

Topology Optimization Extension for ANSYS Mechanical

Hong Dong, Phani Adduri, and Juan Pablo Leiva

Oct 28, 2014

Outline



- GENESIS Topology Optimization Extension for ANSYS Mechanical (GTAM)
- Supported ANSYS Modeling and Analysis Features

Topology Design with GTAM

- Topology Region
- Topology Objective
- Topology Constraints
- Solution Information and Live Design History Plot
- Post Processing
- Export Optimized Geometry
- Examples
- Summary

GENESIS Topology Extension for ANSYS Mechanical (GTAM)



- Easily add Topology optimization to existing ANSYS
 Workbench workflow
- Easy and fast to create Topology optimization data
- Easy and fast to post-process Topology optimization results
- Export optimized geometry in STL or IGES format

GENESIS Topology Extension for ANSYS Mechanical via ACT



Supported ANSYS Analysis Type

- Static
- Modal
- Linear Buckling
- Harmonic
- Random Vibrations
- Support Multiple Analysis Systems
- Support Multi-Step Analysis

- Supported ANSYS Elements Type
 - First and second order solid elements (Pyramid element is now supported in GENESIS 14.0)
 - First and second order shell elements
 - Beam element
 - Point Mass
- Support Rigid Body Definition
- Support Isotropic, Orthotropic and Anisotropic material for both analysis and topology design

- Supported ANSYS Loading and Boundary Conditions
 - Force, Moment, Pressure, Gravity, Acceleration, Rotational Velocity, Thermal conditions, Displacement
 - Bearing Load, Joint Load, Bolt Pretension
 - Fixed Support, Simple Support, Frictionless Support, Fixed Rotation, Cylinder Support
 - Remote Boundary Conditions
 - Constraint Equations

- Supported ANSYS Connection Type
 - Contact
 - Bonded, Frictionless, Frictional
 - Bonded No Separation (supported in GENESIS 14.0)
 - Springs
 - Joints
 - Beam Connections

Topology Design with GTAM



- Topology Objective
- Topology Constraints
- Solution Information and Live Design History Plot
- Post Processing
- Export Optimized Geometry

Topology Region



] GENESIS Structural Optimization 🎓 🍥 🚛 🗊 | 📐 | 🧇 💁 🍕 🤹 🤹 📙 🔚 📳 😏 📀 |

De	Details of "Topology Regions" 4				
Ξ	Design Region Definition				
	Scoping Method	Geometry Selection			
	Geometry	1 Body			
Ξ	Frozen Region				
	Define Frozen Region	Yes			
	Frozen Region Definitio	n			
	Scoping Method	Geometry Selection			
	Geometry	2 Faces			
	Definition				
	Init Mass Fraction	0.3			
	Fabrication Constraints				
	Coordinate System	symmetry			
	Constraint 1	FOX : Fill X axis (plane to + and -)			
	Constraint 2	MZX : Mirror about XZ plane			
	Constraint 3	None			
	Minimum Size Control	Yes			
	Minimum Member Size	4 [mm]			
	Spread Fraction	0.5			
	Maximum Size Control	No			

- Design Region
- Frozen Region (Optional)
- Fabrication Constraints
 - Symmetry
 - Extrusion
 - Filling
 - Filling Symmetric (new in GENESIS 14.0)
 - Stamping
 - Uniform
 - Radial Filling and Spokes (new in GENESIS 14.0)
- Minimum Member Size
- Maximum Member Size (new in GENESIS 14.0)

Topology Objective

GENESIS Structural Optimization 🌒 🗿

🗊 | 📐 | 🧇 💁 🍕 🇣 🧶 🍕 | 📇 🗐 😰 🕗 |

Details of "Topology Objectives" 4		
Definition		
Response Type	Strain Energy	
LoadCase Selection	Tabular Data	
Objective Definition	Switch	
Goal	Min 💌	
Weight	1	
Shifted	No	
	etails of "Topology C Definition Response Type LoadCase Selection Objective Definition Goal Weight Shifted	

LoadCase Select	tion		X
LoadCase	Name	Step	Selection
1	Static Structural	1	Yes 💌

Response Type

- Strain Energy
- Frequency Mode Number
- Mass Fraction
- Displacement
- Relative Displacement
- System Inertia
- Buckling Load Factor
- Dynamic Displacement
- Dynamic Velocity
- Dynamic Acceleration
- Goal
 - Min
 - Max
 - Min-Max (Beta Method)
- Support Multiple Objectives

Topology Constraints

GENESIS Structural Optimization 🏮 🎯 🚛

ا 📀 🕺 🗐 🕒 🗞 🍕 🤹 象 📙 🗐 🧃

De	Details of "Topology Constraints" 4		
-	Definition		
	Response Type	Mass Fraction	
Region All Desigr		All Designed Groups	
-	Constraint Bou	nds	
	Lower Bound	None	
	Upper Bound	0.3	
	Bound Type	Actual	

