Design Studio for GENESIS

EXAMPLES

STEP BY STEP INSTRUCTIONS

VERSION 11.0

October 2009

1767 SOUTH 8TH STREET SUITE 200
COLORADO SPRINGS, CO 80905
Phone: (719) 473-4611 Fax: (719) 473-4638
http://www.vrand.com
email: genesis.support@vrand.com

CHAPTER 1

Introduction

- Design Studio
- Conventions
- Get Familiar with Design Studio

1.1 Design Studio

Design Studio for Genesis is a design oriented pre- and post-processor for the Genesis Structural Analysis and Optimization Software.

At the heart of Design Studio data is a structural design model, which consists of a structural finite element model and design data that describes how the structure is allowed to change and defines the design goals and constraints.

While Design Studio does allow some analysis data to be created and edited, it is not intended to be a full-fledged analysis preprocessor. Rather, the focus of Design Studio is to allow design data to be easily created and edited and to allow easy visualization of optimization results.

The Design Studio user-interface is divided into three primary windows:

1. The Main Window
2. The Viewport Window
3. The Messages Window

The Main Window

The Main Window is positioned along the right-hand side of the screen. The Main Window serves as the primary window for listing and editing data inside of Design Studio.

The Viewport Window

The Design Studio Viewport Window provides a 3-D view of the finite element model. The viewpoint can be changed and the model can be zoomed to allow any part of the model to be observed. When the Viewport Window is active, depressing function keys F1-F4 and moving the mouse will result in changing the view as follows:

F1 - Pan the model. The model will move left/right and up/down on the screen to follow the mouse.

F2 - Zoom the model. Moving the mouse up will zoom closer to the model (the model will get bigger), and moving the mouse down will zoom away from the model (the model will get smaller).

F3 - Rotate the model. Moving the mouse up/down and left/right will rotate the model about the screen x and y axes, respectively. The point about which the model rotates can be changed with the Zoom button in the Viewport Window Toolbar

F4 - Rotate the model bounding box. This moves the model in the same way as F3, except that only a box showing the model bounds is drawn on the screen. This is useful for the case when model is very large, and the F3 rotate does not draw the model quickly enough to keep up with the mouse movement.

If you have a mouse with three buttons (or a scroll wheel that can act like a button), then you can press the middle mouse button in the Viewport Window and drag the mouse to change the view of the model. The type of view change is controlled by the radio buttons in the Middle Mouse Button Control.

Pan: The model will move left/right and up/down on the screen to follow the mouse.

Zoom: Moving the mouse up will zoom closer to the model (the model will get bigger), and moving the mouse down will zoom away from the model (the model will get smaller).

Rotate: Moving the mouse up/down and left/right will rotate the model about the screen x and y axes, respectively. The point about which the model rotates can be changed with the Zoom button in the Viewport Window Toolbar

In the lower left corner of the Viewport Window, there is an axis triad that shows the Viewport Window view's orientation by showing the basic coordinate system axes directions. Note that while this triad's coordinate axes are parallel to the basic coordinate system's, the view triad is always located at a fixed point on the screen, and does not show the location of the origin of the basic coordinate system.

Groups, grids, elements, and coordinate systems can be selected by clicking with the left mouse button in the viewport window.

The Messages Window

The Design Studio Messages Window fills the lower left portion of the screen. This window contains any Design Studio messages printed by commands. For example, the Identify Grids command in the Display Tab of the Main Window will print information about selected grids in the Messages Window.

The top left corner of the Messages Window also shows the current View Coordinate System.

The top center of the Messages Window shows the current Viewport Selection Mode, if any. If there is no current Viewport Selection Mode, then the top center gives a reminder of the meaning of the Viewport Window view function keys.

1.2 Conventions

These exercises are intended to familiarize the user with: using Genesis; the data requirements for creating a Genesis optimization problem; the methods to create that data using Design Studio for Genesis; and the methods to visualize results with Design Studio for Genesis.

The step-by-step instructions in these exercises follow these conventions:

Select means that the user should choose a menu item or an item from a list.

Push means that the user should click on an on-screen button.

Enter means that the user should type in data.

Pick means that the user should choose a node or element by positioning the cursor on it and pressing the left mouse button.

1.3 Get Familiar with Design Studio

Introduction

The purpose of this exercise is to get familiar with Design Studio. In this example you will learn how to show or hide groups and how to identify grids or elements. You will also study the Help menu which contains many details of how Design Studio works and summaries of the different analysis and optimization capabilities available in both Design Studio and Genesis. Additionally, this example will show you how to examine load cases, how to run Genesis and how to perform basic post processing.

Examine a Genesis Input Data

1. Edit file: `GFDSG001.dat` and examine the analysis data

   Genesis analysis data uses the same format as Nastran

Start and Load an Existing Genesis Input Data

2. Start Design Studio
3. From the main menu bar, select **File** → **Import** → **Input Data ...** (or Ctrl-I)

   Note that **File** → **Open** is reserved for opening a Design Studio database file (`*.dsg`).
   File → **Import** → **Input Data...** is for loading a Genesis (or Nastran) input file.
4. Go to the directory where the `GFDSG001.dat` is stored and select it
5. Push the **Open** button

Study the Main Window

6. Select the **Display** tab
7. Using the **Group Display Style** buttons at the bottom of the panel**,** change **Wire Frame** to another style (e.g., **Flat Shaded**)

   Try out all the different options and see how the model appears for each.
8. Push the **Show/Hide Groups** button
9. Turn groups on and off by selecting group from the **Show/Hide Groups** list or the screen

   An eyeglasses icon displays next to visible groups in the list
10. Push the **Show All** button to show all the groups
11. Push the **Up** button, to return to the main page of the **Display** tab

12. Push the **Identify Elements** button
13. Pick some elements from the screen

    Information about the chosen elements is printed in the Messages Window
14. Push the **Up** button
15. Push the **Identify Grids** button
16. Pick some grids from the screen

    Information about the chosen grids is printed in the Messages Window
17. Push the **Up** button
18. To clear the labels from the screen, right click in the Viewport Window and select **Clear** → **All**

Study the Design Studio Online Help

19. From the main menu bar, select **Help** → **Design Studio Help**
20. Navigate the help to get familiar with it
21. Double click the **Design Data** entry
22. Click the **Parametric Design Data** entry, browse this page for getting a brief summary of sizing, shape, topometry and topography optimization in Genesis.
23. Close the Help by clicking the **X** on the upper right corner of the Help window

Study the Viewport View Buttons

24. Select different views of the model by simply pushing different view buttons
25. In the Design Studio Viewport, push the button with the mouse icon, then select one of the three radio buttons: **Pan**, **Zoom** or **Rotate**. Go to the viewport and move the mouse holding the middle button to pan, zoom or rotate the model

Study the Genesis Menu

26. From the main menu bar, select **Genesis** → **Model Summary**

    This will bring up a dialog box. Study the summary data printed.
    You can export the summary data to any text editor by selecting it, copying it with Ctrl-C, and pasting it with Ctrl-V.

27. Push the **Close** button in the model summary window when finished
28. From the main menu bar, select **Genesis** → **Options...**

    This will bring up a dialog box. Study the options available. Consult the Genesis User's Manual Volumes 1 and 2 for details about the options.

29. Push the **Cancel** button in the Genesis Options window
30. From the main menu bar, select **Genesis** → **Single Analysis**

    This will run Genesis using the analysis data in loaded in Design Studio.
    A console window will appear showing Genesis module times.

31. **Close** the **Genesis Console Output** window when done

Check the Loadcases

The purpose of the following steps is to study the loadcases of this problem.

32. Select the **Analysis** tab
33. From the category chooser, select **Loadcases**
34. Select a loadcase from the list and observe the loading conditions displayed on the model
35. Push the **Modify Loadcase** button from the Edit Menu toolbar
36. Navigate the loadcase using the **Next>** and **<Back** buttons
37. Select the **Cancel** button to cancel editing the loadcase.

Output Requests

Now we will request for displacements and stresses for post processing:

38. Select the **Analysis** tab in the Main Window
39. Select **Loadcases** from the category chooser
40. Select all Loadcases.

    Click on the first loadcase in the list, then shift-click on the last loadcase to select all in-between

41. Push the **Modify Loadcase** button from the Edit Menu toolbar
42. Check the check box to the left of the **Displacement** label
43. Select **Post** and **All** options for Displacement
44. Check the check box to the left of the **Element Stress** label
45. Select **Post** and **All** options for Element Stress
46. Push the **Finish** button

    No changes are made to the loadcases until the trail is **Finish**ed

Run Genesis

47. From the main menu bar, select **Genesis** → **Single Analysis**
48. **Close** the **Genesis Console Output** window when done

Post Processing Analysis Results with Design Studio

49. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...** (or Ctrl-R). Select `GFDSG001_dsg00.pch`

    When you run Genesis from within Design Studio, it automatically exports a Genesis input file with `_dsg` added to the filename. This is to preserve any original file you may have imported. Genesis adds the `00` for the design cycle number. Design cycle 0 means the initial model. The extension `pch` is for punch-formatted post-processing files.
50. Push the **Open** button
51. Select the **Post** tab
52. Push the **Deform Mesh/Color Mesh** button
53. Select a stress results from the Color Mesh list

    The corresponding displacement result from the top list is automatically selected when a result from the bottom list is choosen.
54. Select **Normal-X at Z1** result from the pull down menu at the bottom of the form

    Try visualizing other stress components by selecting from the chooser.
55. Push the **Up** button when you are finished

Quit Design Studio

From the main menu bar, select **File** → **Quit**

56. Push the **Don’t Save** button

CHAPTER 2

List of Examples

- Overview
- List of Topology Examples
- List of Sizing Examples
- List of Topometry Optimization Examples
- List of Shape Optimization Examples
- List of Topography Optimization Examples
- List of Freeform Optimization Examples
- List of Composite Optimization Examples
- List of Frequency Response Optimization Examples
- List of Modal Correlation Examples
- List of Super Element Examples
- List of User Responses Examples
- List of Analysis - Meshing Examples
- List of Analysis - Solution Control Examples

2.1 Overview

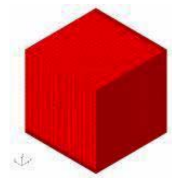
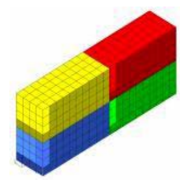
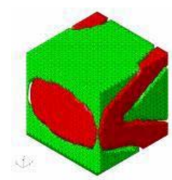
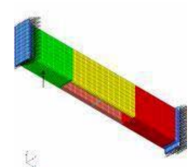
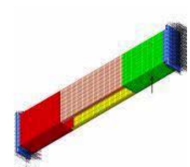
The purpose of this manual is to provide a variety of examples to assist the user in becoming familiar with the *GENESIS* design optimization capabilities and how Design Studio can be used to create the input data easily. These examples are provided with the *GENESIS* software so they may be executed for comparison.

