
# *GENESIS* Structural Optimization for ANSYS® Mechanical

An Integrated Extension that adds Structural Optimization to ANSYS® Environment

## New Features and Enhancements

***Release 2019.06***

***GENESIS* VERSION 18.0**

June 2019

## COPYRIGHT NOTICE



## WARNING



## DISCLAIMER



## TRADEMARKS MENTIONED IN THIS MANUAL

# Contents

# 1 Introduction

*GENESIS* Structural Optimization for ANSYS® Mechanical (GSAM) is an integrated extension that adds structural optimization to the ANSYS environment. The extension provides an easy-to-use interface which allows the user to setup Structural optimization problems, post-process them and export the optimization results within the ANSYS environment.

GSAM is a super set of GTAM (*GENESIS* Topology Optimization for ANSYS® Mechanical). GSAM can perform any function that GTAM does. The extra functionality is to perform topography, freeform, sizing and topometry design.

This document covers the new features and enhancements added to GSAM/GTAM version 18.0, which matches the version of *GENESIS* engine.

# 2 Executive Summary

The key new features and enhancements include the following:

**Optimization of Homogenized Lattice Structures**: Solid elements in GENESIS can now be used to simulate lattice structures. GENESIS can use homogenized materials that represent either some new built-in lattice patterns or user supplied lattice patterns. Sizing and/or topometry can be applied to the homogenized lattice properties on solid elements. The program can optimize for the diameter/thickness of the lattice structures.

**New Fabrication Constraints for Topology with Additive Manufacturing Constraints (Overhang Angle Constraints):** Overhang Angle constraints can now be defined on topology region: ABX, ABY, ABZ, ATX, ATY, and ATZ. This helps eliminating or reducing the need of non-structural support when parts are design for additive manufacturing (3D printing). An overhang critical angle can be specified.

**Maximum Member Size Results Improvement:** Existing methods has been tune up to produce better answer for topology optimization with maximum member size.

**Support Bar/Beam in Topology Optimization:** The elements in a wire body (bar/beam elements) can be designed with topology optimization.

**Support Bar/Beam in Sizing/Topometry Optimization:** The cross-sectional dimensions of the bar/beam can be designed with sizing/topometry optimization.

**Support Parameterization in Optimization Settings and Design Data Definitions:** Many parameters in analysis and optimization settings or in the design data definitions can be parameterized using standard ANSYS parameterization technique. And these parameters can be used in parametric study, DOE or optimization.

**Enhancement in Random Analysis Setup:** An extra modal analysis will be created automatically if performing random analysis with enforced boundary conditions. The large mass method in GENESIS for calculating enforced boundary conditions will be applied on this extra modal loadcase, not on the original modal analysis. Therefore, the user can still use the regular mode number in optimizing the original modal analysis.

**Enhancement in ESL Analysis Setup:** A new option is added in optimization with ESL analysis. The user can choose to add soft springs for all geometries in the model. This helps the case when there are unsupported APDL commands in boundary conditions definition in the ANSYS analysis.

**Tolerance for Badly Shaped Solid Elements**: First order elements that are flat or near flat are allowed in the model. When finding such elements, the software will ignore them. This complements the improvement in 17.0 where second element elements with problems were accepted.

**Map Density/Dimensions to Lattice**: This function allows the user to map density/dimensions from solid geometry to an existing lattice geometry (STL format) to vary the dimension/thickness of the lattice.

**Other Enhancements in UI:** Other enhancements in UI include collapsing the property groups in Analysis Settings, results request for different analysis types and etc.

# 3 Structural Optimization Enhancement

1. Additive Overhang Angle Constraints

The options are: Build from top, bottom of X, Y or Z axis (ATX, ATY, ATZ or ABX, ABY or ABZ). An overhang angle need to be specified.


Topology Optimization results with overhang angle constraints

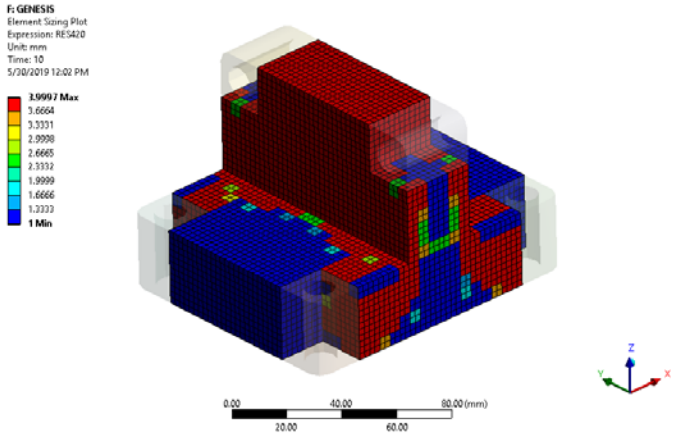
The figure above shows a comparison between standard topology and 3 alternative answers using different overhang angle constraints limits (30, 45 and 60 degrees)

2. Maximum member size improvements

The existing method to impose maximum member size has been tune-up to get more spreaded answers.