LoadCase Selection				3
				_
LoadCase	Name	Step	Selection	
1	Static Structural	1	Yes	·
				_

Response Type

- Strain Energy
- Frequency Mode Number
- Mass Fraction
- Displacement
- Relative Displacement
- System Inertia
- Buckling Load Factor
- Dynamic Displacement
- Dynamic Velocity
- Dynamic Acceleration
- Constraint Bounds
- Support Multiple Constraints

Solution Information

Genesis Insert	•		
Analy Molect			
Topo Solve	🗭 Multiple Systems - Mechanical [ANSYS Mechanical]		
Topol Clear	File Edit View Units Tools Help	7./ Show Errors 1 1 10 10 10 10 10 10 10 10 10 10 10 10	
🖻 🥠 🚱 Solut			Thislas Associations
🥠 🥵 🔤 🖉	Solution Information 🗠 Result Tracker 🛫 🖉 Retrieve		Inicken Annotations
in the second se	Joint Configure Configure Assemble $\Delta = 0$	能 Set 现象Revert	
	GENESIS Structural Optimization 🏮 🍥 🚛 🛐 🧇 🙆	🍕 🍕 🦉 🗐 🖓 👔 IronPythonConsole 🗰 🛛 ACT Development 🥃 🖵	
	Outline 4	Worksheet	4
	Filter: Name 🔻 🗘 🖉	NUMBER OF CHEXA (9-21 NODE) ELEMENTS: 3176	*
	Project	NUMBER OF REE3 ELEMENTS: 4	
	🗄 🖓 Geometry	NUMBER OF ELEMENT PROPERTIES: 2896 NUMBER OF MATERIALS: 2893	
	Coordinate Systems	NUMBER OF DEGREES OF FREEDOM: 97228	
	⊡	TOPOLOGY OPTIMIZATION PROBLEM SUMMARY	
	Static Structural (A5)	TOTAL NUMBER OF TOPOLOGY INDEP. VARIABLES: 2080	
	Analysis Settings	NUMBER OF TOPOLOGY DEPEN. DESIGN VARIABLES: 2892	
	Cylindrical Support	TOTAL NUMBER OF TOPOLOGY DESIGN VARIABLES: 4972 NUMBER OF TOPOLOGY DESIGNABLE ELEMENTS: 2892	
	Frictionless Support	NUMBER OF TOPOLOGY DIRECT RESPONSES: 2	
	Pressure	NUMBER OF POTENTIAL TOPOLOGY CONSTRAINTS: 2	
	Solution (A6)	LOAD CASES SUMMARY NUMBER OF STATIC LOAD CASES: 1	
	2 Total Deformation	TOTAL NUMBER OF LOAD CASES: 1	
	Equivalent Stress		
	GENESIS (B5)	ELAPSED TIME IN MODULE PRINTI = 0.00 SEC, IN TOTAL = 17. SEC	
	Analysis Settings	CPU TIME SPENT IN MODULE PRINTU = 0.00 SEC, IN TOTAL = 17. SEC	E
	Topology Objectives	ELAPSED TIME IN MODULE PRINTU = 0.00 SEC, IN TOTAL = 18. SEC	
	Topology Constraints	ELAPSED TIME IN MODULE ESL = 0.00 SEC, IN TOTAL = 18. SEC	
	Solution (B6)	START DESIGN CYCLE 0	
	Topology Density Plot	CPU TIME SPENT IN MODULE FEH = 0.00 SEC, IN TOTAL = 17. SEC	
	Topology Density Isosurface	CPU TIME SPENT IN MODU ANSYS Workbench Solution Status	
	Details of "Solution Information"	ELAPSED TIME IN MODU Overall Progress	· ·
	Solution Information	Graphics worksneet	
	Solution Output Solver Output	Messages Solving the mathematical model	
	Newton-Raphson Residuals 0	Text	
	Update Interval 2.5 s	Stop Solution	
	1		

Progress Update in Solution Information Worksheet View

Live Design History Plot





- Element Density
- Density Isosurface
- Displacement
- Element Stress
- Grid Stress
- Strain





- GENESIS Structural Optimization 🏮 🍥 🚛 🚯 📐 🗇 🤷 🦓 🍕 🗣 🎼 📜 🕎 🕗
- File Format
 - STL and IGES
- Coarsening Level
 - Coarse to Fine
- Export Visible Parts Only



Optional: Write GENESIS Input File and Launch Design Studio for GENESIS

GENESIS Structural Optimization 🏮 🍥 🚛 🛐 📃 📐 🛛 🧇 🤷 🍕 🍕 🦉 🗏 🔲 🗐 🧕 🧿

- Write Genesis Input
 - Mesh, materials, properties, load and boundary conditions, design data

Launch Design Studio For Genesis

- Model is imported to Design Studio automatically
- Access additional functionality