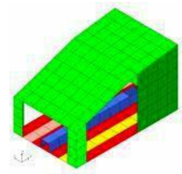
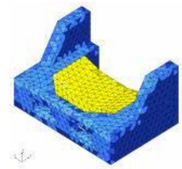
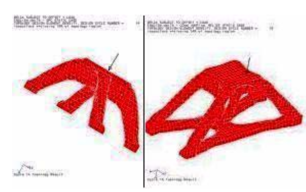
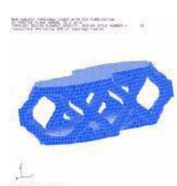
It should be remembered that, because of different machine precisions, the results may differ slightly from those presented here.

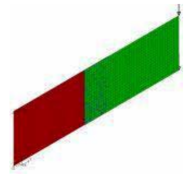
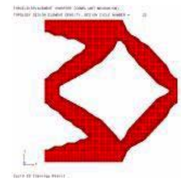
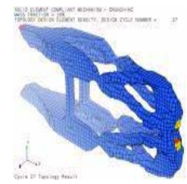
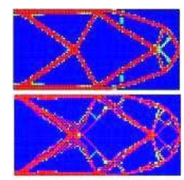
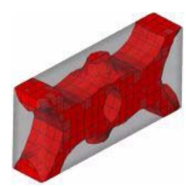
To assist the user in understanding the design capabilities of the package, a variety of examples are presented. Some of these examples correspond to examples explained in Volume IV.

2.2 List of Topology Examples

The following table lists the example that demonstrate topology optimization problems (note that the files starts with `TP`):

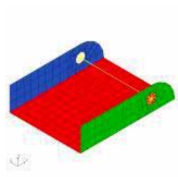
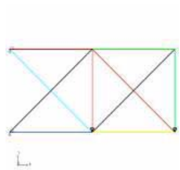
File Name	Problem Title	Special Features	Figure
`TPDSG001.dat`	Topology Optimization of a Support	• None: Classical optimization problem.	
`TPDSG002.dat`	Designing with Fabrication Constraints	• Imposing fabrication constraints on the topology design	
`TPDSG003.dat`	Creating Surface Meshes That Enclose Topology Results	• Create CAD meshes • Exporting surface mesh files using Design Studio for Genesis	
`TPDSG004.dat`	Using Multiple Designable Regions with Symmetry Constraints	• Setting up of symmetry constraints • Creating Topology optimization data for multiple designable regions	
`TPDSG005.dat`	Using Multiple Designable Regions with Symmetry and Fabrication Constraints	• Casting Constraints	

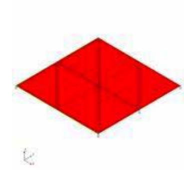
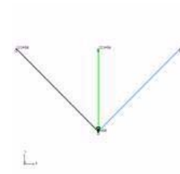
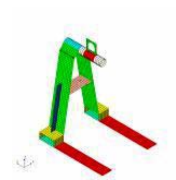
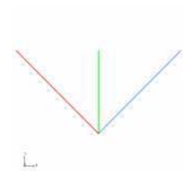
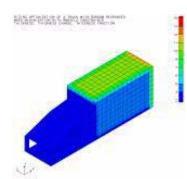
TPDSG006.dat	Torsional Frequency Maximization Using Ribs generated by AUTORIB and Mode Tracking	• AUTORIB option • Creating an updated input file containing optimal results	
TPDSG007.dat	Reinforcing a Solid Part Using Topology Optimization on a Design Studio Generated Skin Layer	• Creation of Skin layer using Design Studio for Genesis • Removing those elements of the skin that contribute less to the Stiffening of the Solid part	
TPDSG008.dat	Stiffness Maximization with Buckling Constraints	• Ease of imposing multiple constraints of the same nature	
TPDSG009.dat	Optimization of Reinforcement Member With Random Response	• Using random response in topology optimization • Antichecker board Filter • Control of Move Limits for the optimization process	
TPDSG010.dat	Differences in Topology Optimization Using Single Point Constraints and Inertial Relief	• Highlights the issue concerning the topology optimization using INERTIA RELIEF for static solutions	
TPDSG011.dat	Solid Block Subject to Torsional Loads with Stampable Sheet Constraints	• Imposing Stampable Sheet fabrication constraints • Stampable sheet Constraints with/without punch holes	

`TPDSG012.dat`	Optimal Bonding Location for Thin Sheets Subject to Shear Loads	• Using topology optimization to determine optimal bonding location	
`TPDSG013.dat`	Topology Optimization of a Compliant Mechanism	• Designing a 2-D compliant mechanism using Topology optimization	
`TPDSG014.dat`	Topology Optimization of a Solid Element Compliant Mechanism	• Designing a 3-D compliant mechanism using Topology optimization	
`TPDSG015.dat`	Study the Effects of Anti-Checkerboard Filter	• To study the effects of Anti-Checkerboard Filter on the optimization	
`TPDSG016.dat`	Creating and Using Synthetic Responses (TRESP2)	• To learn how to create and use synthetic responses (TRESP2)	

2.3 List of Sizing Examples

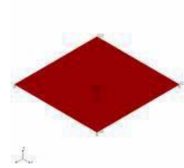
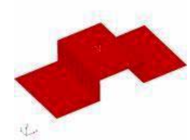
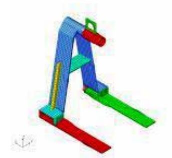
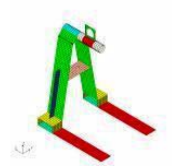
The following table lists the example that demonstrate sizing optimization (note that the files starts with SZ):

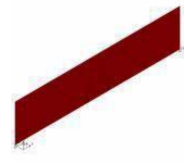
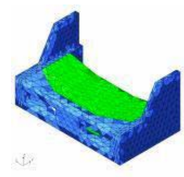
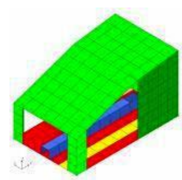
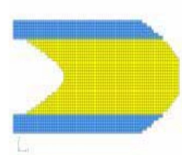
File Name	Problem Title	Special Features	Figure
SZDSG001.dat	Simple Sizing Optimization Setup	• None: Simple sizing optimization problem	
SZDSG002.dat	Using One Variable to Design Multiple Regions	• Designing multiple regions with the same variable	
SZDSG003.dat	Quick Sizing - Designing ROD elements	• Defining optimization data for multiple designable properties at once	
SZDSG004.dat	Defining Non-linear Relationship between Property and Design Variable	• Using equations to define relations between design variables and designable properties	
SZDSG005.dat	Using the Design Element Library	• Designing cross-sectional dimensions using the design element library	

`SZDSG006.dat`	Design of a Square Plate with Volume Heat Generation and Enforced Temperature	• Designing using heat transfer response	
`SZDSG007.dat`	Design Using Multidisciplinary Load Cases - Three Rod Truss	• Optimization using responses from multidisciplinary loadcases	
`SZDSG008.dat`	Quick Sizing Shell Elements - Design of a Pallet Lifter	• Defining optimization data for multiple designable properties at once	
`SZDSG009.dat`	Discrete Optimization - Three Rod Truss	• Using discrete values for the design variables	
`SZDSG010.dat`	Design Based on Random Response	• Using random responses in the optimization problem	

2.4 List of Topometry Optimization Examples

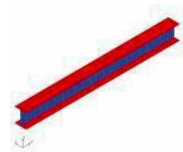
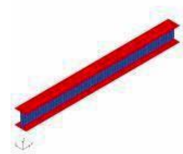
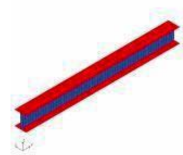
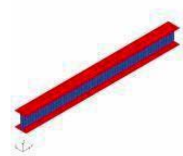
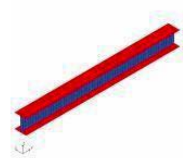
The following table lists the example that demonstrate topometry optimization problems (note that the files starts with `TM`):

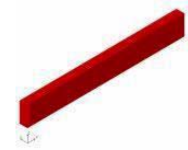
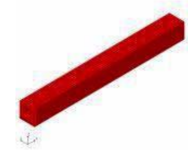
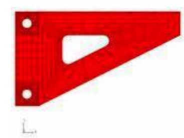
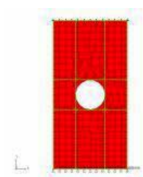
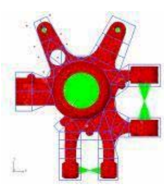
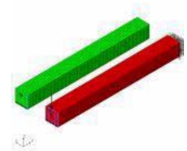
File Name	Problem Title	Special Features	Figure
`TMDSG001.dat`	Topometry Optimization of a Flat Plate	• Simple Topometry optimization problem • Plotting Thickness Distributions	
`TMDSG002.dat`	Topometry Optimization of a Hat Structure	• Simple Topometry optimization problem • Plotting Thickness Distributions	
`TMDSG003.dat`	Topometry Optimization of a Pallet Lifter Using Symmetries	• Creation of a Topometry optimization data with multiple designable regions and symmetries	
`TMDSG004.dat`	Coarse Topometry Optimization of a Pallet Lifter	• Using of Coarsening option in topometry definition	
`TMDSG005.dat`	Creating a CAD Mesh that Represents Results of a Topometry Optimized Shell Structure	• Creating CAD surface mesh to represent topometry results	

