Topology Optimization Extension for ANSYS Mechanical

Hong Dong, Phani Adduri, and Juan Pablo Leiva

Oct 28, 2014

Outline

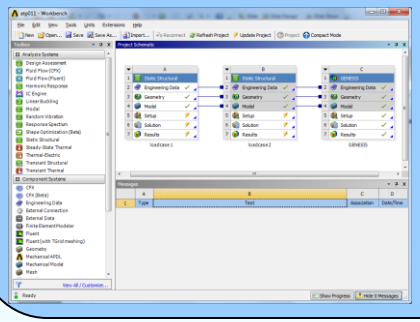
- **GENESIS Topology Optimization Extension for ANSYS Mechanical (GTAM)**
- **Supported ANSYS Modeling and Analysis Features**
- **Topology Design with GTAM**
  - Topology Region
  - Topology Objective
  - Topology Constraints
  - Solution Information and Live Design History Plot
  - Post Processing
  - Export Optimized Geometry
- **Examples**
- **Summary**

GENESIS Topology Extension for ANSYS Mechanical (GTAM)

- **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

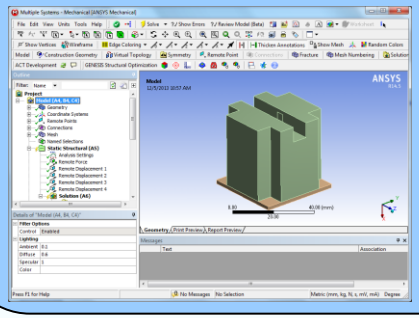
GENESIS Topology Extension for ANSYS Mechanical via ACT

ANSYS



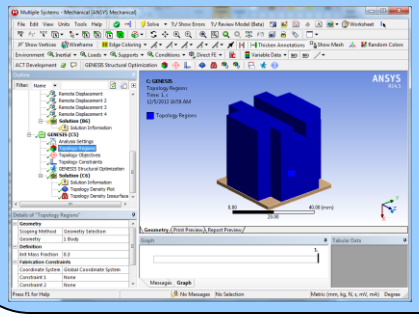
- Add GENESIS Extension to Workflow

ANSYS Pre-processing



- Meshing
- Load
- Boundary Conditions

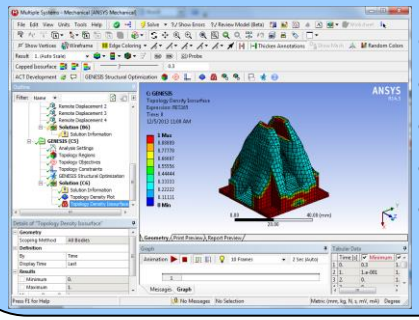
ANSYS Topology Design via ACT



- Topology Region
- Objective
- Constraints
- Solve

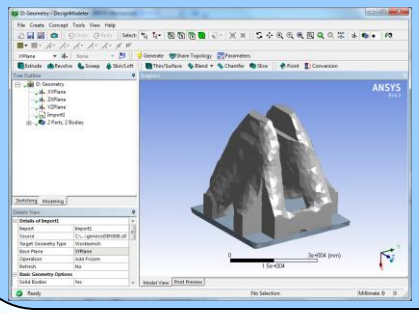
Run GENESIS (in the background)

ANSYS Post-processing via ACT



- Element Density
- Density Isosurface
- Deformation
- Stress
- Strain

ANSYS Import Optimized Geometry



- STL or IGES

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

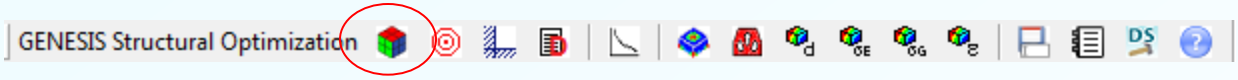
Supported ANSYS Modeling and Analysis Features

- **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

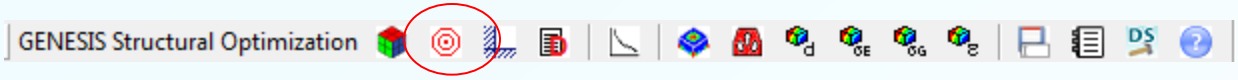
Topology Region



Details of "Topology Regions"	
Design Region Definition	
Scoping Method	Geometry Selection
Geometry	1 Body
Frozen Region	
Define Frozen Region	Yes
Frozen Region Definition	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Init Mass Fraction	0.3
Fabrication Constraints	
Coordinate System	symmetry
Constraint 1	F0X : 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



Details of "Topology Objectives"

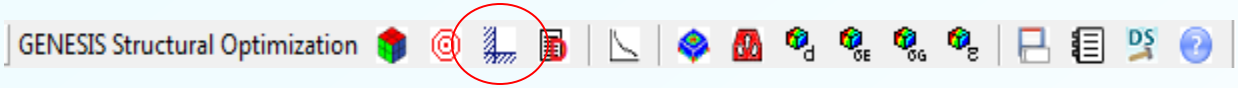
Definition	
Response Type	Strain Energy
LoadCase Selection	Tabular Data
Objective Definition Switch	
Goal	Min
Weight	1
Shifted	No

LoadCase Selection

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



Details of "Topology Constraints"

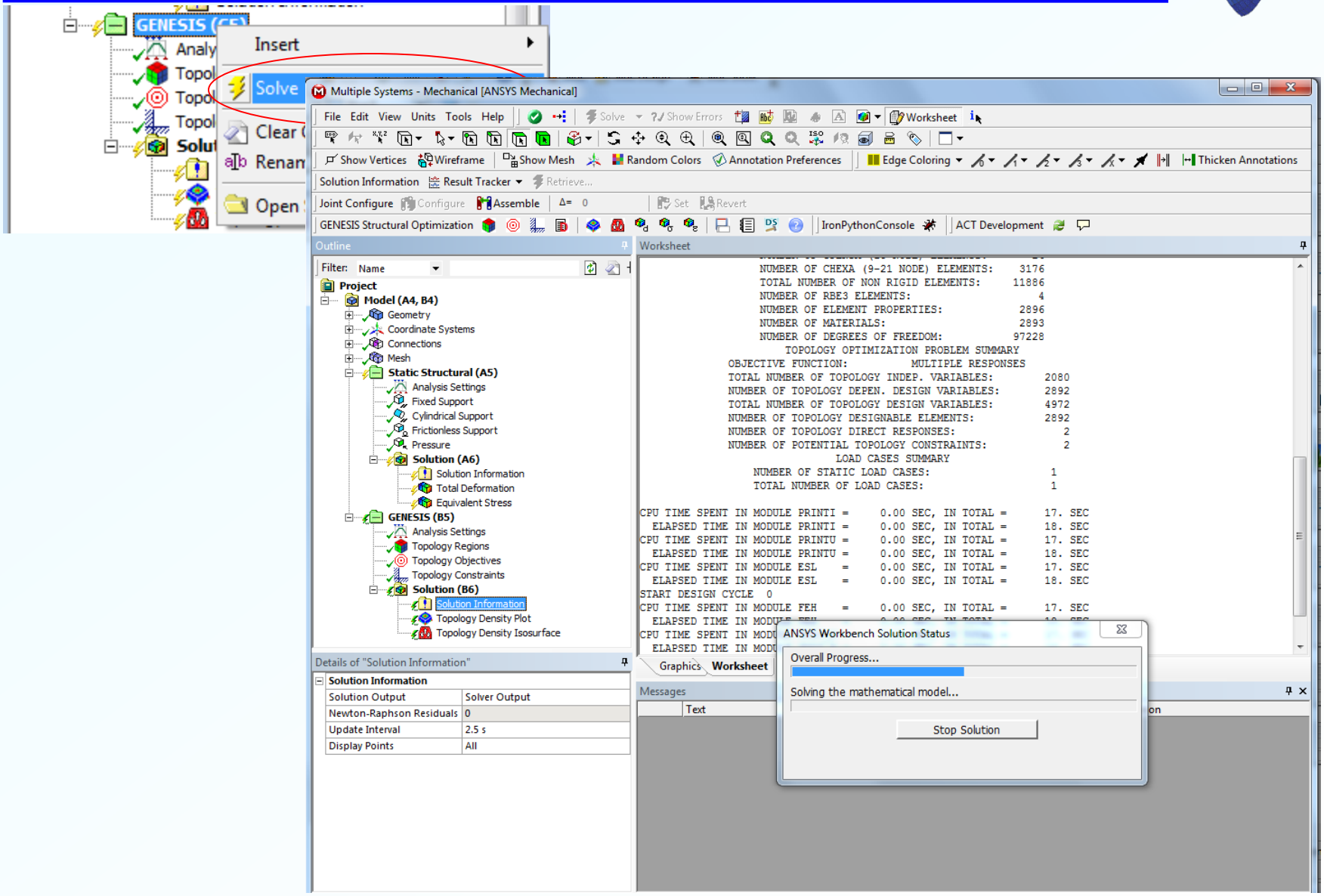
Definition	
Response Type	Mass Fraction
Region	All Designed Groups
Constraint Bounds	
Lower Bound	None
Upper Bound	0.3
Bound Type	Actual

