GENESIS Structural Optimization for ANSYS® Mechanical

An Integrated Extension that adds Structural Optimization to ANSYS® Environment

New Features and Enhancements

Release 2017.03

***GENESIS* VERSION 16.0**

Mar 2017

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 16.0, which matches the version of *GENESIS* engine. Please note that GSAM/GTAM version 16.0 does not work with versions of *GENESIS* that are older than 16.0

2 Executive Summary

The key new features and enhancements include the following:

Topology Optimization for Heat Transfer Analysis: Heat transfer analysis now is supported for topology optimization. Temperature response can be used for topology optimization with heat transfer analysis. An additional new response, Heat Transfer Compliance (HTC), is added for heat transfer load case.

New Fabrication Constraints for Topology: New no-hole option is added for filling fabrication constraints. This new option will not create through holes for filling.

External Eigenvalue Solver (EES): Now it is possible to use ANSYS as external eigenvalue solver during optimization for Modal, Harmonic, or Random Vibrations.

Standalone ESL for ANSYS: Optimization based on ESL method for ANSYS now can be run directly from command line instead of within ANSYS Mechanical.

RSM Support for ESL based Optimization: ESL based optimization is supported for ANSYS Remote Solve Manager (RSM).

RSM Support for EES based Optimization: EES based optimization is supported for ANSYS Remote Solve Manager (RSM).

Support for ANSYS Superelement: ANSYS superelement is now supported in GSAM/GTAM.

Using Updated External Thermal Load (ETL): The user can now use ANSYS thermal analysis to compute the updated ‘Imported Body Temperature’ for optimization of thermal-mechanical coupled systems

Updated DOT Optimizer: *GENESIS* now uses DOT 7.2 which is the latest version of the DOT optimizer.

3 Structural Optimization Enhancement

1. Topology Optimization for Heat Transfer Analysis: Heat transfer analysis now is supported for topology optimization. Temperature response can be used for topology optimization with heat transfer analysis. An additional new response, Heat Transfer Compliance (HTC), is added for heat transfer load case.

2. Heat Transfer Compliance Response: In thermal load case, the heat transfer compliance (HTC) is defined as $HTC = \frac{1}{2}\{T\}^T\{F\}$, where {T} is the vector of grid temperatures, and {F} is the vector of applied heat fluxes. When HTC is minimized, temperature at grids where heat fluxes are applied is minimized.

3. Strain Response: Strain response are now available through GSAM/GTAM interface.

4. Grid Stress: Grid stress and dynamic grid stress are now available through GSAM/GTAM interface.

5. Updated DOT: The latest version of the DOT optimizer, 7.2, is now available with *GENESIS*.

4 Topology Optimization Enhancement

1. No Hole Option for Filling: A new option is added for filling fabrication constraints. With no hole option, topology design will not create through holes. No hole option is available for filling from general plane (FGX, FGY, FGZ), filling from top (FTX, FTY, FTZ), and filling from bottom (FBX, FBY, FBZ).

5 External Eigenvalue Solver

Instead of using *GENESIS* eigenvalue solver for frequency calculation, the user can now use ANSYS eigenvalue solver as the solution method. At each design cycle, *GENESIS* will call the **E**xternal **E**igenvalue **S**olver (ANSYS in this case) to solve the eigenvalue problem, while *GENESIS* is used for optimizing the structure.

The user can use EES to solve optimization for Modal, Harmonic, or Random Vibrations.

6 Standalone ESL for ANSYS

Instead of solving the ESL based optimization within ANSYS environment, the user can export the necessary files and run ESL for ANSYS as a standalone program. The user also needs to set environment variables pointing to the *GENESIS* executable location and ESL for ANSYS install location on the machine where the optimization will be performed. The details are explained in the Users Manual.

7 Support for ANSYS RSM

In previous versions, ANSYS RSM is only supported for optimization using *GENESIS* solver only. In this new version, the user can run optimization based on ESL method or EES method which requires both *GENESIS* and ANSYS through RSM tools. The user needs to set environment variable called VRAND_ESLANSYS_INSTALL on the RSM server machine. This environment variable should point to the location of ‘eslansys’ folder which comes with the extension installation.