Analysis Settings



- Design Control
- Analysis Control
- Output Control
- Coarsened Surface

De	Details of "Analysis Settings" 4			
	Design Control			
	Max. Design Cycles	15		
	Fractional Move Limit	1E-06		
	Min. Move Limit	0.2		
	Hard Relative	0.001		
	Hard Absolute	0.001		
	Hard Max. Violation	0.005		
	Analysis Control			
	Processors	2		
	Memory (MB)	Default		
	Console Output	Off		
	Diag. Level	85		
	GENESIS Solver	On		
	GENESIS Solver Directory	C:\Program Files (x86)\vrand\genesis		
	Design Studio Directory	C:\Program Files (x86)\vrand\genesis		
Ξ	Design Output Control			
	Topology Density	Yes		
	Output for Design Cycles	Last		
	Analysis Output Control			
	Deformation	Yes		
	Stress	Yes		
	Strain	No		
	Output for Design Cycles	Last		
	Post-Processing Control			
	Import for Design Cycles	Last		
	Coarsened Surface			
	Cut Off Value	0.3		
	Level of Details (0~20)	10		
+	Analysis Data Manageme	nt		

Topology Design of A Connecting Rod



Problem Statement:

To obtain a stiff structure and satisfy a given mass requirement

- Objective
 - Min. Strain Energy
- Constraint

Mass Fraction < 0.3

Designable Region

Connecting rod Symmetric manufacturing constraint



ANSYS Workbench Flow



Add GENESIS extension to workflow



Set up Analysis Model in ANSYS



- Create geometry in ANSYS Design Modeler
- Meshing, define load and boundary conditions in ANSYS Mechanical



Define Topology Regions



Define Topology Objectives



GENESIS Structural Optimization 🍵 🍥 爀 🚯 🛛 📐

Outline 4				
Filter: Name	•	😰 🕢 H		
Project Project Model (A4, Geomet Coordin C	B4) ry ate Systems tions Structural (A5) malysis Settings ked Support vindrical Support ictionless Support essure oblution (A6) Solution Information Distical Deformation Distical Deform			
	🔗 Topology Density Plot	-		
Details of "Topology Objectives"				
Definition				
Response Type	Strain Energy			
LoadCase Selection	Apply	Cancel		
Objective Definition	Switch			
Goal	Min			
Weight	1			

De	Details of "Topology Objectives"		
Ξ	Definition		
	Response Type	Strain Energy 🔹	
	LoadCase Selection	Strain Energy	
Ξ	Objective Definition	Frequency Mode Number Mass Fraction	
	Goal	Displacement	
	Weight	System Inertia	
		Buckling Load Factor	
	Select Response Type		

🧇 🙆 🗣 🍕 🍕 🍕 📃 🗐 💇 📀

	LoadCase Select	tion		X	
	LoadCase	Name	Step	Selection	
	1	Static Structural	1	Yes 💌	
3					ł

Specify loading cases for this objective

Define Topology Constraints

Q .

GENESIS Structural Optimization 🏻 🌍

🖥 📐 🗇 💁 🍕 🍕 🍕 🧶 📃 🗐 🤒



De	Details of "Topology Constraints" 4		
-	Definition		
	Response Type	Mass Fraction	
	Region	Strain Energy	
-	Constraint Bou	Frequency Mode Number Mass Fraction	
	Lower Bound	Displacement	
	Upper Bound	System Inertia	
	Bound Type	Actual	

Select Response Type

Post-Processing



GENESIS Structural Optimization 🎁 🍥 🚛 🚯 | 📐 🔷 💁 🍕 🍕 🗣 | 📇 🗐 🧏 🕢

Topology Element Density







GENESIS Structural Optimization 🏮 🍥 🚛 🚯 🕓 🛛 🍕 🤷 🍕 🎭 目 📳 🧏 🕢

Topology Density Isosurface



Export Coarsened Surface



] GENESIS Structural Optimization 🏮 🍥 🚛 🚯 | 📐 | 🧇 🙆 🍕 🍕 🤹 🖗 🗌 📃 🧾 😏 📀 |

- Export Coarsened Surface in STL or IGES Format
- Import back to CAD



Clean Up Geometry with SpaceClaim



Validation Run in ANSYS

Deformation Result



Validation Run in ANSYS



Stress Result



Topology Optimization with Frozen Surface

Problem Statement:

To obtain a stiff structure and satisfy a given mass requirement

- Objective
 - Min. Strain Energy
- Constraint
 - Mass Fraction < 0.3
- Designable Region
 - Filling symmetric along X Symmetric about XZ plane
- Frozen Region
 - Two cylindrical surfaces





Loading and Boundary Conditions



Topology Density Isosurface



Topology Design of a Hook

Problem Statement:

To obtain a stiff structure and satisfy given mass and stress requirements

- Objective
 - Min. Strain Energy
- Constraint
 - Mass Fraction < 0.3 Stress < 65 GPa
- Designable Region

Blue region



Topology Region



Loading and Boundary Conditions



Topology Density Isosurface







- Topology optimization can be easily configured and solved in ANSYS with the extension
- Next steps: support more loadings, boundary conditions and analysis types