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)

- An integrated extension that adds Topology optimization to the ANSYS® environment
  - 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

- Add GENESIS Extension to Workflow
- Meshing
- Load
- Boundary Conditions
- Topology Region
- Objective
- Constraints
- Solve
- Element Density
- Density Isosurface
- Deformation
- Stress
- Strain
- STL or IGES

Pre-processing
Run GENESIS (in the background)
Supported ANSYS Modeling and Analysis Features

- Supported ANSYS Analysis Type
  - Static
  - Modal
  - Linear Buckling
  - Harmonic
  - Random Vibrations

- Support Multiple Analysis Systems

- Support Multi-Step Analysis
Supported ANSYS Modeling and Analysis Features

• 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 Modeling and Analysis Features

- 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 Modeling
and Analysis Features

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

- **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

- **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

- **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

Progress Update in Solution Information Worksheet View
Post-Processing

- Element Density
- Density Isosurface
- Displacement
- Element Stress
- Grid Stress
- Strain
Export Optimized Geometry

- 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

• 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
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

GENESIS Structural Optimization Toolbar
Define Topology Objectives

Select Response Type

Specify loading cases for this objective
Define Topology Constraints

Select Response Type

**Outline**
- Project
  - Model (A4, B4)
    - Geometry
    - Coordinate Systems
    - Connections
    - Mesh
  - Static Structural (A5)
    - Analysis Settings
    - Fixed Support
    - Cylindrical Support
    - Frictionless Support
    - Pressure
  - Solution (A6)
    - Solution Information
    - Total Deformation
    - Equivalent Stress
- GENESIS (B5)
  - Analysis Settings
  - Topology Regions
  - Topology Objectives
  - Topology Constraints
- Solution (B6)
  - Solution Information
  - Topology Density Plot

**Details of “Topology Constraints”**
- **Definition**
  - Response Type: Mass Fraction
  - Region: All Designed Groups
- **Constraint Bounds**
  - Lower Bound: None
  - Upper Bound: 0.3
  - Bound Type: Actual
Post-Processing

- Topology Element Density
Post-Processing

- Topology Density Isosurface
Export Coarsened Surface

- Export Coarsened Surface in STL or IGES Format
- Import back to CAD
Clean Up Geometry with SpaceClaim

Topology Isosurface STL

SpaceClaim Geometry Clean Up

Import Model back to ANSYS Mechanical
Validation Run in ANSYS

• Deformation Result

[Images showing initial and optimized deformation results]
Validation Run in ANSYS

• Stress Result

Initial

Optimized
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
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

Loading and Boundary Conditions

Topology Region

Topology Density Isosurface
Summary

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