8 Support for ANSYS Superelement

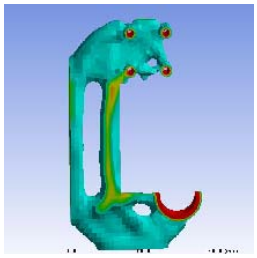
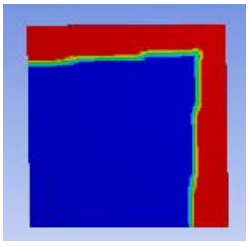
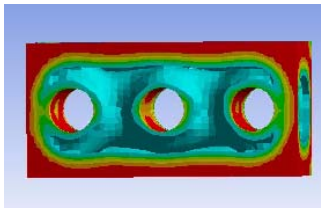
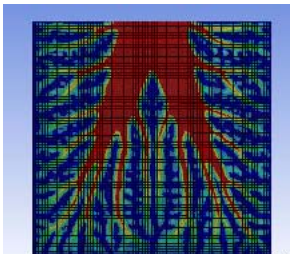
Substructuring is a procedure that condenses a group of finite elements into one element represented as a matrix. This matrix is called superelement. In GSAM/GTAM, the user can use superelement generated by ANSYS in a *GENESIS* analysis or optimization. Typically, the user can reduce the non-designable region as superelement. With GSAM/GTAM extension, the procedures for utilizing ANSYS superelement is differentiated in two cases based on if the reduced part/body is part of the ANSYS Mechanical model.

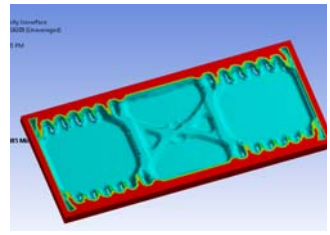
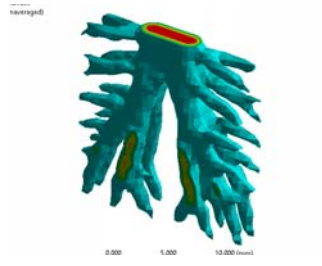
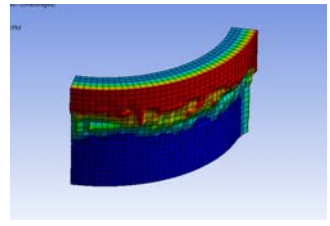
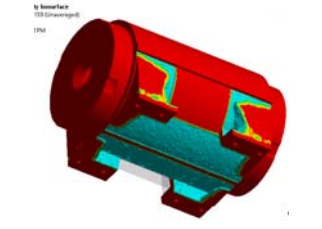
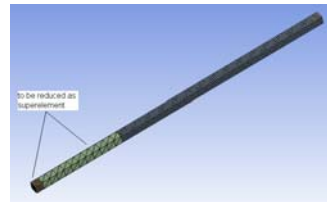
The details about how to setup a *GENESIS* simulation using ANSYS superelement is described in Users Manual.

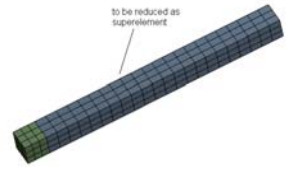
9 Using Updated External Thermal Load (ETL)

In case of thermal-mechanical coupled systems, instead of using *GENESIS* for thermal analysis, the user can now use ANSYS thermal analysis to compute the updated ‘Imported Body Temperature’ for static system. The optimization will be performed for the static system using *GENESIS*.

10 New Examples

File Name	Problem Title	Special Features	Figure
ATP019	Topology Optimization with Strain Constraints	• Define strain constraints	
ATP020	Topology Optimization for Heat Transfer	• Define Temperature as Objective	
ATP021	Topology Optimization for Thermal-Mechanical Applications	• Optimize for Temperature and Stiffness	
ATP022	Topology Optimization with Heat Transfer Compliance	• Define Heat Transfer Compliance as Objective	

ATP023	Topology Filling Fabrication Constraints: Allow through holes or No Holes	• Filling Fabrication Constraints: Allow through Holes or No Holes	
ATP024	Topology Optimization with Internal Heat Generation	• Thermal topology parameter for Internal Heat Generation	
EES001	Freeform Optimization with External Eigenvalue Solver	• Optimization with External Eigenvalue Solver (EES)	
EES002	Topology Optimization with Prestressed Modal Analysis	• Optimization with External Eigenvalue Solver (EES) for prestressed modal analysis	
AAT001	Use ANSYS Superelement: the Part/body Reduced as Superelement is Part of the ANSYS Mechanical Model	• Steps to use ANSYS superelement in GENESIS	

AAT002	Use ANSYS Superelement: the Part/body Reduced as Superelement is NOT Part of the ANSYS Mechanical Model	• Steps to use ANSYS superelement in GENESIS	
ETL001	Topology Optimization with Imported Thermal Load	• Define ETL for using imported load from transient thermal analysis