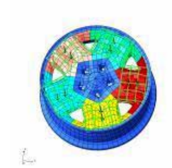
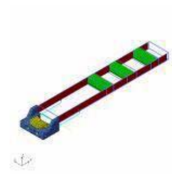
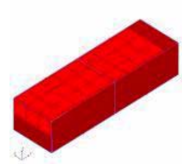
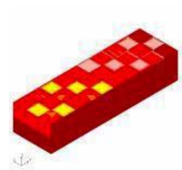
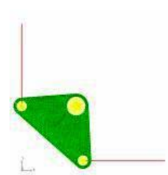
`TMDSG006.dat`	Creating a Finite Element Mesh that Represents Results of a Topometry Optimized Structure using Re-grouping	• Interpret and create an FE mesh of the topology result • Using the re-grouping capabilities exhibited by Design Studio for Genesis	
`TMDSG007.dat`	Reinforcing a Solid Part on a Design Studio Generated Composite Skin Layer	• Create a composite skin on a solid part using Design Studio • Use topometry to find reinforcement locations on the skin • Trade off study between Mass and Frequency gains	
`TMDSG008.dat`	Topometry Optimization of Welds	• Use topometry to determine optimal location of welds from a group of candidate locations • Creation of Smart Assemblies	
`TMDSG009.dat`	Mode Shape Optimization	• Using Mode Shape as Objectives • Saving animation results as AVI files	
`TMDSG010.dat`	Optimization of Launch Vehicle to Maximize Frequency	• Frequency Maximization of launch vehicle	
`TMDSG011.dat`	Re-grouping based on Topometry Optimization Results	• Move elements into different groups based on topometry results	

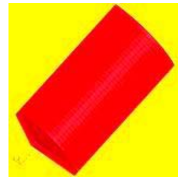
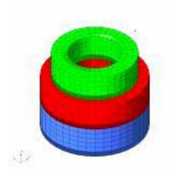
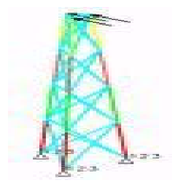
2.5 List of Shape Optimization Examples

The following table lists the example that demonstrate shape optimization (note that the files starts with SH):

File Name	Problem Title	Special Features	Figure
`SHDSG001.dat`	Design using 1-D (LINE) Domains	• Utilize 1-D domains to design a structure	
`SHDSG002.dat`	Design using 2-D (QUAD) Domain - Uniform Variation	• Utilize 2-D domains to design a structure • Obtain uniform variation along an edge of the domain.	
`SHDSG003.dat`	Design using 2-D (QUAD) Domain - Linear Variation	• Obtain linear variation along edges of the domain • Use of Geometric (DRESPG) constraints	
`SHDSG004.dat`	Design using 2-D (QUAD) Domain - Quadratic Variation	• Use of Mid-side perturbations to achieve quadratic variation along the edge of the domain	
`SHDSG005.dat`	Design using Multiple 2-D (QUAD) Domains	• End perturbations are applied on intersection edges of multiple domains to achieve linear variation along the domains associated with that edge.	

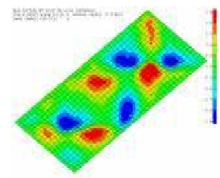
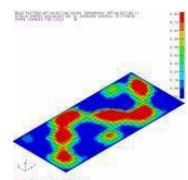
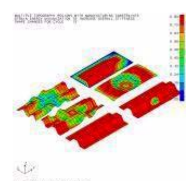
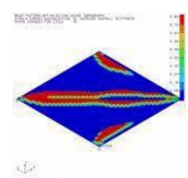
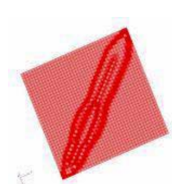
`SHDSG006.dat`	Design using 3-D (HEXA) Domains	• Create a domain by dragging in the viewport • Adjusting the size of the dragged domain • move the created domain to an appropriate location	
`SHDSG007.dat`	Design using 3-D (HEXA) and 2-D (QUAD) Domains	• Multiple domains are created by splitting a large domain	
`SHDSG008.dat`	Design a Feature - Using 2-D (TRIA & QUAD) Domains	• Perturbation are applied to design a feature with the structure.	
`SHDSG009.dat`	Design a Feature - Using 2-D (QUAD) Domains	• Designing the feature of a structure	
`SHDSG010.dat`	Design of a Steering Knuckle - Review of Shape Domains	• Designing several features of a structure	
`SHDSG011.dat`	Using Different Domains to Perform Similar Design	• Using different types of domains to achieve the same design • Using only domains where perturbation should act. • Multiple Objectives	

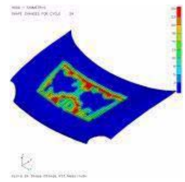
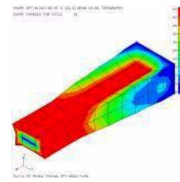
`SHDSG012.dat`	Making Copies of Existing Domains - Wheel Example	• Creating copies of existing domains • Perturbations in all domains are controlled by the same design variable to maintain symmetry	
`SHDSG013.dat`	Design Location of Member - Using Synthetic Responses	• Design location of a feature in the structure • Synthetic responses to define objectives as well as some of the constraints.	
`SHDSG014.dat`	Design of a Solid Cantilever Beam using DOMAINs	• Simple Shape optimization problem	
`SHDSG015.dat`	Design of a Solid Cantilever Beam using Natural Perturbation Vectors	• Using natural perturbation vectors for shape design • Using deformed shape to create perturbations	
`SHDSG016.dat`	Using Hexa Domains	• Simple Shape optimization problem • Create simple FE mesh using Design Studio	
`SHDSG017.dat`	Using Multiple Quad Domains	• Creating multiple domains at the same time	

SHDSG018.dat	Using Axisymmetric Bar Domains	• Using Axisymmetric domains	
SHDSG019.dat	Using Axisymmetric Quad and Axisymmetric Bar Domains	• Using Axisymmetric domains	
SHDSG020.dat	Combining Shape and Sizing Optimization I	• Simple shape optimization problem coupled with sizing optimization.	
SHDSG021.dat	Combining Shape and Sizing Optimization II - Use of Domains and Grid Perturbations	• Shape optimization problem coupled with sizing optimization. • Creating Grid (Raw) Perturbations	

2.6 List of Topography Optimization Examples

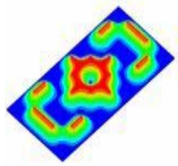
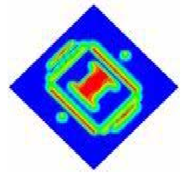
The following table lists the example that demonstrate topography optimization (note that the files starts with `TG`):

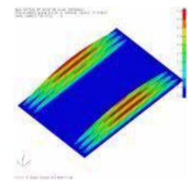
File Name	Problem Title	Special Features	Figure
`TGDSG001.dat`	Simple Topography Optimization Setup	• Simple Topography setup.	
`TGDSG002.dat`	Allowing Grid Movement in One Direction	• Controlling the design variables to allow bead formation in one direction	
`TGDSG003.dat`	Design with Manufacturing Constraints - Extrusion, Symmetry, Cyclic, Axisymmetric Constraints	• Enforcing different types of manufacturing constraints	
`TGDSG004.dat`	Bead Fraction - Constraining Grid Movement	• Enforcing Bead Fraction constraint to restrict the movement of grids	
`TGDSG005.dat`	Modify FE mesh based on Topography Optimization Result	• Update the FE mesh based on the topography results	

`TGDSG006.dat`	Design With and Without Bead Fraction Constraint	• Comparing design with and without Bead Fraction constraint	
`TGDSG007.dat`	Designing Shape of Solids with Topography	• Methodology to design the shape of a solid using topography optimization	

2.7 List of Freeform Optimization Examples

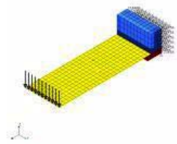
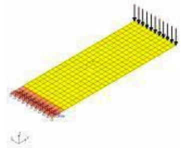
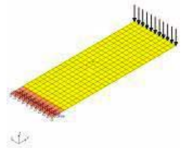
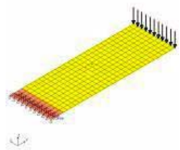
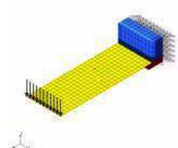
The following table lists the example that demonstrate freeform optimization (note that the files starts with FF):

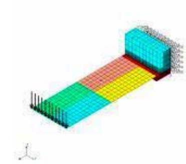
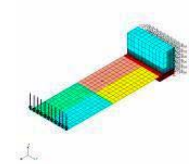
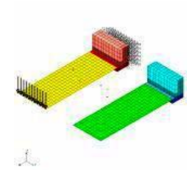
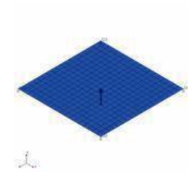
File Name	Problem Title	Special Features	Figure
`FFDSG001.dat`	On Raw Perturbations (DVGRID) - Mirror Symmetries	• Simple Freeform setup using raw perturbations • Use of Mirror symmetries • Use of Grid Fraction constraint	
`FFDSG002.dat`	On Raw Perturbations (DVGRID) - Manufacturing Constraints and Coarsening	• Using different types of Manufacturing constraints • Use of Grid Fraction constraint • Using coarsening to reduce variability	
`FFDSG003.dat`	On Domain Perturbations (DVGRIDC) - Axisymmetry and Grid Fraction Constraints	• Simple Freeform setup using domain perturbations • Use of Axisymmetry constraint • Use of Grid Fraction constraints	
`FFDSG004.dat`	On Domain Perturbations (DVGRIDC) - Mirror Symmetry	• Use of Mirror symmetries • Use of Grid Fraction constraints	
`FFDSG005.dat`	Design of a Connecting Rod - Using Domain Perturbations	• Using domain perturbations with and without freeform	

`FFDSG006.dat`	On Grid Perturbations Generated Using Domains	• Freefrom on grid perturbations generated by DOMAINs • Using Extrusion constraints	