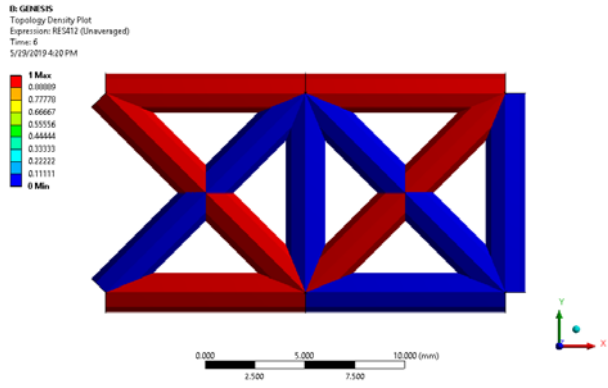
3. Optimization of Homogenized Lattice Structures

Sizing and/or topometry can be applied to the solid elements with homogenized lattice properties. GENESIS can use homogenized materials that represent either some new built-in lattice patterns or user supplied lattice patterns.



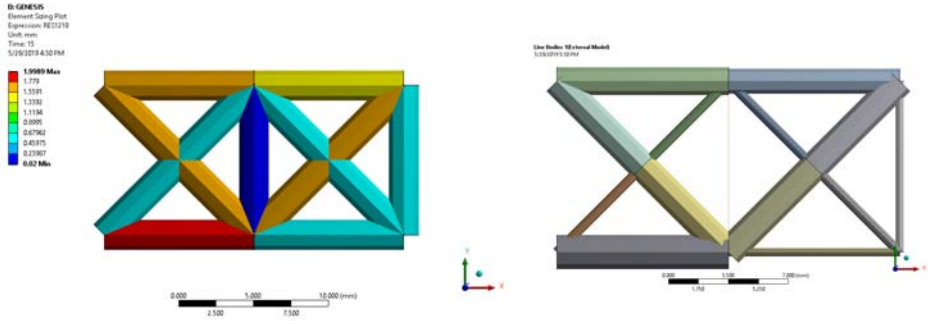

4. Support Bar/Beam in Topology Optimization

The elements in a wire body (bar/beam elements) can be designed with topology optimization.




5. Support Bar/Beam in Sizing/Topometry Optimization

The cross-sectional dimensions of the bar/beam can be designed with sizing/topometry optimization.



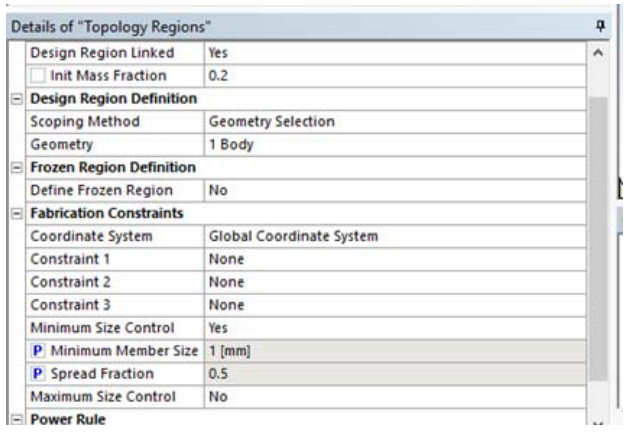
# 4 Finite Element Enhancement

1. **Tolerance for Badly Shaped Solid Elements**: First order elements that are flat or near flat are allowed in the model. When finding such elements, the software will ignore them. This complement the improvement in 17.0 where second element elements with problems were accepted.

# 5 Enhancements in Analysis and Optimization Setup

1. Support Parameterization in Optimization Settings and Design Data Definitions

Many parameters in analysis and optimization settings or in the design data definitions can be parameterized using standard ANSYS parameterization technique. And these parameters can be used in parametric study, DOE or optimization.



