frequency: Existing DRESP1 and TRESP1 response types MDISPR, MVELOR, MACCER, DDISPR, DVELOR and DACCER used to calculate the loading frequency corresponding to the highest peak of the corresponding displacement/velocity/acceleration magnitude within the calculation range has now been updated to also calculate: the loading frequency of the first peak, the lowest peak, and the first antipeak.

Bulk Data Statements - DRESP1/TRESP1, updated RTYPE=MDISPR, MVELOR, MACCER, DDISPR, DVELOR, DACCER

8. New eigenvector component response for topology: Topology optimization can now use the ETECT type to select eigenvector components. In previous versions these types could only be selected for the other non-topology types of optimization.

Bulk Data Statements - DRESP1, RTYPE=ETECT

9. Inertia responses are now available for subsystems: DRESP1 can now use the INERTIA types to select inertia components or a center of gravity of a list of properties. In previous versions these types could only be selected for the whole system.

Bulk Data Statements - DRESP1, RTYPE=INERTIA

10. Optimize third-party software for stress/plastic strain in a nonlinear analysis: With ESLMATCH, *GENESIS* will match some selected third party software stress or plastic strain result from a nonlinear analysis and use them for optimization. Therefore the user can optimize for the nonlinear stress/strain directly.

Executive Control Command- ESLELEM, ESLGRID.

Solution Control Command - ESLMATCH.

11. User-supplied libraries for responses and cross section properties can now be created using Lua scripts. This avoids having to compile native code and create native object libraries.

Executive Control Command - DRESP3, DLIB

## New Features

12. The html summary file now contain a new topology density fraction histogram.

Solution Control Command - GRAPH

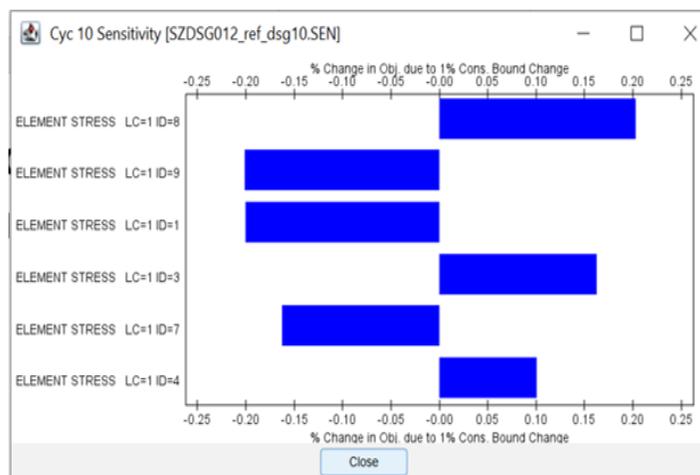
## 4 Post Processing File Enhancements

1. Element kinetic strain energy: Element kinetic energy results can be output to punch/output2 format post-processing files for eigenvalue and frequency response loadcases. In frequency response, the kinetic energy oscillates at twice the loading frequency about a non-zero average value. The amplitude is the maximum extent above or below the average, and the peak value is the average plus the amplitude. The output for each element is a real value comprising one of average, amplitude, or peak value, selected in parenthesis on the EKE command.

Solution Control Command - EKE

2. Optimal Sensitivities: Now, optimal sensitivities are printed together with standard sensitivities in the sensitivity file and Design Studio can plot them as shown in the figure bellow.

Solution Control Command - SENSITIVITY=POST, BOTH



Optimal Sensitivity Plot

---

## 5 General Enhancements

1. Sensitivity calculation for eigenvector components are now much faster. The adjoint method has been adapted to deal with these types of responses. This improvement has allowed *GENESIS* to include eigenvector components in topology problems.

---

## 6 New Input Data

### 6.1 Executive Control

ESLELEM	Defines the name of an ESL reader that will read element stress/strain to be used with the new solution control command ESLMATCH
ESLGRID	Defines the name of an ESL reader that will read grid stress to be used with the new solution control command ESLMATCH

### 6.2 Solution Control

ENABLE	Control enabling or disabling loadcase and/or loadcoms
EKE	Control the printing of element kinetic energy. This request is for modal or direct frequency response loadcases and also for natural frequency loadcases
ESLMATCH	Defines an equivalent static stress/strain matching based on data returned from an external ESL reader
MEFFMASS	Control the printing of a modal effective mass output

### 6.3 Bulk Data

MSMOOTH	Defines regions where mesh smoothing is to be performed
CONM1	Defines a 6x6 symmetric mass matrix at a grid point of the structural model
TFROZEN	Defines frozen regions for topology optimization. This enforces density values or ranges on a set of designable elements.