2.8 List of Composite Optimization Examples

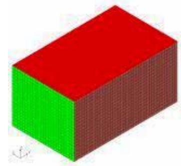
The following table lists the example that demonstrate optimization of composites(note that the files starts with `CM`):

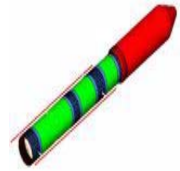
File Name	Problem Title	Special Features	Figure
`CMDSG001.dat`	Layer Thickness Optimization of a Cantilever Plate	• Basic steps to create and solve a simple sizing optimization problem with composites.	
`CMDSG002.dat`	Composite Optimization of a Cantilever Plate Considering the Location of the Reference Plane	• Design the location of the reference plane of the PCOMP	
`CMDSG003.dat`	Layer Angle Optimization of a Cantilever Plate	• Basic steps to create and solve a simple sizing optimization problem with composites.	
`CMDSG004.dat`	Shape Optimization of Composite Cantilever Plate using Shape Domains and Morphing Sets	• Basic steps to create and solve a shape optimization problem with composites.	
`CMDSG005.dat`	Topometry Optimization of a Composite Plate using Discrete Angles with/without Coarsening	• Topometry optimization of composites • Using Coarsening to form groups	

`CMDSG006.dat`	Topometry Optimization of a Composite Plate using Linked Groups	• Linking different composite group properties to achieve symmetry • Use of the REPEAT option.	
`CMDSG007.dat`	Topometry Optimization of a Composite Plate without splitting the Angles	• Linking different composite group properties to achieve symmetry • Topometry design of the layer angles without splitting	
`CMDSG008.dat`	Aligning the Element and Material Coordinate Systems of the Mesh of a Composite Plate	• Studying the alignment of the element and material coordinate systems for the PCOMP	
`CMDSG009.dat`	Topometry using Composite Failure Equations	• Using user-defined failure equations for PCOMP • Optimization based on the user-defined failure equation	

2.9 List of Frequency Response Optimization Examples

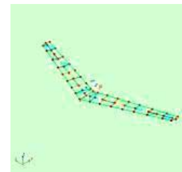
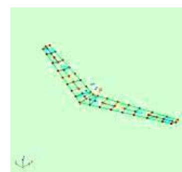
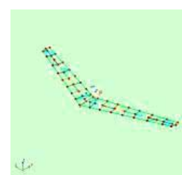
The following table lists the example that demonstrate optimization based on frequency response analysis(note that the files starts with FR):

File Name	Problem Title	Special Features	Figure
`FRDSG001.dat`	Frequency Response Optimization using Beta Method	• Using Beta method to minimize the maximum of a frequency response	
`FRDSG002.dat`	Driveline Vibrational and Modal Frequency Response Analysis	• Frequency response analysis and post-processing of the analysis results	
`FRDSG003.dat`	Driveline design using the beta method I	• Using Beta method to minimize the maximum of a frequency response • Use of one loadcase for the entire loading frequency range	
`FRDSG004.dat`	Driveline design using the beta method IIa	• Using Beta method to minimize the maximum of a frequency response • Use of multiple loadcases depending on the peaks for loading frequency range	
`FRDSG005.dat`	Driveline design using the beta method IIb	• Using Beta method to minimize the maximum of a frequency response • Use of multiple loadcases depending on the peaks for loading frequency range	

`FRDSG006.dat`	Payload Integration Design of a Launch Vehicle	• Design based on dynamic accelerations using beta method	

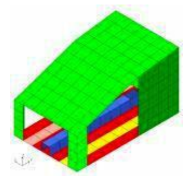
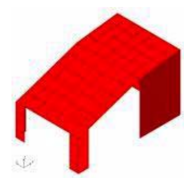
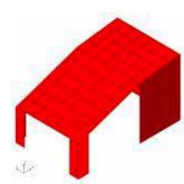
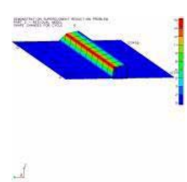
2.10 List of Modal Correlation Examples

The following table lists the example that demonstrate how optimization can be used to correlate experimental data with finite-element results(note that the files starts with `MC`):

File Name	Problem Title	Special Features	Figure
`MCDSG001.dat`	Compare Test Modes to Analytical Modes	• Perform a modal analysis by creating a Modal analysis loadcase • Import mode shapes obtained from Ground Vibration Test (GVT) set up • Compare FEM and GVT mode shapes	
`MCDSG002.dat`	Test Data Correlation with FEM Using Sizing Optimization	• Use optimization to match the mode shapes of the FE model with GVT mode shapes	
`MCDSG003.dat`	Add the Exact Target Frequencies to the Design Objective	• Use optimization to match the mode shapes of the FE model with GVT mode shapes • Also optimize to match the frequency values	
`MCDSG004.dat`	Adding Design Constraints to Tune the Airfoil Spar Stiffness Profile	• Use optimization to match the mode shapes and frequencies of the FE model with GVT mode shapes • Add additional constraints for tuning the design	

2.11 List of Super Element Examples

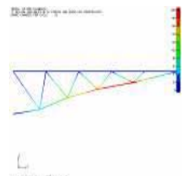
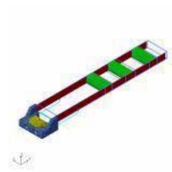
The following table lists the example that demonstrate the use of superelements(note that the files starts with SE):

File Name	Problem Title	Special Features	Figure
`SEDSG001.dat`	Truck Cabin - Full System Design	• Baseline full system design	
`SEDSG002.dat`	Truck Cabin - Superelement Simulation using Component Mode Synthesis	• Using Component Mode synthesis for superelement reduction	
`SEDSG003.dat`	Truck Cabin - Design Using Imported External Superelement	• Using imported external superelement in the design optimization problem	
`SEDSG004.dat`	Truck Cabin - Superelement REDUCE Mode	• Using static reduction for superelement creation • Use of REDUCE executive control statement	
`SEDSG005.dat`	Truck Cabin - Design Using Statically Reduced Superelement	• Using statically reduced superelement information in design	

SEDSG006.dat	Driveline - Superelement reduction using the Craig-Bampton modes	• Using Craig-Bampton modes for Superelement reduction	

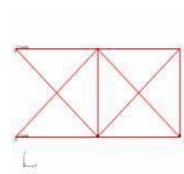
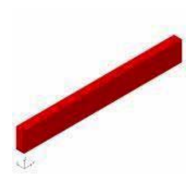
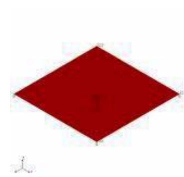
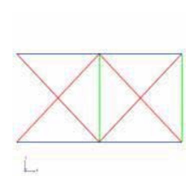
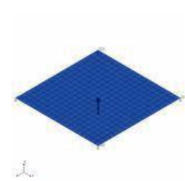
2.12 List of User Responses Examples

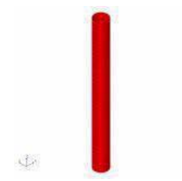
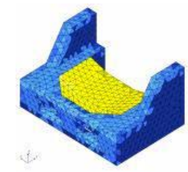
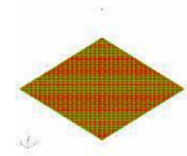
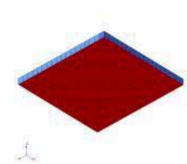
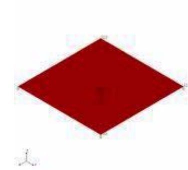
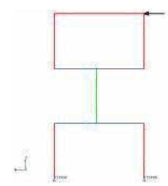
The following table lists the example that demonstrate how to use usre defined responses in optimization (note that the files starts with UR):

File Name	Problem Title	Special Features	Figure
`URDSG001.dat`	Incorporating User-defined Subroutines (DRESP3)	• Using user subroutines to calculate responses and optimize based on these response	
`URDSG002.dat`	Using Synthetic Responses (DRESP2)	• Synthetic responses (DRESP2) to define objectives as well as some of the constraints.	

2.13 List of Analysis - Meshing Examples

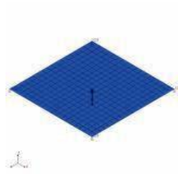
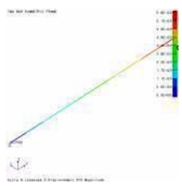
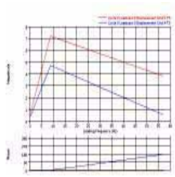
The following table lists the example that demonstrate meshing and mesh modification capabiltiies of design studio (note that the files starts with AM):

File Name	Problem Title	Special Features	Figure
`AMDSG001.dat`	Ten Rod Truss Model	• Create an FEM model with 1-D elements • Analyze the FEM model for a static load case.	
`AMDSG002.dat`	Cantilever Beam with Solid Elements	• Create a FE mesh by generating grids and elements at the same time.	
`AMDSG003.dat`	Simply Supported Plate with Shell Elements	• Create a FEM mesh with Shell elements	
`AMDSG004.dat`	Changing Element Groups	• Moving elements from one group to the other	
`AMDSG005.dat`	Composite Plate	• Create a PCOMP group • Moving elements from a PSHELL to a PCOMP	

AMDSG006.dat	Hollow Tube with Shell Elements	• Generate a FE mesh by copying elements • Perform Modal Analysis using Eigen Methods.	
AMDSG007.dat	Generating a Shell Skin on a Solid Part	• Generate a Shell skin over a Solid Part	
AMDSG008.dat	Automatic Creation of Ribs - Bar Elements	• Create 1-D ribs using AUTORIB	
AMDSG009.dat	Automatic Creation of Ribs- QUAD Elements	• Create 2-D ribs using AUTORIB	
AMDSG010.dat	Mesh Refinement	• Refining an existing mesh using Design Studio	
AMDSG011.dat	Three Story Facade with Panel	• Create Rigid (RBE) Elements	

2.14 List of Analysis - Solution Control Examples

The following table lists the example that demonstrate setting up the analysis using design studio(note that the files starts with AS):