LoadCase Selection

LoadCase	Name	Step	Selection
1	Static Structural	1	Yes

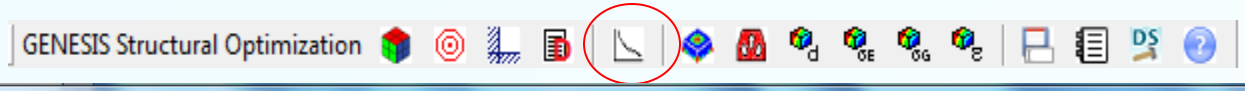
- **Response Type**
  - Strain Energy
  - Frequency Mode Number
  - Mass Fraction
  - Displacement
  - Relative Displacement
  - System Inertia
  - Buckling Load Factor
  - Dynamic Displacement
  - Dynamic Velocity
  - Dynamic Acceleration
- **Constraint Bounds**
- **Support Multiple Constraints**

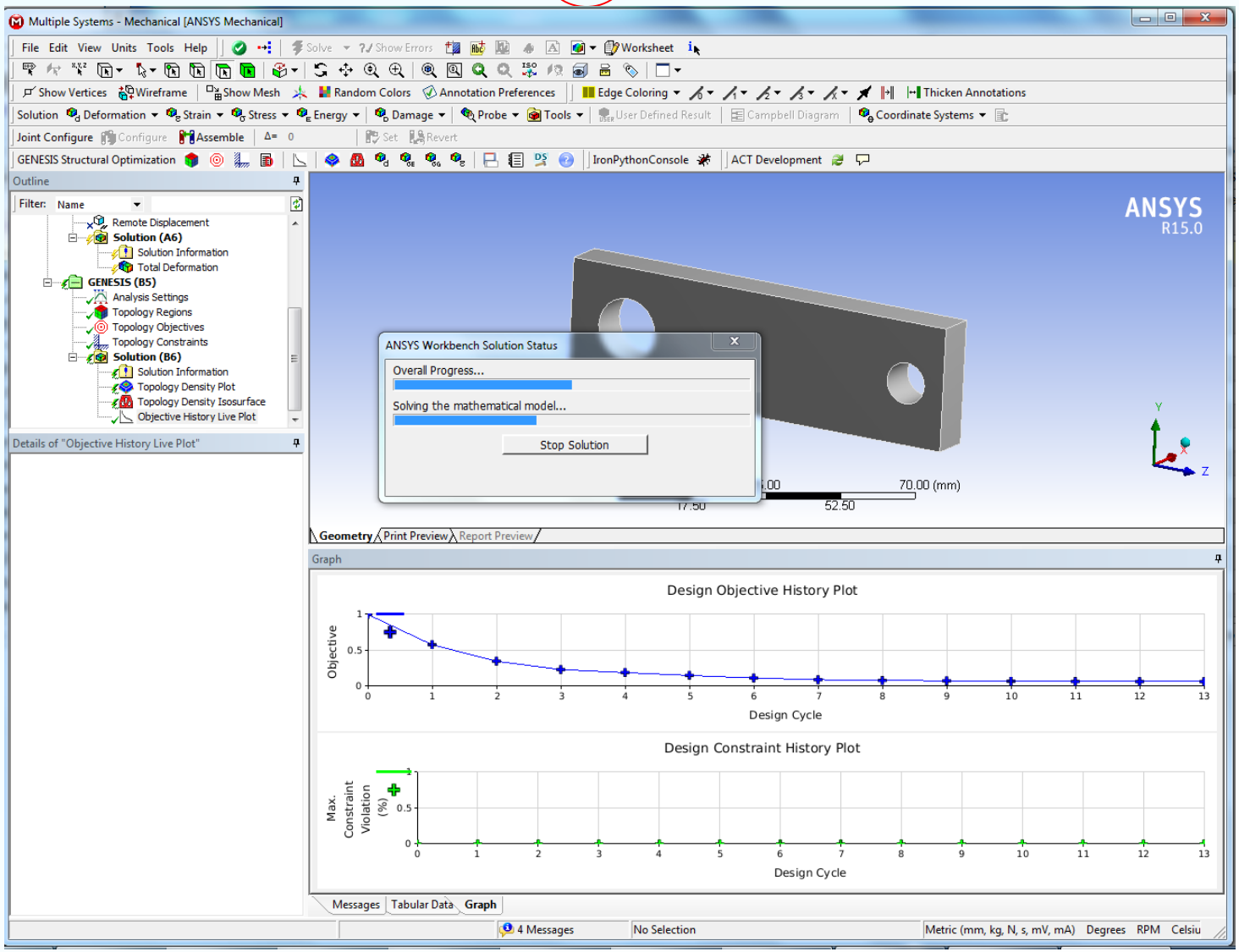
Solution Information



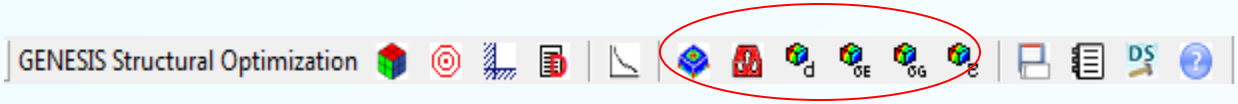
Progress Update in Solution Information Worksheet View

Live Design History Plot



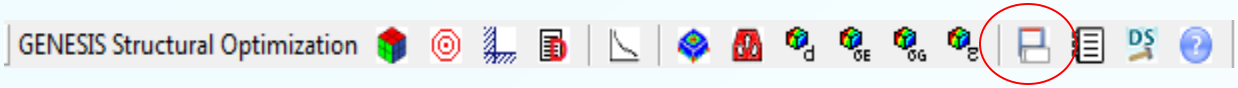


Post-Processing



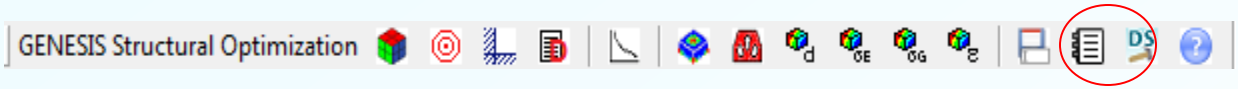
- Element Density
- Density Isosurface
- Displacement
- Element Stress
- Grid Stress
- Strain

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

Details of "Analysis Settings"

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...
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 Management	

Topology Design of A Connecting Rod

Problem Statement:

To obtain a stiff structure and satisfy a given mass requirement

- **Objective**

  Min. Strain Energy

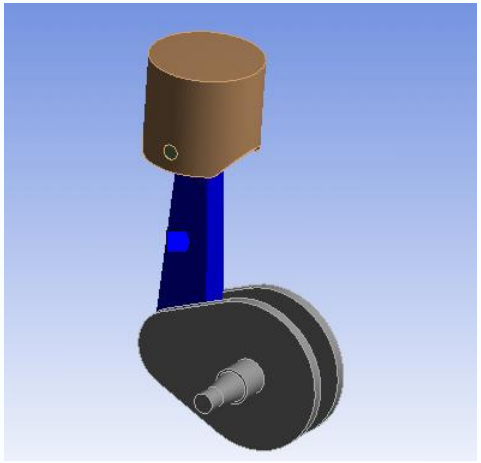
- **Constraint**

  Mass Fraction < 0.3

- **Designable Region**

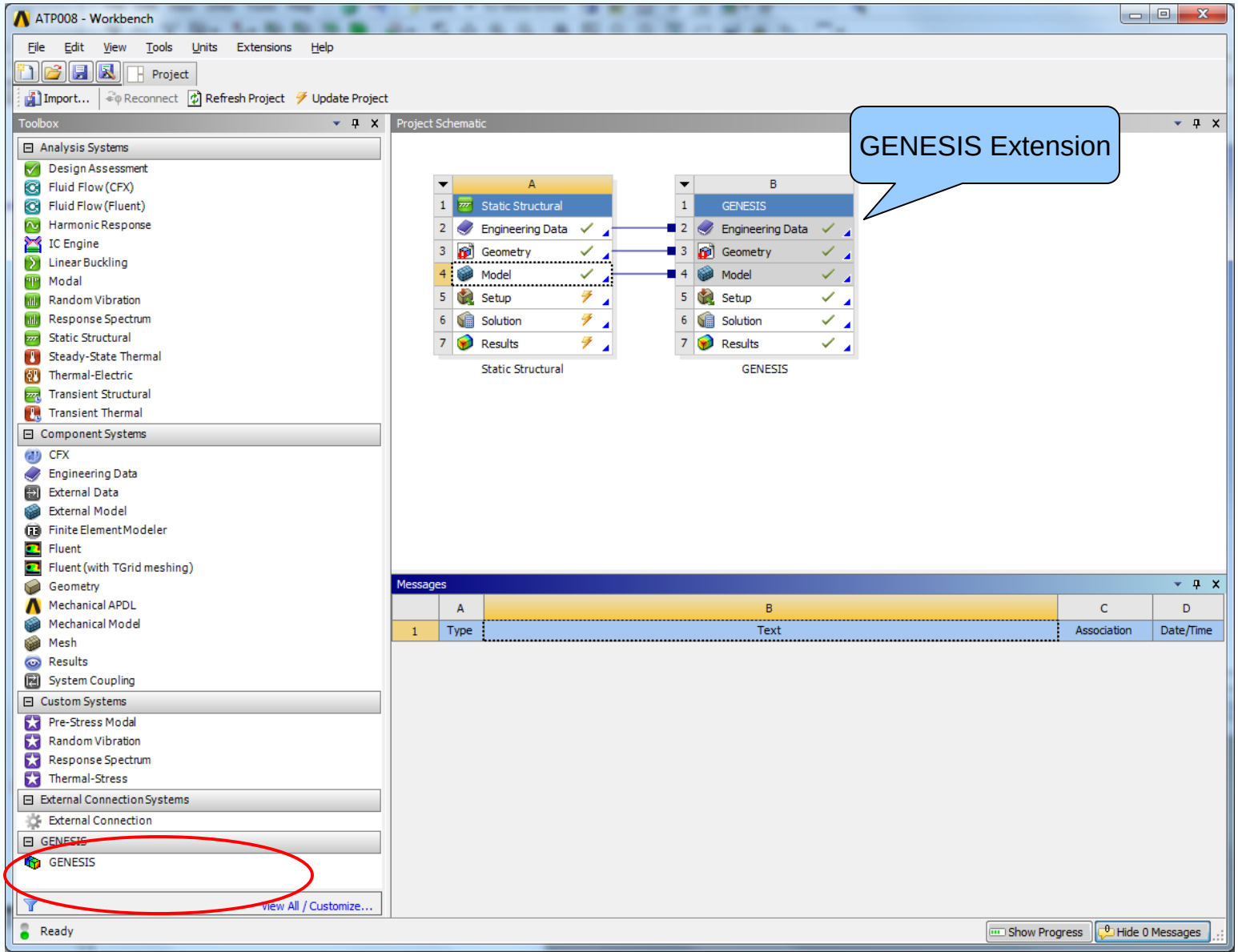
  Connecting rod

  Symmetric manufacturing constraint