### 6.4 Analysis PARAMeters

E2TOE1	Parameter to convert second order elements to first order elements
--------	--

### 6.5 DOPT Parameters used in Topology Optimization

POLEG	A value of 0 will cause <i>GENESIS</i> to interpolate the density values to the elements directly; this is a newer method. A value of 1 will cause <i>GENESIS</i> to interpolate the density values to grids and from grids to elements. This parameter is only used for geometry based topology. The default is 1.
-------	---

---

## 7 Enhanced Input Data

### 7.1 Executive Control

DLIB	User cross sections used with DVPROP3 can now be written using a Lua script
DRESP3	User subroutine sections used with DRESP3 can now be written using a Lua script
ESLCONF	Can now be used with ELSELEM and ELSGRID. Earlier it could only be used with ESLDISP
ESLDISP	Displacement readers can now be written using a Lua script
GNMASS	User mass matrices can now be written using a Lua script

### 7.2 Solution Control

LOADCASE	Can be disabled by the new ENABLE=NO command. Static loadcases can now reference new solution control commands ESLMATCH. Natural frequency or dynamic loadcases can now reference EKE
----------	---

### 7.3 Bulk Data

BCPAIR	Field GPAD can now use the "SETU0" option. Field 9 is now U0.
TSYM1	New normal to a surface filling constraints. Compatible with: EX, EY, EZ, FBX, FBY, FBZ, FTX, FTY and FTZ. New curved extrusion constraints: Compatible with: EX, EY an EZ.
TSYM2	New normal to a surface filling constraints. Compatible with: EX, EY, EZ, FBX, FBY, FBZ, FTX, FTY and FTZ. New curved extrusion constraints: Compatible with: EX, EY an EZ.
TSYM3	New normal to a surface filling constraints. Compatible with: EX, EY, EZ, FBX, FBY, FBZ, FTX, FTY and FTZ. New curved extrusion constraints: Compatible with: EX, EY an EZ,.

### 7.4 DRESP1/TRESP1- New enhanced response types RTYPE

DACCER	Frequency corresponding to the first resonance or the first antipeak or the lowest antipeak, for acceleration from direct frequency response, can now be selected with this RTYPE. Peak frequencies can still be used with this RTYPE.
DDISPR	Frequency corresponding to the first resonance or the first antipeak or the lowest antipeak, for displacement from direct frequency response can now be selected with this RTYPE. Peak frequencies can still be used with this RTYPE.

DVELOR	Frequency corresponding to the first resonance or the first antipeak or the lowest antipeak, for velocity from direct frequency response can now be selected with this RTYPE. Peak frequencies can still be used with this RTYPE.
MACCER	Frequency corresponding to the first resonance or the first antipeak or the lowest antipeak, for acceleration from modal frequency response can now be selected with this RTYPE. Peak frequencies can still be used with this RTYPE.
MDISPR	Frequency corresponding to the first resonance or the first antipeak or the lowest antipeak, for displacement from modal frequency response can now be selected with this RTYPE. Peak frequencies can still be used with this RTYPE.
MVELOR	Frequency corresponding to the first resonance or the first antipeak or the lowest antipeak, for velocity from modal frequency response can now be selected with this RTYPE. Peak frequencies can still be used with this RTYPE.
EVECT	Eigenvector components can now be used for topology optimization. Eigenvector components calculations are now faster for all types of optimization.
INERTIA	Inertia responses can now be requested by list of property IDs.

## 7.5 Analysis PARAMeters- Enhanced parameters

MIDSIDE	The Parameter MIDSIDE commonly used for shape optimization has a new optional value 3. With this value, each midside grid of each designable second order elements is moved to its respectively physical midpoint. Also, midside perturbations are automatically created or modified to force the midside grids to remain at their midside during the optimization process.
---------	---

---

## 8 *GENESIS* Manual Updates

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

<b>Manual Title</b>	<b>Filename</b>	<b>Status</b>
<i>GENESIS</i> : Analysis Manual	volume1.pdf	Updated to reflect all enhanced and new features.
<i>GENESIS</i> : Design Manual	volume2.pdf	Updated to reflect all enhanced and new features.
<i>GENESIS</i> : Analysis Examples	volume3.pdf	Updated.
<i>GENESIS</i> : Design Examples	volume4.pdf	Updated.
<i>GENESIS</i> : Quick Reference Manual	quickref.pdf	Updated to reflect all changes and new data entries.
<i>GENESIS</i> : New Features and Enhancements	newfeat.pdf	This document which describes the changes between <i>GENESIS</i> versions 19.0 and 2022.

---

## 9 Changes in *GENESIS* 2022 with Respect to Version 19.0

*GENESIS* 2022 should run any problem that was successfully running in version 19.0 without any changes.