File Name	Problem Title	Special Features	Figure
ASDSG001.dat	Buckling Analysis of a Tripod	• Performing a buckling analysis	
ASDSG002.dat	Using Composite Failure Equations	• Defining composite failure based on user defined equations	
ASDSG003.dat	Two Bar Symmetric Frame	• Using multi-point constraints (MPC) to take advantage of symmetric in structures	
ASDSG004.dat	Dynamic Analysis of Cantilever Beam	• Performing a frequency response analysis • Create a Frequency Response plot in Design Studio	

CHAPTER 3

Topology Optimization Examples

- **Topology Optimization of a Support**
- **Designing with Fabrication Constraints**
- **Creating Surface Meshes That Enclose Topology Results**
- **Using Multiple Designable Regions with Symmetry Constraints**
- **Using Multiple Designable Regions with Symmetry and Fabrication Constraints**
- **Torsional Frequency Maximization Using Ribs generated by AUTORIB and Mode Tracking**
- **Reinforcing a Solid Part Using Topology Optimization on a Design Studio Generated Skin Layer**
- **Stiffness Maximization with Buckling Constraints**
- **Optimization of Reinforcement Member With Random Response**

- **Differences in Topology Optimization Using Single Point Constraints and Inertial Relief**
- **Solid Block Subject to Torsional Loads with Stampable Sheet Constraints**
- **Optimal Bonding Location for Thin Sheets Subject to Shear Loads**
- **Topology Optimization of a Compliant Mechanism**
- **Topology Optimization of a Solid Element Compliant Mechanism**
- **Study the Effects of Anti-Checkerboard Filter**
- **Creating and Using Synthetic Responses (TRESP2)**

3.1 Topology Optimization of a Support

Introduction

The purpose of this example is to introduce the basic steps to create and solve a simple topology optimization problem. This example will first examine the static loadcase. Then, this example will show how to make a topology region designable, how to create an objective function, and how to create a constraint. Finally, this example will show how to display topology results using element densities, isodensity surfaces and animations.

Problem Statement:

To obtain a stiff structure, the objective function of the problem is to minimize the strain energy of the structure subject to a mass fraction $\leq$ 0.40. The designable region comprises of all the elements in the structural model. The structure is simply supported at 4 corner grids at one end with a point load applied at the opposite end.

Example ID

`TPDSG001`

Files Used in This Problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`TPDSG001.dat`	`Input data`	Provided: This file is the GENESIS input data that is read into Design Studio
`TPDSG001_ref.dat`	`Input data`	Provided: Reference file ready to be optimized
`TPDSG001_dsgDENSxx.pch`	`DSG file`	Created: Punch file containing the topology optimization results for the xx design cycle
`TPDSG001_dsgTSURFxx.dat`	`DSG file`	Created: Bulk data file of the topology surface (shells) for the xx design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG001.dat`

View Model Cutaway

3. Select the **Display** tab
4. For the **Model Cutaway**, select the **Cut V.C.S X axis, hide - side** option

   Observe that the block is hollow inside
5. Use the Slider for moving the cutting plane along the x-axis
6. For the **Model Cutaway**, select the **None** option to display the entire model

Examine the Static Loadcase

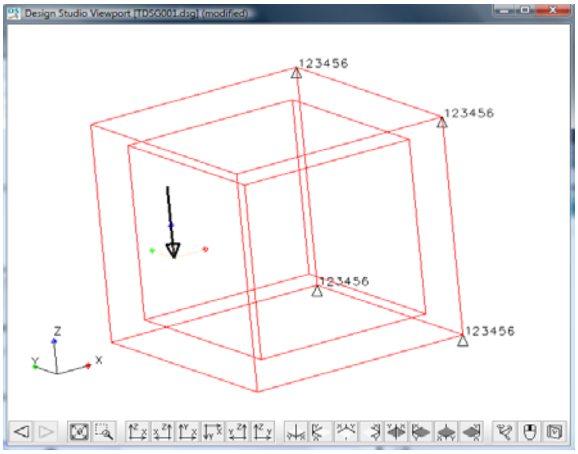
7. Select the **Analysis** tab
8. From the category chooser, select **Loadcases**
9. Select `Point_Load_Loadcase`

   Observe the applied static force and the boundary conditions in the main viewport.
10. Push the **Modify Loadcase** button from the Edit Menu toolbar
11. Push the **Next>** button

    Observe the SPC that has been selected.
12. Push the **Next>** button

    Observe the Load Set that has been selected.
13. Push the **Cancel** button
14. Select the **Display** tab
15. In the Group Display Style section at the bottom of the viewport push the **Feature Lines** Icon

    The applied load and boundary conditions on the hollow cube should look like the picture below.



Clearing the Selection

16. Right click the Viewport, Select **Clear**, **All**
17. In the Group Display Style section at the bottom of the viewport push the **Wire Frame** Icon

Create the Designable Region

18. Select the **Topology** tab
19. From the category chooser, select **Topology Regions**
20. Select the existing PSOLID
21. Push the **Modify Topology Design** button
22. Use `0.4` for the initial mass fraction
23. Push the **Finish** button to complete the design data for the topology region

    Verify that there is the hammer icon next to the PSOLID 4.
    The hammer icon indicates that the group is being designed.

Clearing the Selection

24. Right click the Viewport, Select **Clear**, **All**

Defining the Design Objective

25. From the category chooser, select **Topology Objectives**
26. Push the **New Topology Objective** button in the Edit Menu toolbar
27. Accept the default, **Strain Energy** response type. Accept the **Min** Objective Definition switch
28. Push **Next>**
29. Select the existing loadcase
30. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Defining the Design Constraints

31. From the category chooser, select **Topology Constraints**
32. Push the **New Topology Constraint** button in the Edit Menu toolbar

33. Enter Name `Whole_Structure`
34. Accept default of the **Mass Fraction** response
35. Accept default of **All Designed Groups** option of Mass Fraction
36. Enter `0.4` for the **Upper Bound**

    This constraint will cause Genesis to try to use only 40% of the material.
37. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Optimize the Structure Using Genesis

38. From the main menu bar, select **Genesis** → **Optimize**

    As with the Single Analysis run, a Genesis console window will appear.
    Additionally, two design history charts will display the objective and the maximum constraint violation versus each design cycle. These charts will update as the Genesis run progresses.
39. Study the **Design History** charts; when done, push the **Close** button
40. Study the **Genesis Console Output** window

Import the Post-Processing Files (Density Results)

41. From the **Genesis Console Output** window, select the **Import Post...** button
42. Using the **Ctrl**/**Shift** button, select all the post processing files to be imported

    File name: `TPDSG001_dsgDENSxx.pch` file where xx is the design cycle number is the file containing the Density results
    File name: `TPDSG001_dsgxx.pch` file where xx is the design cycle number is the file containing the analysis results
43. Push the **Import** button
44. From the **Genesis Console Output** window, select the **Close** button

Post-Processing the Results (Density Results)

45. Select the **Post** tab
46. Push the **Deform Mesh/Color Mesh** button
47. Select a Topology Result for any design cycle

    Study the results.
48. Select a Topology Result for the last design cycle

    Study the results.

49. Push the **Options...** button. Slide the lower cutoff slidebar to mask out element with low density values
50. Push the **Close** button

Post-Processing the Results (Density & Displacement Results)

51. Push the **Oscillate** button
52. Select a Displacement Result from the **Deform Mesh** list for the last design cycle

    Study the results.

53. Select **Topology** Result for a different design cycle

    Note that as you change design cycles for topology, the Displacement results change, as well.

54. Push the **Filled Contours** radio button
55. Select a Displacement Result from the **Color Mesh** list for any design cycle
56. Push the **Options...** button. Slide the lower cutoff slidebar to mask out element with low displacement values
57. Push the **Close** button

Finding the Top 10 Displacements:

58. Compare the maximum displacement values of the first and the last design cycle

    Hint: Right click the viewport and select **List Top 10** to see the 10 highest displacements.

59. Push the **Up** button

Post-Processing the Results (Animate results)

60. Select the **Post** tab
61. Push the **Animation** button
62. Change the **Color Result Type** chooser to **Topology Result**
63. Select the **Isosurface** radio button
64. Push **Next>**
65. Select all the results you want to animate
66. Push **Next>**
67. Push the **Options...** button; Slide the **Lower cutoff** slide bar to mask out element with low density values
68. Select the checkbox for **Show Topology Region** to view the initial design space

69. Push the **Close** button
70. Select the **Shaded Feature** icon in the bottom of the frame
71. Select the **Flat Shaded** icon in the bottom of the frame
72. Push the **Up** button

Post-Processing the Results (Density Isosurfaces)

73. Select the **Post** tab
74. Push the **Density Isosurface** button
75. Select a Topology Result for the last design cycle
76. Push the **Options...** button
77. Select the checkbox for **Show Topology Region** to view the initial design space
78. Push the **Close** button

Post-Processing the Results (Density Isosurfaces & Displacements)

79. Select the Displacement for the last design cycle
80. Push the **Oscillate** button
81. Push the **Up** button

Quit Design Studio

82. From the main menu bar, select **File** → **Quit**
83. Push the **Don't Save** button

3.2 Designing with Fabrication Constraints

Introduction

The purpose of this exercise is to get familiar with the creation of topology optimization data and impose fabrication constraints on the final design. This problem also goes through the steps on how to post process topology optimization results.You will also learn how to export a coarsened topology isodensity surface and to import isodensity surface (TSURF) files.

The following optimization problem will be created and solved:

Minimize Strain Energy

Subject to:

Mass Fraction $\leq$ 0.30

The structure should be castable in the Z direction

Designable region:

All solid elements in the structure

Note:

Minimizing strain energy is a way to get a stiff structure and is a very common objective function used in topology problems. The mass fraction of 0.3 will make Genesis select the most important 30% of the material.