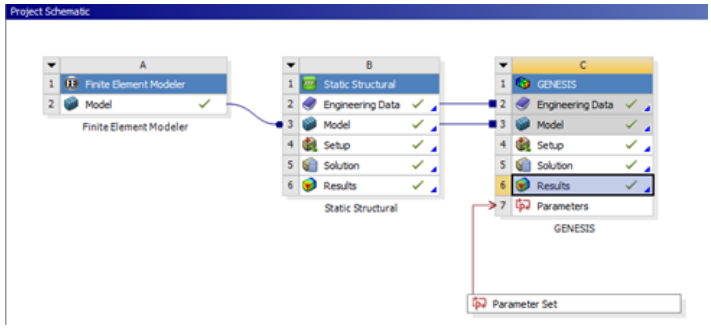




Table of Design Points

| | A | B | C | D | |
|---|---|---|---|---|---|
| 1 | Name | P2 - Topology Regions Spread Fraction | P3 - Topology Regions Minimum Member Size | Retain | Reta |
| 2 | Units | | mm | | |
| 3 | DP 0 (Current) | 0.5 | 1 | ☑ | ✓ |
| 4 | DP 1 | 0 | 1 | ☑ | ⚡ |
| 5 | DP 2 | 0.25 | 1 | ☑ | ⚡ |
| 6 | DP 3 | 0.75 | 1 | ☑ | ⚡ |
| 7 | DP 4 | 1 | 1 | ☑ | ⚡ |
| 8 | DP 5 | 0 | 1.5 | ☑ | ⚡ |
| 9 | DP 6 | 0.25 | 1.5 | ☑ | ⚡ |
| 10 | DP 7 | 0.5 | 1.5 | ☑ | ⚡ |
| 11 | DP 8 | 0.75 | 1.5 | ☑ | ⚡ |
| 12 | DP 9 | 1 | 1.5 | ☑ | ⚡ |

2. Enhancement in Random Analysis Setup

An extra modal analysis will be created automatically if performing random analysis with enforced boundary conditions. The large mass method in GENESIS for calculating enforced boundary conditions will be applied on this extra modal loadcase, not on the original modal analysis. Therefore, the extra modes introduced due to large mass method won't affect the original modal analysis. The user can still use the regular mode number in optimizing the original modal analysis.

3. Enhancement in ESL Analysis Setup

A new option is added in optimization with ESL analysis. The user can choose to add soft springs for all geometries in the model. This helps the case when there are unsupported APDL commands in boundary conditions definition in the ANSYS analysis. The extra soft springs helps removing the rigid body motion and produces correct analysis result in an ESL analysis.

4. Request RMS and PSD results in a random analysis

Now the RMS result and PSD result can be requested separately.

By default, for RMS result, the CRMS result won't be exported. This helps reducing file size.

If RMS results are requested, or the PSD results for stress and strain are requested, the output file format will be set as PUNCH automatically since these type of results are not supported in binary format.

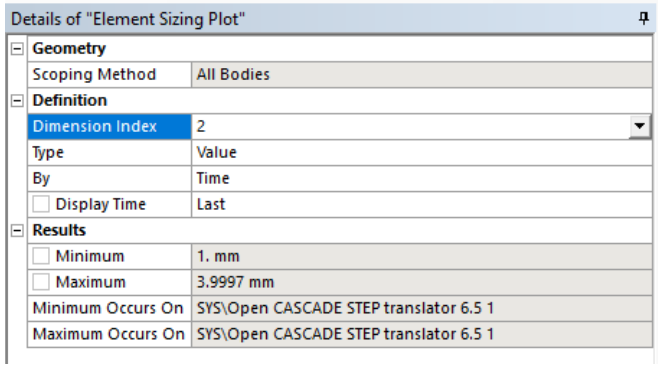
5. Request results for different analysis type

The user can request results to be printed in output file based on the analysis type. This gives more flexibility in requesting results. It helps reducing file size especially for large models.

# 6 Post-processing Enhancement

1. Support Post-processing for topology density result on wire bodies (bar/beam)

2. Support Post-processing for element sizing result on wire bodies (bar/beam) or solid elements with homogenized lattice properties. If the cross-section contains more than one dimension, correct Dimension Index need to be selected to view the corresponding results.