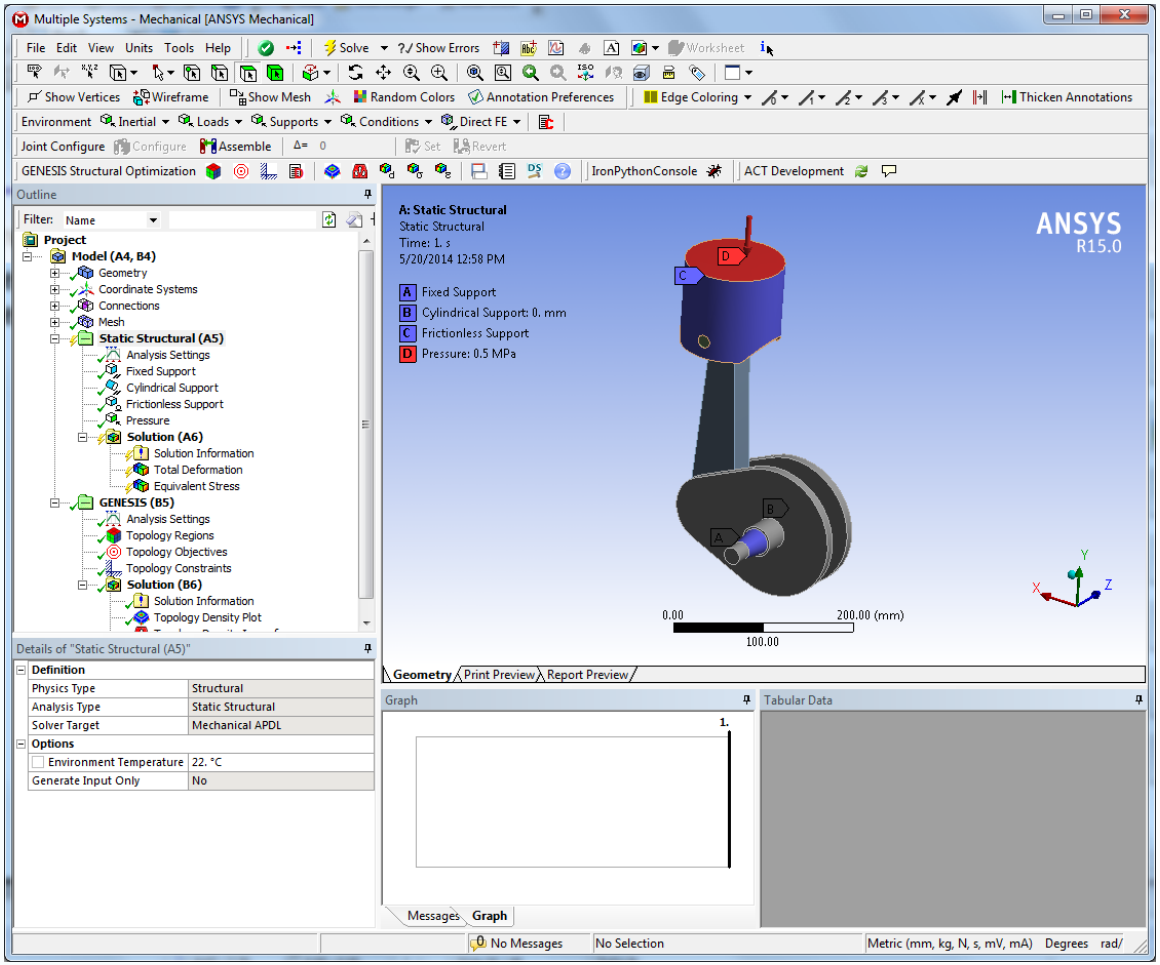
ANSYS Workbench Flow

- **Add GENESIS extension to workflow**

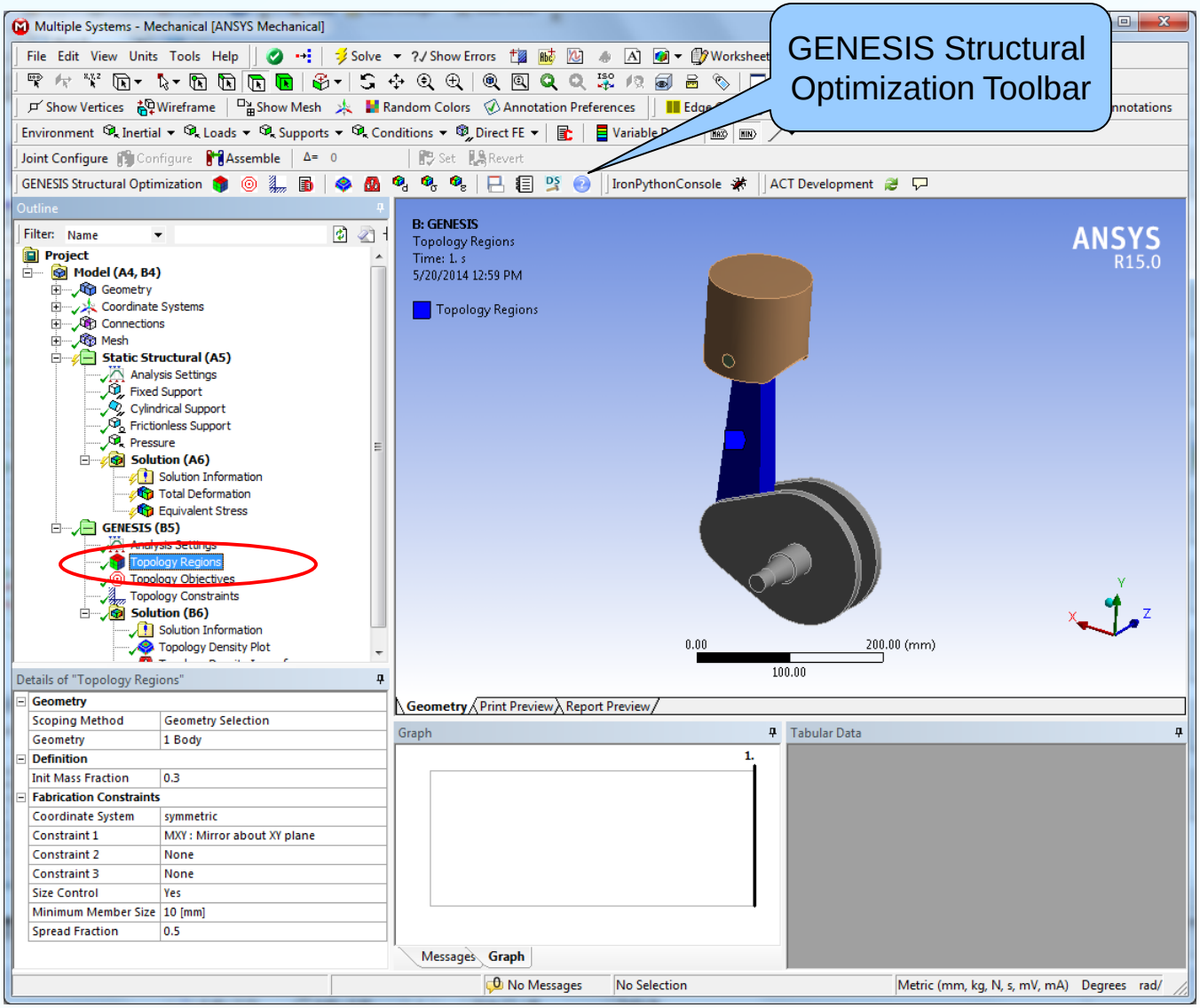


Set up Analysis Model in ANSYS

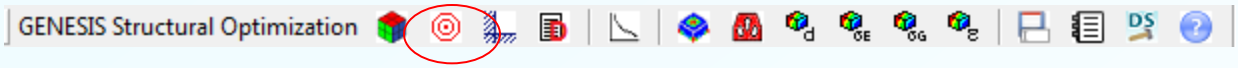
- Create geometry in ANSYS Design Modeler
- Meshing, define load and boundary conditions in ANSYS Mechanical