Example ID

TPDSG002

Files Used in This Problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
TPDSG002.dat	Input data	Provided: This file is the GENESIS input data that is read into Design Studio
TPDSG002_dsgDENSxx.pch	DSG file	Created: Punch file containing the topology optimization results for the xx design cycle

TPDSG002_dsgTSURFxx.dat	DSG file	Created: Bulk data file of the topology surface (shells) for the xx design cycle

Start Design Studio

1. Start Design Studio
2. Load the Genesis data file: `TPDSG002.dat`

Create a Designable Region

3. Select the **Topology** tab
4. From the category chooser, select **Topology Regions**
5. Select all the existing PSOLID property groups

   Use the **Shift** or **Ctrl** keys or from the main menu bar, select **Edit** → **Select All**
6. Push the **Modify Topology Design** button
7. Accept the default of `0.3` for the initial mass fraction
8. Push the **Next>** button
9. Push the **Change** button to change the coordinate system
10. Select `Coord_11`
11. Push the **Next>** button
12. For Constraint 1: Select **FGZ**
13. For Minimum Size enter `1.0`
14. For Spread Fraction enter `0.50`
15. Push the **Finish** button to complete the design data for the topology region

    Verify that there is the hammer icon next to each PSOLID label.
    The hammer icon indicates that the group is being designed.

Defining Design Objective

16. From the category chooser, select **Topology Objectives**
17. Push the **New Topology Objective** button in the Edit Menu toolbar
18. Accept the default, **Strain Energy** response type. Accept the **Min** Objective Definition switch
19. Push the **Next>** button
20. Select the 3 existing loadcases

21. Push the **Finish** button

    Verify that now there are three responses in the objectives list

Defining the Design Constraints

22. From the category chooser, select **Topology Constraints**
23. Push the **New Topology Constraint** button in the Edit Menu toolbar
24. Select the **Mass Fraction** response
25. Accept the **All Designed Groups** option of Mass Fraction
26. Enter `0.3` for the **Upper Bound**
27. Push the **Finish** button

    Verify that now there is 1 response in the constraint list

Optimize the Structure Using Genesis

28. From the main menu bar, select **Genesis** → **Optimize**

    As with the Single Analysis run, a Genesis console window will appear. Additionally, two Design History charts will display the objective and maximum constraint violation versus design cycle. These charts will update as the Genesis run progresses.

29. Study the **Design History** charts, when done push the Close button
30. Study the **Genesis Console Output**, when done push the Close button

Import the Post Processing Files

31. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
32. Select the `TPDSG002_dsgDENS00.pch` file, and check the **Import Similar Results for All Design Cycles** check box.

    Checking the check box will cause Design Studio to load many result files (one for each design cycle) in one step.

33. Push the **Open** button

    Alternatively, one can also use the **Import Post..** button in the **Genesis Console Output** window to import the post-processing files

Postprocessing the Results (Density Isosurfaces)

34. Select the **Post** tab
35. Push the **Density Isosurface** button

36. Select a Topology Result for any design cycle

    Identify the created cavities. Is this structure castable?

37. Push the **Options...** button. Slide the bar to display different isosurface values
38. Select the **Show Topology Region** checkbox to view the initial topology space
39. Push the **Close** button
40. Push the **Up** button

Postprocessing the Results (Density Values)

41. Select the **Post** tab
42. Push the **Deform Mesh/Color Mesh** button
43. Select a Topology Result for any design cycle

    Study the results.

44. Push the **Options...** button. Slide the lower cutoff slidebar to mask out element with low density values
45. Push the **Close** button
46. Push the **Up** button

Quit Design Studio

47. From the main menu bar, select **File** → **Quit**
48. Push the **Don't' Save** button

3.3 Creating Surface Meshes That Enclose Topology Results

Introduction

The purpose of this example is to learn how to create and to export a file with a surface that encloses the topology results. The optimization data will be provided. Here you will focus just on how to create the surface file. You will create the surface file using two formats: IGES and the Genesis Input data format. Most CAD programs should be able to read the IGES file while the second format can be used with many pre-processors including Design Studio. The provided input file solves the following problem.

Problem Statement:

Objective function of the problem is to minimize the strain energy of the structure subject to a mass fraction $\leq$ 0.30. The designable region is every element in the structure. The structure is simply supported at 4 corner grids at one end with a point load applied at the opposite end.

Example ID

`TPDSG003`

Files Used in This Problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`TPDSG003.dat`	`Input data`	Provided: This file is the GENESIS input data that is read into Design Studio
`TPDSG003_dsgDENSxx.pch`	`DSG file`	Created: Punch file containing the topology optimization results for the xx design cycle
`TPDSG003_dsgTSURFxx.dat`	`DSG file`	Created: Bulk data file of the topology surface (shells) for the xx design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG003.dat`

Optimize the Structure Using Genesis

3. From the main menu bar, select **Genesis** → **Optimize**

Import the Post-Processing Files (Density Results)

4. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
5. Select the `TPDSG003_dsgDENSxx.pch` file
6. Push the **Open** button

Post-Processing the Results (Density Isosurfaces)

7. Select the **Post** tab
8. Push the **Density Isosurface** button
9. Select the Topology Result for the last design cycle

Export a Surface Representation of the Topology Result, Use Input Data Format

10. From the main menu bar, select **File** → **Export** → **Coarsened Surface...**
11. Enter `TPDSG003TSURF.dat`
12. Push the **Save** button

Export a Surface Representation of the Topology Result, Use IGES Format

13. From the main menu bar, select **File** → **Export** → **Coarsened Surface...**
14. Enter `TPDSG003TSURF`
15. Select **IGES**
16. Push the **Save** button

    This will generate a file named: `TPDSG002TSURF.igs.` Design Studio can not read this file; however, most CAD programs should be able to read it, as IGES is a standard CAD format.

17. Push the **Up** button in the **Post** tab

Import the Surface Representation of the Topology Results

Here you will study the coarse file that you generated above.

18. Import the Genesis data file: `TPDSG003TSURF.dat`

19. Select the **Display** tab
20. Push the **Show/Hide Groups** button
21. Hide PSOLID 4
22. Study the Viewport
23. Using the **Group Display Style** chooser, choose **Flat Shaded**

Import a Genesis Generated Surface Representation of the Topology Results

Genesis itself, creates a surface that represents the solution, as well.

24. Import the Genesis data file: `TPDSG003_dsgTSURFxx.dat`, where xx is the last design cycle (e.g. xx =08)
25. Hide PSHELL 8
26. Hide PSHELL 13500012

    This is the discarded material

27. Compare the two surfaces by turning on and off PSHELL 8 and PSHELL 3500012

    Note that the surface you created with Design Studio has less elements than the one Genesis created. Note that Genesis also creates a surface using the discarded material.
    Can you make Design Studio create a mesh that contains the elements with discarded material?

28. Study the Viewport

Quit Design Studio

29. From the main menu bar, select **File** → **Quit**
30. Push the **Don't Save** button

3.4 Using Multiple Designable Regions with Symmetry Constraints

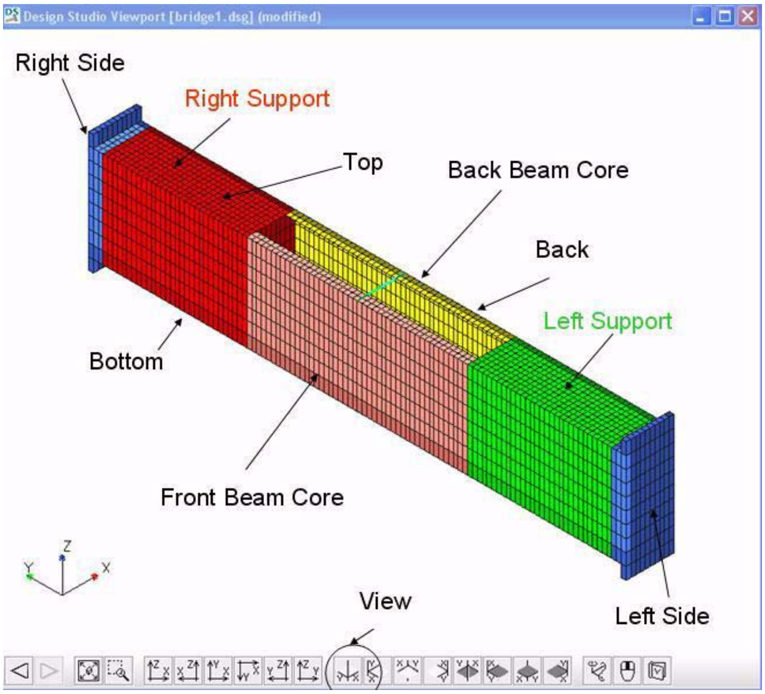
Introduction

The purpose of this example is to demonstrate the creation of topology optimization data using multiple designable regions and to learn how to impose symmetry conditions.

Problem Statement:

The design objective is to minimize the normalized sum of the strain energies of three static loadcases. The normalizations are with respect to initial strain energies. The topology optimization will be subject to a mass fraction $\leq$ 0.20 for each designable part.

The designable regions are the core structure and its supports. The core structure is assembled using two regions of shell elements. The supports are built using solid elements. The supports have to be symmetric with respect to two planes that are parallel to the top and to the front of the structure and pass through the main axis of the structure. The core of the structure has to be symmetric with respect to the plane that passes through the center of the structure and is perpendicular to the main axis of the structure.



The picture above shows the structure and its components. Get familiar with the left and right supports and with the core, as you will design them. The core itself has two main components: the front beam and the back beam.

Example ID

TPDSG004

Files Used in This Problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
TPDSG004.dat	Input data	Provided: This file is the GENESIS input data that is read into Design Studio
TPDSG004_ref.dat	Input data	Provided: Reference file ready to be optimized
TPDSG004_dsgDENSxx.pch	DSG file	Created: Punch file containing the topology optimization results for the xx design cycle
TPDSG004_dsgTSURFxx.dat	DSG file	Created: Bulk data file of the topology surface (shells) for the xx design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG004.dat`

Examine the Center Coordinate System

3. Select the **Display** tab
4. Push the **Feature Lines** Icon
5. Set the **Coordinate System Labels:** radio button to **On**

   Verify that there are two coordinate systems in the model and that one is in the center of the structure.
6. Push the **Flat Shaded** Icon

Create the Designable Regions

Make The Supports Designable:

7. Select the **Topology** tab
8. From the category chooser, select **Topology Regions**

9. Select `Right_Support` (**PSOLID 30**) and `Left_Support` (**PSOLID 40**)

   When selecting from the list, to select more than one group, hold the **Ctrl** key while selecting the second group. When selecting from the viewport, you do not need to hold the **Ctrl** key.