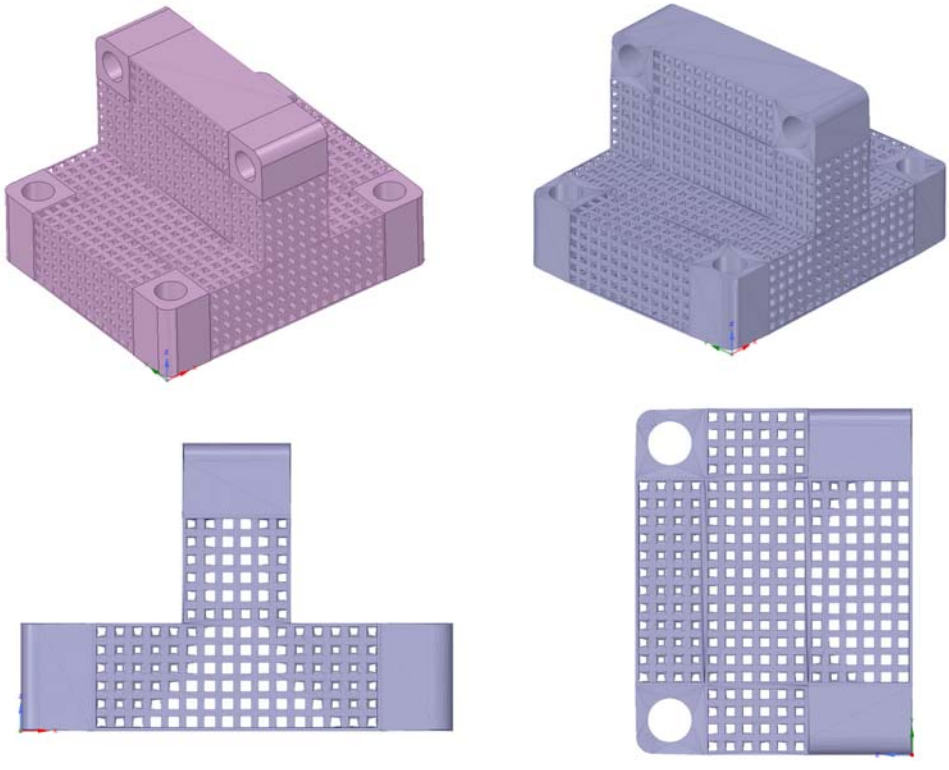


# 7 New Utility Tools

1. Map Density/Dimensions to Lattice

This function allows the user to map density/dimensions from solid geometry to an existing lattice geometry (STL format) to vary the dimension/thickness of the lattice.

| Details of "Map Density/Dimension to Lattice" | |
|---|---|
| **Region to be Mapped** | |
| Scoping Method | Geometry Selection |
| Geometry | 1 Body |
| **Optimization Result** | |
| Type | Element Sizing |
| **STL File** | |
| File Name | C:\Users\hdong\qa\v18\addtivie\lattice\bracket_lattice.stl |
| **Lattice Dimensions** | |
| Lattice Cell Size | 5 [mm] |
| Original Member Size | 2 [mm] |
| Allowable Min. Member Size | 1 [mm] |
| Allowable Max. Member Size | 4 [mm] |
| **Offset Option** | |
| Vertex Normal Weight | By Angle |

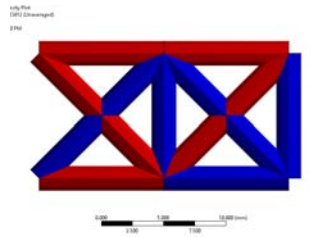
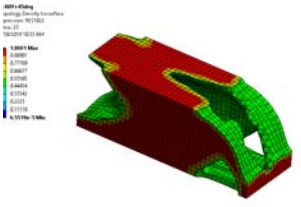
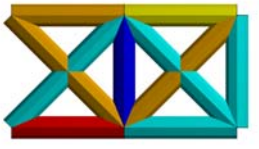
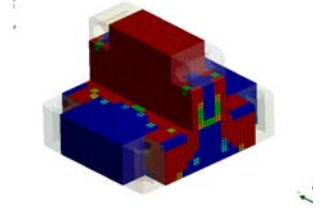


Original uniform lattice and mapped lattice with variable sizes

2. Collapse/Expand Analysis Settings Group

By default, most of the properties group in Analysis Settings are collapsed. The user can use this tool to expand or collapse all.

# 8 New Examples

| File Name | Problem Title | Special Features | Figure |
|---|---|---|---|
| ATP026 | Topology Optimization for 10-Bar Truss | • bar (beam) |  |
| ATP027 | Additive Overhang Constraints for 2D Beam | • additive overhang for 2d |  |
| ATP028 | Additive Overhang Constraints for 3D Beam | • additive overhang for 3d |  |
| AST004 | Topometry optimization of 10-bar truss | • bar (beam) |  |
| AST005 | Topometry Optimization of Homogenized Lattice | • Homogenized lattice |  |