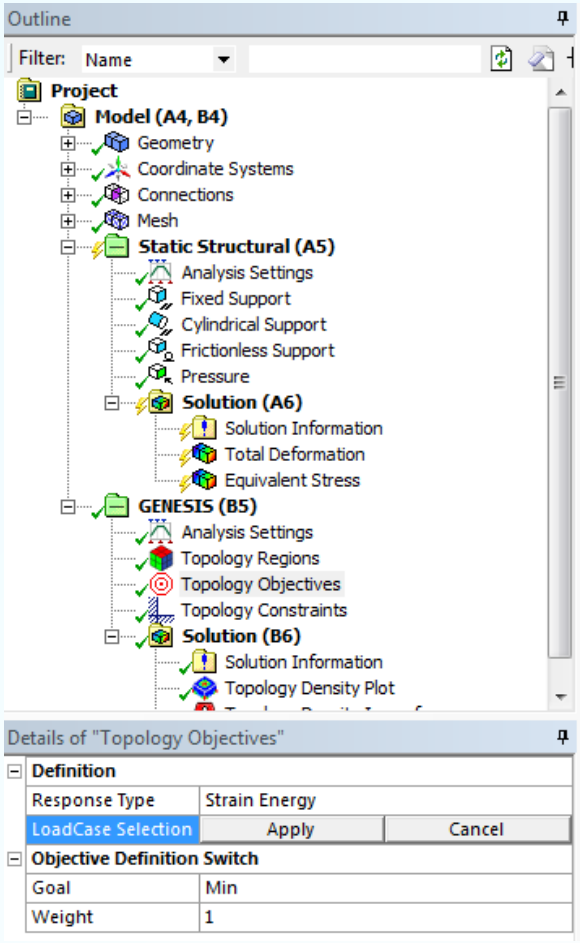


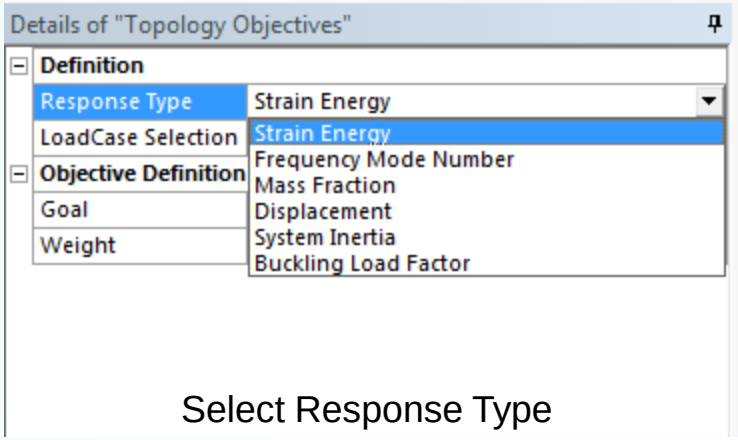
Define Topology Regions



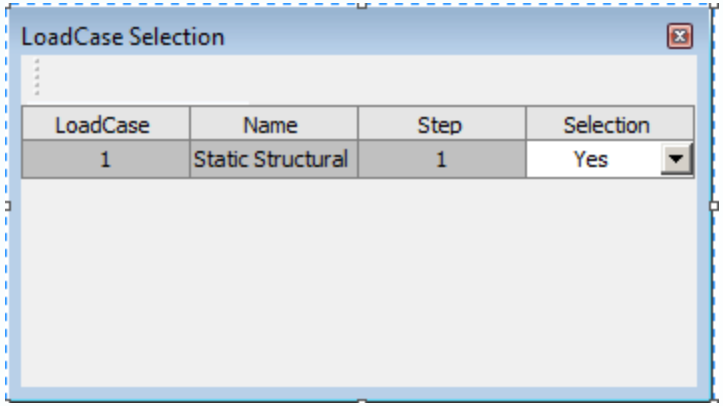
Define Topology Objectives





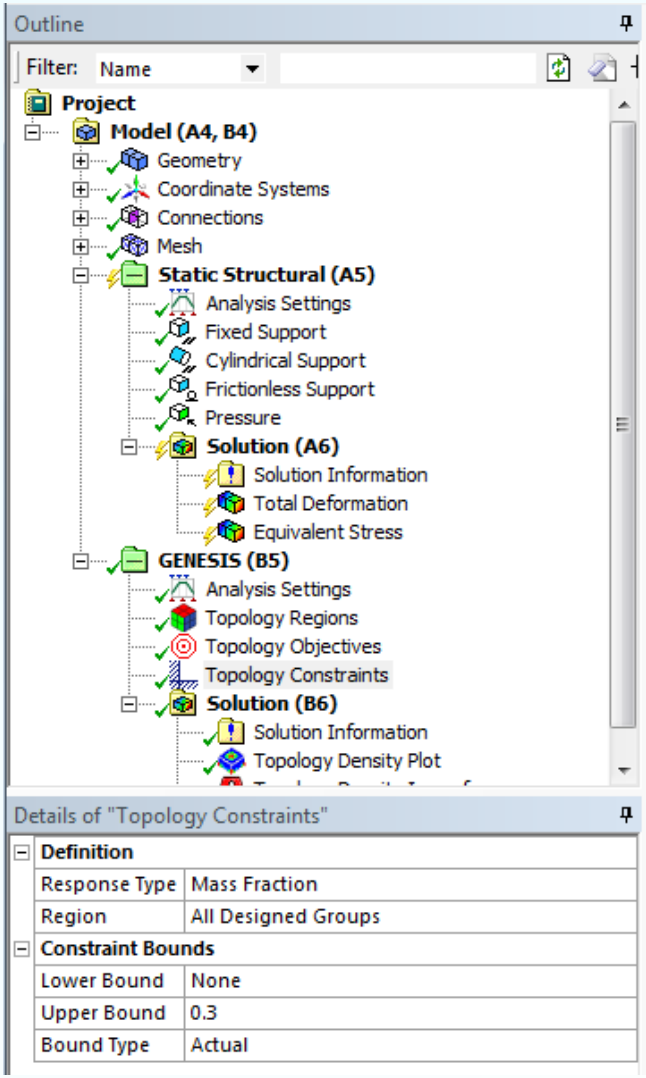


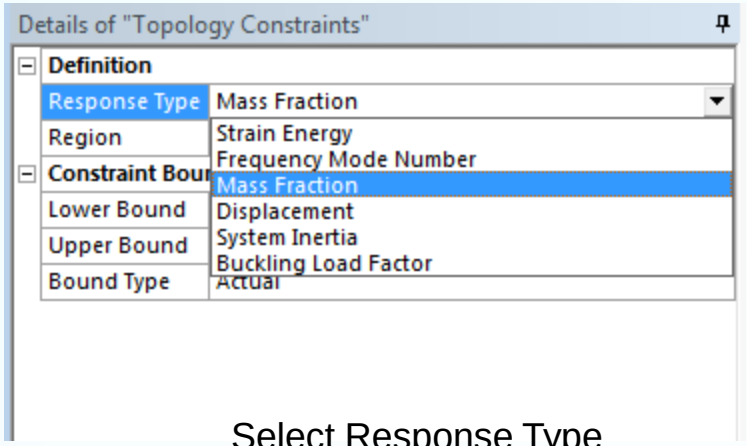
Select Response Type



Specify loading cases for this objective

Define Topology Constraints

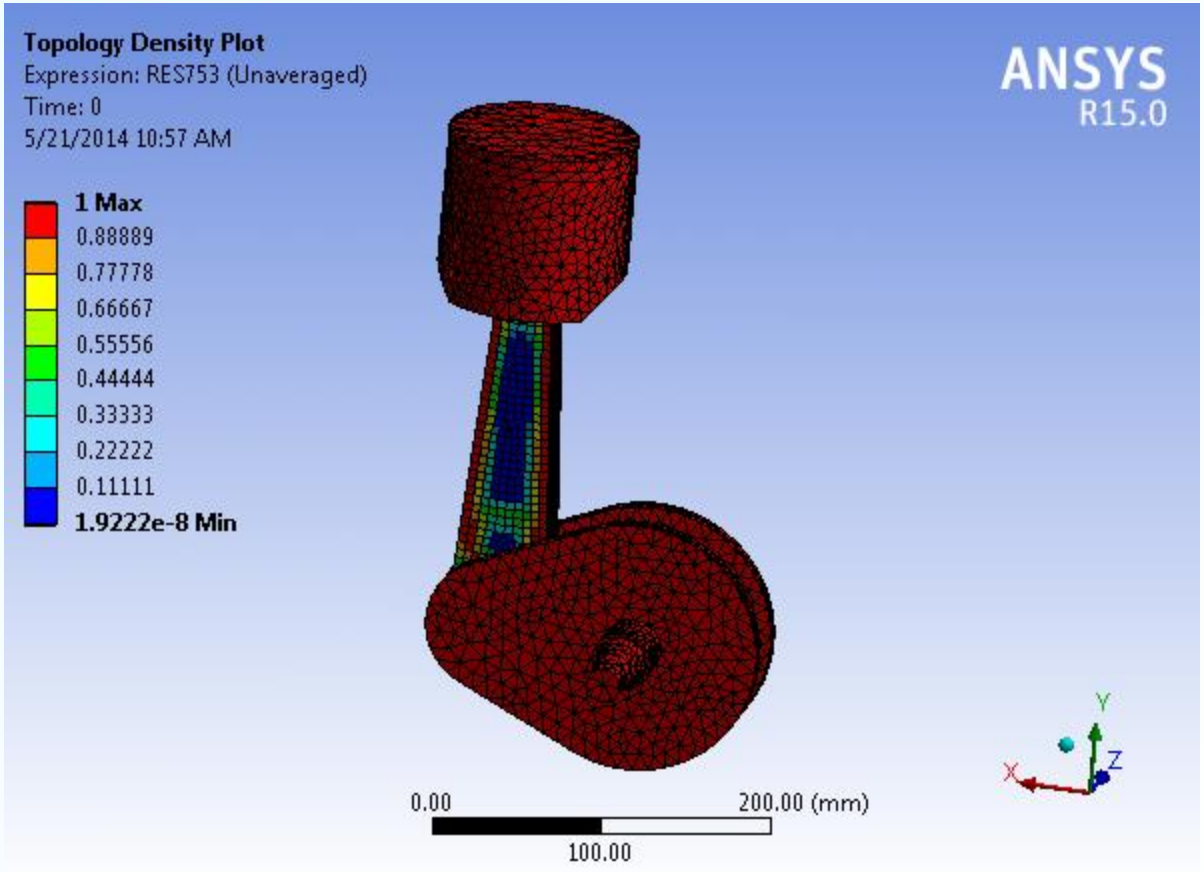




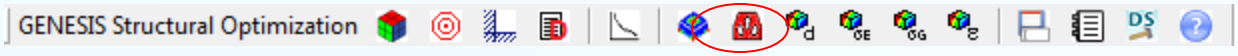