10. Push the **Modify Topology Design** button
11. Use `0.2` for the initial mass fraction
12. Push **Next>**
13. Push the **Change** button to change the coordinate system
14. Select `Coord_Center`
15. Push **Next>**
16. For Constraint 1: Select **MYZ: Mirror about YZ plane**
17. For Constraint 2: Select **MXY: Mirror about XY plane**
18. Push the **Finish** button to complete the design data for the current topology regions

    Verify that there is a hammer icon next to PSOLID 30 and PSOLID 40.
    The hammer icons indicate that the groups are being designed.

Make the Core Structure Designable:

Stay in the **Topology Regions** category of the **Topology** Tab.

19. Select **PSHELL 10** and **PSHELL 20**

    Make sure that only those two groups are selected.

20. Push the **Modify Topology Design** button
21. Use `0.2` for the initial mass fraction
22. Push **Next>**
23. Push the **Change** button to change the coordinate system
24. Select `Coord_Center`
25. Push **Next>**
26. For Constraint 1: Select **MZX**
27. Push the **Finish** button to complete the design data for the topology region

    Verify that the hammer icon is next to the PSHELL 10, PSHELL 20, PSOLID 30 and PSOLID 40.

Defining the Design Objective

28. From the category chooser, select **Topology Objectives**

29. Push the **New Topology Objective** button in the Edit Menu toolbar
30. Accept the default, **Strain Energy** response type. Accept the **Min** Objective Definition switch
31. Push **Next>**
32. Select the 3 existing loadcases

    Use the **Shift** or **Ctrl** keys or from the main menu bar, select **Edit** → **Select All**
33. Push the **Finish** button

    Verify that now there are three responses in the objectives list.

Note:

By default when there are two or more objectives, GENESIS will combine all the objectives using a normalized sum. This is what we want in this example. In other words, in this case GENESIS will use the following objective function:
Objective=1.0*Senergy(Loadcase 1)/Senergy(Loadcase 1, calculated in Design Cycle 0) + 1.0*Senergy(Loadcase 2)/Senergy(Loadcase 2, calculated in Design Cycle 0) + 1.0*Senergy(Loadcase 3)/Senergy(Loadcase 3, calculated in Design Cycle 0)

If you would like to change this default, for example, to add the objectives without normalizing: Go to the **Genesis** Menu, select **Options...**, select the **Design Control** tab, push the **Advanced...** button, select the **Methods** Tab and finally for the **Topology Index** options select **Direct**.

Defining the Design Constraints

34. From the category chooser, select **Topology Constraints**
35. Push the **New Topology Constraint** button in the Edit Menu toolbar
36. For **Name** enter `Core+Supports`
37. Select the **Mass Fraction** response
38. Select **Selected Groups** option of Mass Fraction
39. Enter `0.2` for the **Upper Bound**
40. Push **Next>**
41. Select PSHELL 10, PSHELL 20, PSOLID 30 and PSOLID 40
42. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Note:

Using **Selected Groups** you get the following**:**

Mass fraction of PSHELL 10 ≤ 0.20

Mass fraction of PSHELL 20 ≤ 0.20

Mass fraction of PSOLID 30 ≤ 0.20

Mass fraction of PSOLID 40 ≤ 0.20

Now, if you have used **All Designed Groups**, you would get the following:

Mass fraction of PSHELL 10 + PSHELL 20 + PSOLID 30 + PSOLID 40 ≤ 0.20

Clearing the Selection

43. Right click the Viewport, select **Clear** → **All**

Optimize the Structure Using Genesis

44. From the main menu bar, select **Genesis** → **Optimize**
45. Study the **Design History** charts; when done, push the **Close** button

Import the Post-Processing Files

46. From the **Genesis Console Output** window, select the **Import Post...** button
47. Select the file `TPDSG004_dsgDENSxx.pch` file where `xx` is the last design cycle number
48. Push the **Import** button
49. From the **Genesis Console Output** window, select the **Close** button

Post-Processing the Results (Density Isosurfaces)

50. Select the **Post** tab
51. Push the **Density Isosurface** button
52. Select a Topology Result for the last design cycle
53. Select different views to see the Left, Top, Front and other views
54. Is the structure meeting the symmetry requirements?
55. Push the **Options...** button. Slide the bar to display different isosurface values
56. Select the **Show Topology Region** checkbox to view an outline of the initial topology region
57. Push the **Close** button

Creating a Picture File

Select a good view of the results.

58. From the main menu bar, select **File** → **Print to Image File**
59. Push the **Save** button

    Check your working directory. A picture named `TPDSG004.png` should be there.

Showing all Groups in One Color

60. From the main menu bar, select **Color** → **Group Color Style**→**All One Color**
61. Push the **Up** button in the **Post** tab

Quit Design Studio

62. From the main menu bar, select **File** → **Quit**
63. Push the **Don’t Save** button

3.5 Using Multiple Designable Regions with Symmetry and Fabrication Constraints

Introduction

The purpose of this example is to learn how to impose fabrication constraints along with symmetry constraints in topology optimization. Here you will focus mostly on how to create casting constraints. Symmetry constraints already exist in the provided input file.

Problem Statement:

The design objective is to minimize the normalized sum of the strain energies of three static loadcases. The normalizations are with respect to initial strain energies. The topology optimization will be subject to a mass fraction $\leq$ 0.20 for each designable part.

The designable regions are the core structure and its supports. The core structure is assembled using two regions of shell elements. The supports are built using solid elements. The supports have to be symmetric with respect to two planes that are parallel to the top and to the front of the structure and pass through the main axis of the structure. The core of the structure has to be symmetric with respect to the plane that passes through the center of the structure and is perpendicular to the main axis of the structure.

Your first task will be to add two sets of fabrication constraints to the problem:

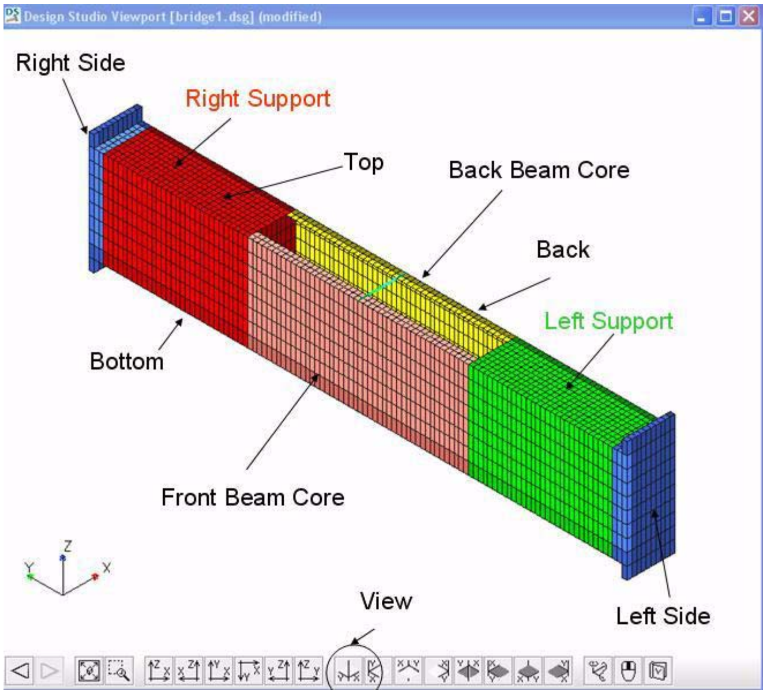
The supports have to be castable in the Y direction

The beams have to be castable in the Z direction

In addition, the mass fraction constraint for the core is 25%. In other words, the mass fraction constraint associated to the shell elements has to be changed to:

Mass Fraction $\leq$ 0.25

The following picture shows the structure and its components. Get familiar with the left and right supports and with the core, as you will design them. The core itself has two main components: the front beam and the back beam.



Example ID

TPDSG005

Files Used in This Problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
TPDSG005.dat	Input data	Provided: This file is the GENESIS input data that is read into Design Studio
TPDSG005_dsgDENSxx.pch	DSG file	Created: Punch file containing the topology optimization results for the xx design cycle
TPDSG005_dsgTSURFxx.dat	DSG file	Created: Bulk data file of the topology surface (shells) for the xx design cycle

Start Design Studio

1. Start Design Studio

2. Import the `TPDSG005.dat` file

Modify the Designable Regions

You will now change the initial mass fraction values of the shell regions to be 0.25. At the same time you will be adding the casting fabrication constraints.

3. Select the **Topology** tab
4. From the category chooser, select **Topology Regions**
5. Select both PSHELL 10, PSHELL 20
6. Push the **Modify Topology Design** button
7. Put a check in the **Initial Mass Fraction** checkbox

   If you do not check the checkbox, your change will only affect one of the selected PSHELLs.
8. Change the Mass fraction value to `0.25`
9. Push **Next>**
10. Put a check in the Constraint 2 checkbox
11. For Constraint 2: Select **FGZ: Fill Z axis (Inside to out)**
12. Put a check in the **Minimum Size** checkbox
13. For **Minimum Size** enter `1.0`

    The minimum size corresponds to minimum member size. Larger values produce larger members.
14. Put a check in the **Spread Fraction** checkbox
15. For **Spread Fraction** enter `0.5`

    The spread fraction parameter introduces smoothing in the topology results. Typical values are between 0.0 to 0.5. A value of 0.0 represents no additional smoothness. A value of 0.5 often gives a good smoothness level. The larger the spread fraction the larger the number of design variables that Genesis will generate and use in the optimization problem.
16. Push the **Finish** button
17. Select PSOLID 30 and PSOLID 40
18. Push the **Modify Topology Design** button
19. Push **Next>**
20. Put a check in the Constraint 3 checkbox
21. For Constraint 3: Select **FGY: Fill Y axis (Inside to out)**
22. Put a check in the **Minimum Size** checkbox
23. For **Minimum Size** enter `1.0`

24. Put a check in the **Spread Fraction** checkbox
25. For **Spread Fraction** enter `0.5`
26. Push the **Finish** button

Modify the Constraints for the Core (PSHELL)

You will now change the upper bound of the mass fraction constraint of the core (PSHELLs) to be 0.25.