Select Response Type

Post-Processing

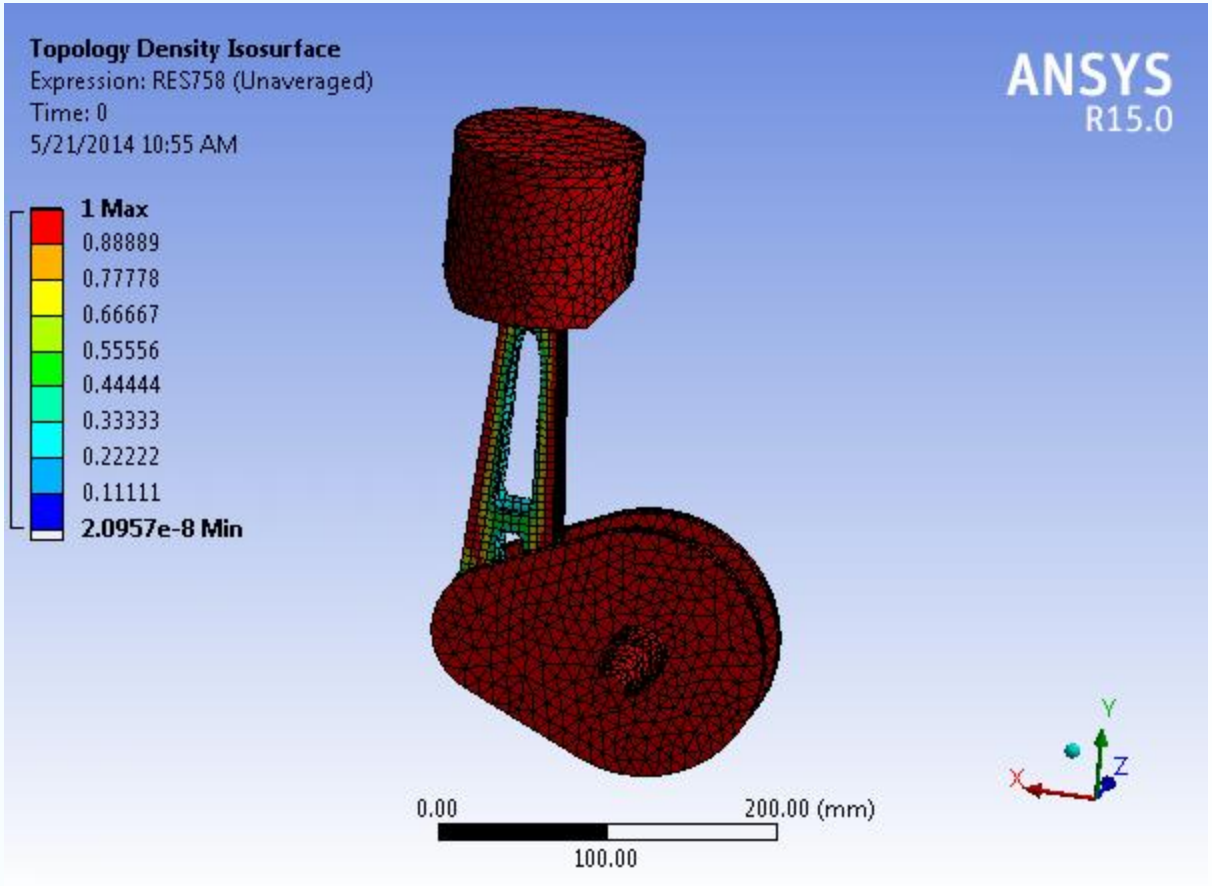
- **Topology Element Density**



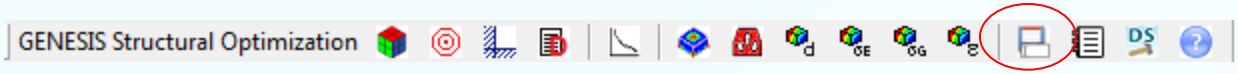
Post-Processing



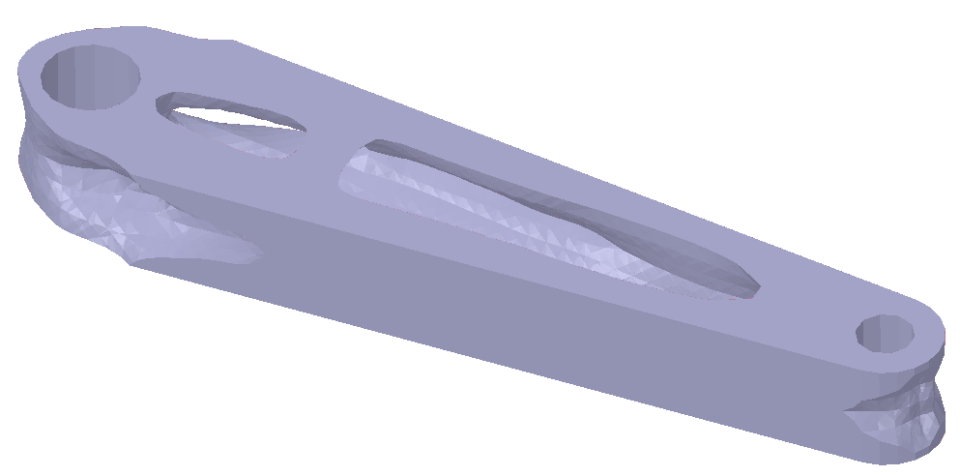
- **Topology Density Isosurface**



Export Coarsened Surface

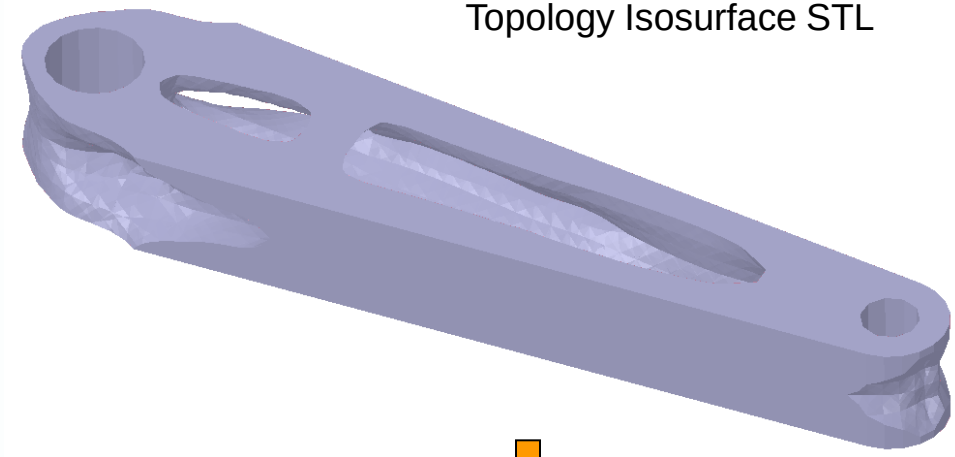


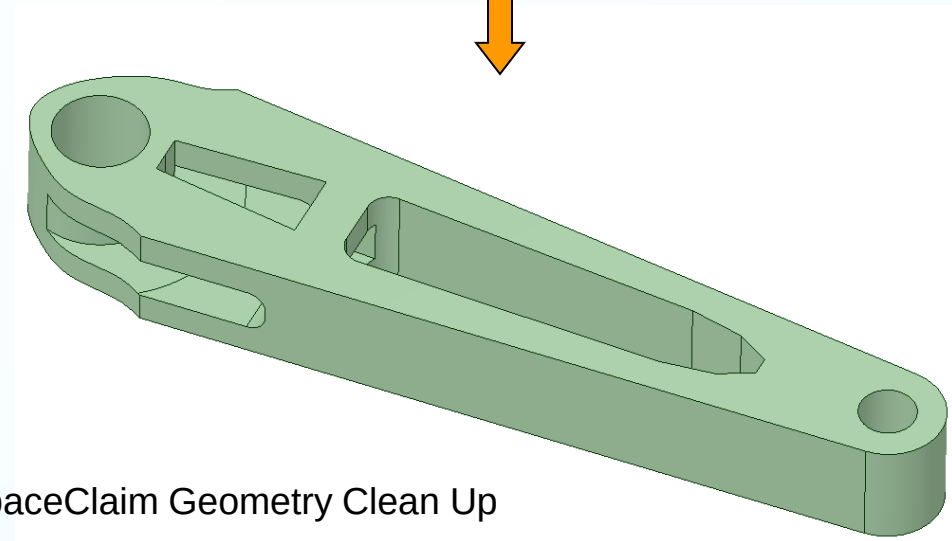
- **Export Coarsened Surface in STL or IGES Format**
- **Import back to CAD**



Clean Up Geometry with SpaceClaim

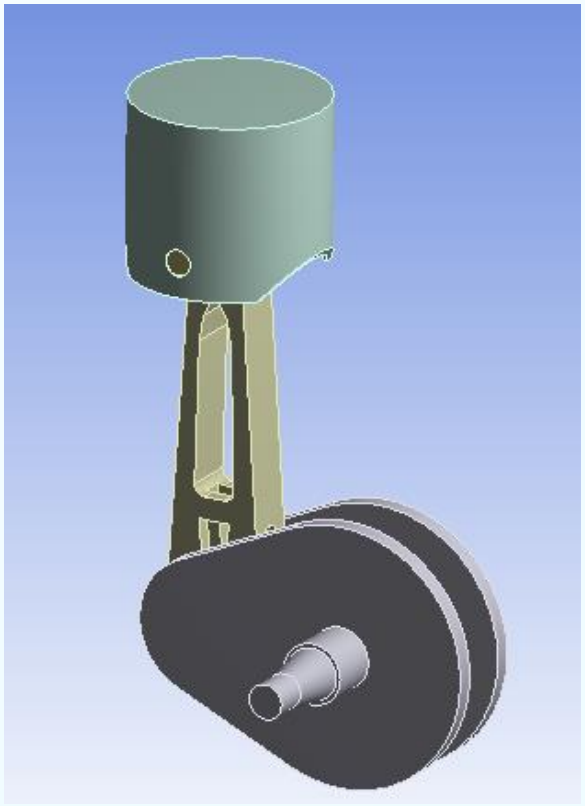
Topology Isosurface STL





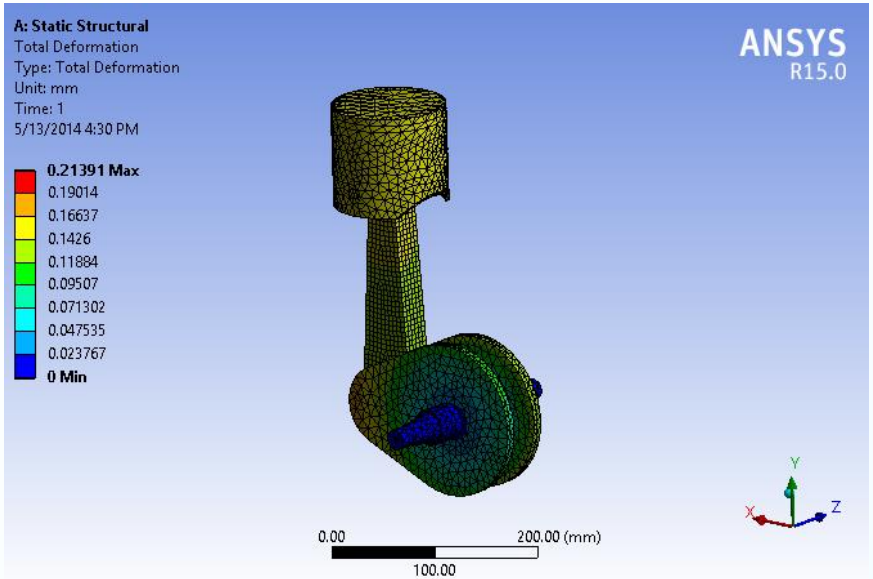
SpaceClaim Geometry Clean Up

Import Model back to ANSYS Mechanical

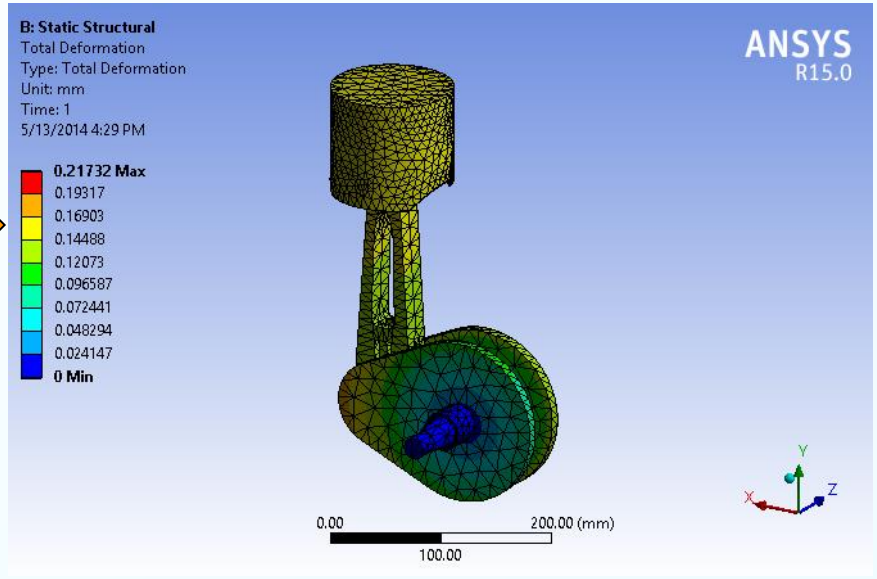


Validation Run in ANSYS

- **Deformation Result**



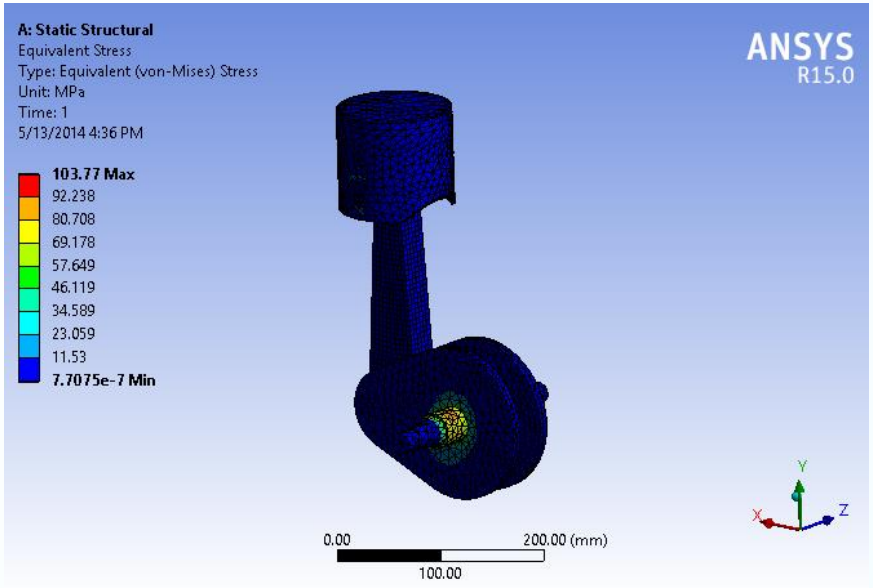
Initial



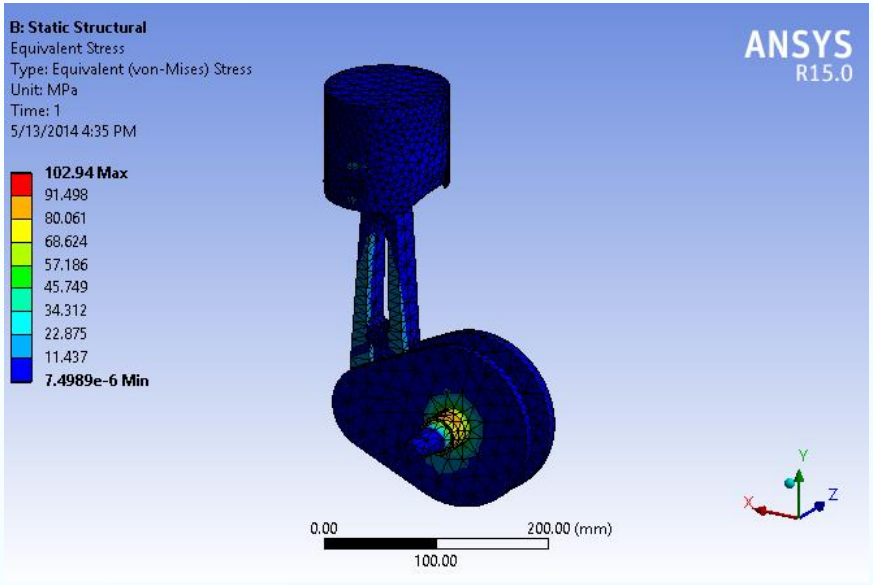
Optimized

Validation Run in ANSYS

- **Stress Result**



Initial



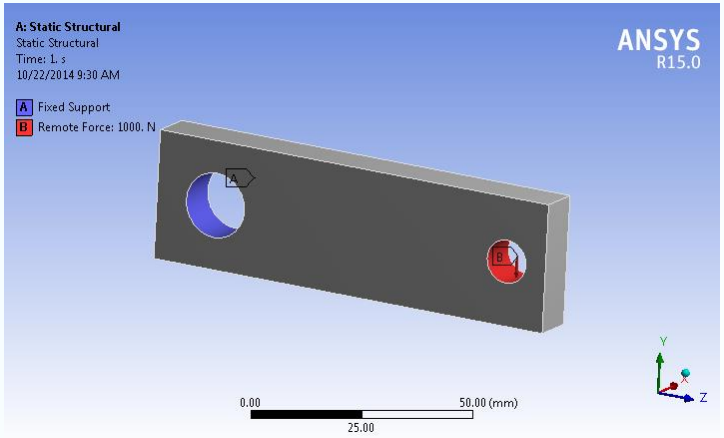
Optimized

Topology Optimization with Frozen Surface

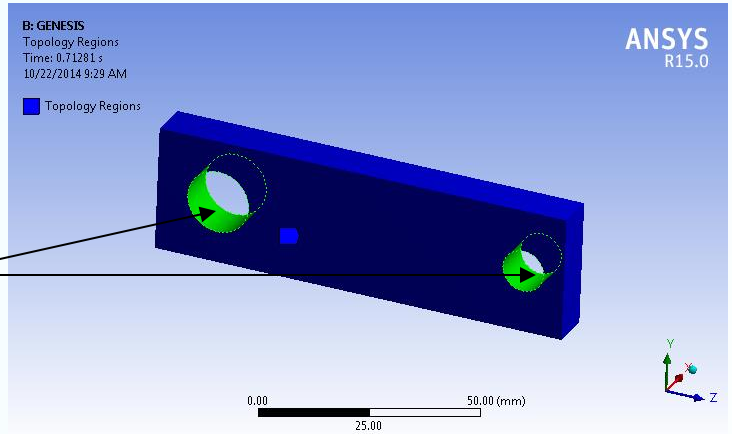
Problem Statement:

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

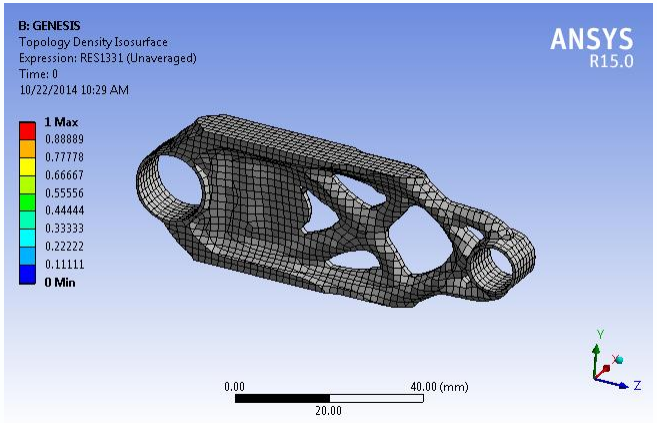


Loading and Boundary Conditions



Frozen Surface

Topology Region with Frozen Surface



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

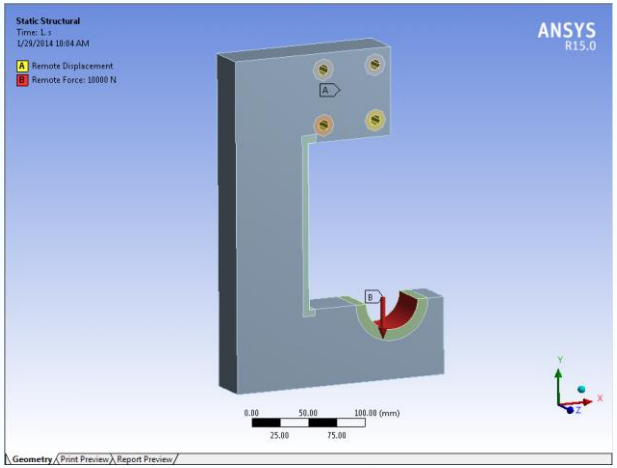
- **Constraint**

  Mass Fraction < 0.3

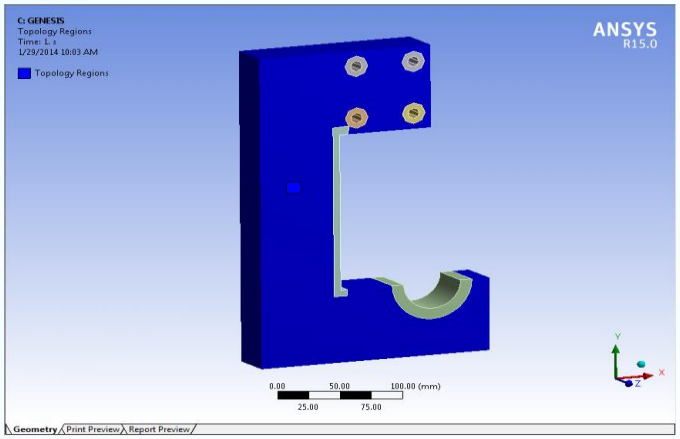
  Stress < 65 GPa

- **Designable Region**

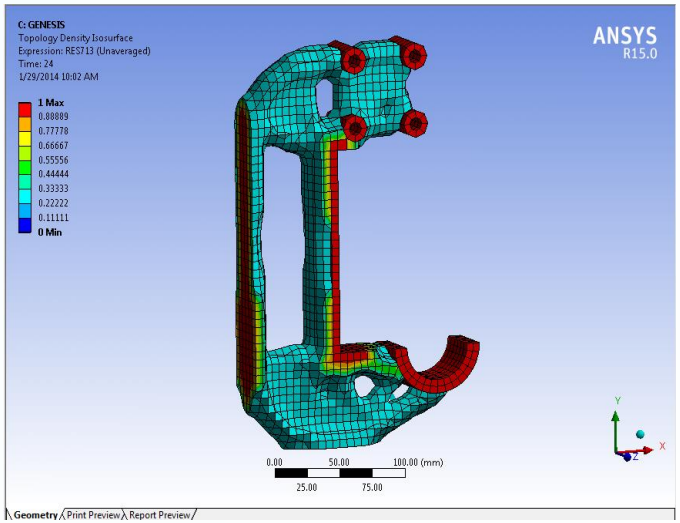
  Blue region



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**