27. From the category chooser, select **Topology Constraints**
28. Select the Constraint named `Core+Supports`
29. Push the **Modify Topology Constraint** button from the Edit Menu toolbar
30. Change **Name** to `Core`
31. Enter `0.25` for the **Upper Bound**
32. Push **Next>**
33. Select the `Back_Beam` and the `Front_Beam` (the PSHELL) groups and deselect the `Right_Support` and the `Left_Support` (the PSOLID) groups

    Later you will need to re-create the constraints associated with the support (PSOLID) groups.
34. Push the **Finish** button

Re-Create the Constraints for the Supports (PSOLIDs)

You will now add back the upper bound of the mass fraction constraint of the core (PSOLIDs) to be 0.20.

35. Push the **New Topology Constraint** button in the Edit Menu toolbar
36. For **Name** enter `Supports`
37. Enter `0.20` for the **Upper Bound**
38. Push **Next>**
39. Select the `Right_Support` and the `Left_Support` (the two PSOLID) groups
40. Push the **Finish** button

Optimize the Structure Using Genesis

41. From the main menu bar, select **Genesis** → **Optimize**

Import the Post-Processing Files

42. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
43. Select the `TPDSG005_dsgDENSxx.pch` file where `xx` is the last design cycle number
44. Push the **Open** button

Post-Processing the Results (Density Isosurfaces)

45. Select the **Post** tab
46. Push the **Density Isosurface** button
47. Select a Topology Result for the last design cycle
48. Is the structure meeting the symmetry requirements?
49. Is the structure meeting the manufacturing requirements?

    Check if the shell elements are growing in the Z direction from inside to out.
    Check if each support is growing in the Y direction from inside to out.
50. Push the **Options...** button. Slide the bar to display different isosurface values
51. Select the **Show Topology Region** checkbox to view the initial topology region
52. Push the **Close** button

Creating a Picture File

Select a good view of the results.

53. From the main menu bar, select **File** → **Print to Image File**
54. Push the **Save** button

    Check your working directory. A picture name `TPDSG005.png` should be there.
55. Push the **Up** button

Quit Design Studio

56. From the main menu bar, select **File** → **Quit**
57. Push the **Don't Save** button

3.6 Torsional Frequency Maximization Using Ribs generated by AUTORIB and Mode Tracking

Introduction

The main purpose of this example is to demonstrate how to improve a given frequency (using mode tracking) of a structure by reinforcing it using structural ribs. This example goes through the process of how to create candidate ribs automatically and how to use topology optimization to select the most important ribs.

Problem Statement:

The design objective is to maximize the first natural torsional frequency, subject to a mass fraction $\leq$ 0.08 of added ribs. The designable region are the added ribs created. The structure is free to vibrate and it has 6 rigid body modes

The problem is divided into three parts that go through the following tasks:

1) How to generate structural ribs

2) How to create a topology optimization problem with natural frequency responses

3) How to create an updated input file that contains the optimal results

The 3 parts of this exercise should be performed sequentially. However, any of the parts can be skipped, as the needed results are provided.

Example ID

TPDSG006

Files Used in This problem

A list of the key files is provided and the ones that will be created during this exercise is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification

Part	File Name	Description
1	TPDSG006_1.dat	Provided: Contains the finite element mesh and the eigenvalue loadcase. There are no ribs in this model.
1,2	TPDSG006_2.dat	Generated in part 1 and used as a starting file for part 2. This file is the same as the TPDSG006_1.dat file but it contains the Autorib data.
1,2	TPDSG006_2_ref.dat	Provided: Reference file same as TPDSG006_2.dat file

2,3	TPDSG006_3.dat	Generated in part 2 and is used as the starting file for part 3. This file is obtained by adding topology data to the TPDSG006_2.dat file
2,3	TPDSG006_3_ref.dat	Provided: Reference file same as TPDSG006_3.dat file
3	TPDSG006_4.dat	Generated in part 3.This is Genesis input data with the FE model of the structure reinforced with the best ribs.
3	TPDSG006_4_ref.dat	Provided: Reference file same as TPDSG006_4.dat file

3.6.1 Part 1

The purpose of this part of the example is to learn how to generate candidate ribs. If you are familiar with this task, you can skip this part and go to Part 2.

When you finish this example, you should have created a file named: `TPDSG006_2.dat`.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG006_1.dat`

Create Material Properties for the AUTORIB elements

3. From the **Analysis** category chooser, select **Materials**
4. Push the **New Material** button in the Edit Menu toolbar
5. Enter `Rib_Material` in the name field
6. Select the **ISOTROPIC (MAT1)** as material type to be created
7. Push **Next**>
8. Enter the **E**=`200000.0`, **Nu** = `0.3` and **Rho** = `7.59E-9`
9. Push the **Finish** button

Create AUTORIB elements

10. From the **Analysis** category chooser, select **Elements**
11. Push the **New Elements** button in the Edit Menu toolbar
12. Select **Generate Autorib Elements** option
13. Push **Next**>
14. Select the `PSHELL 3, PSHELL 10, PSHELL 12` as the groups to cover with ribs.

    Use the **Ctrl** key to select multiple items
15. Push **Next**>
16. Select `Rib_Material` for the **Rib Material**
17. Enter `2.0` for **Rib Thickness** and `10.0` for **Rib Height**
18. Select `1CBAR` for the **Elements Through Rib Height** option.

19. Push the **Finish** button.

    Ribs are created in a new group and named as `Autorib`. To change the properties of the ribs, one can got to the **Analysis** tab to **Group Properties**, select the `Autorib` group and push the **Modify Group Property** button from the Edit Menu toolbar.

Export the Input File

20. From the main menu bar, select **File** → **Export** → **Input Data...**
21. Enter `TPDSG006_2.dat`
22. Push the **Save** button

Quit Design Studio

23. From the main menu bar, select **File** → **Quit**

3.6.2 Part 2

The purpose of this part of the example is to create the necessary topology data and to perform the topology optimization.

If you do not have the `TPDSG006_2.dat` file generated in Part 1, copy the file `TPDSG006_2_ref.dat` to `TPDSG006_2.dat`.

When you finish this example, you should have created a file named: `TPDSG006_3.dat.`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG006_2.dat`

   Visually verify that there are ribs attached to every element in the floor of the model.

Create the Designable Region

3. Select the **Topology** tab
4. From the category chooser, select **Topology Regions**
5. Select the PBARL group `Autorib`
6. Push the **Modify Topology Design** button
7. Use `0.08` for the initial mass fraction
8. Push the **Finish** button to complete the design data for the topology region

   Verify that there is the hammer icon next to the PBARL group.
   The hammer icon indicates that the group is being designed.

Defining the Design Objective

9. From the category chooser, select **Topology Objectives**
10. Push the **New Topology Objective** button in the Edit Menu toolbar
11. Select **Frequency Mode Number** response type
12. For the **Frequency Mode Number,** enter `8`

    We will need to verify this number. After the ribs have been added, the torsion mode might have changed (if you run the problem without the rubs the torsional mode should be 8). For now we use 8.

13. Select the **Max** Objective Definition switch
14. Push **Next>**

15. Select the existing loadcase
16. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Defining the Design Constraints

17. From the category chooser, select **Topology Constraints**
18. Push the **New Topology Constraint** button in the Edit Menu toolbar
19. Enter Name `RIBS`
20. Select the **Mass Fraction** response
21. Enter `0.08` for the **Upper Bound**
22. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Save the Design Studio Database File

23. From the main menu bar, select **File** → **Save As...**
24. Enter `TPDSG006_3` as the Filename and push **Save (**as a Design Studio File**)**

Export the Input File

25. From the main menu bar, select **File** → **Export** → **Input Data...**
26. Enter `TPDSG006_3.dat`
27. Push the **Save** button

Analyze the Structure Using Genesis

We will perform one single analysis to check if mode 8 is still the torsional mode.

28. From the main menu bar, select **Genesis** → **Single Analysis**
29. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

30. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
31. Select the `TPDSG006_3_dsg00.pch` file
32. Push the **Open** button

Post-Processing the Results (Mode Shape of Analysis)

33. Select the **Post** tab
34. Push the **Deform Mesh/Color Mesh** button
35. Push the **Filled Contour** Button, this button is for the Color Mesh Window
36. Animate the first 8 to 10 modes searching for the torsional mode
37. Verify that the first 6 modes are rigid body modes
38. Verify that the 8th mode is still the first torsional mode

    Since the mode has not changed, our objective function is still mode 8. If the torsion mode is not 8, change the number in the Objective Function definition.

39. Push the **Up** button

Optimize the Structure

40. From the main menu bar, select **Genesis** → **Optimize**

Study the Output File

41. In the text editor load the file: `TPDSG006_3_dsg.out`
42. Study briefly the "MODE TRACKING TABLE" in every design cycle. Make sure that the status is 0 at the end of the optimization

    If the status is not 0, mode tracking might have lost the mode. When a mode is lost, the easier fix is to re-run Genesis using smaller move limits. Sometimes, the number of modes or the upper level of frequencies V2 has to be increased as frequencies move up in value.

    The best check, to know if mode tracking has correctly tracked the mode, is to animate the modes of the last design cycle. If the mode is lost, you might find it and might be able to re-start Genesis with the new number from the last design cycle that ran without problems.

Clean the Post-Processing Results

43. Select the **Post** tab
44. Push the **Manage Result Dataset** button
45. From the main menu bar, select **Edit** → **Select All**
46. Push the **Delete Result Set** button from the Edit Menu toolbar

Import the Post-Processing Files

47. From the **Genesis Console Output** window, select the **Import Post...** button
48. Select all the `TPDSG006_3_dsgDENSxx.pch` and `TPDSG006_3_dsgxx.pch` files where `xx` is the design cycle number
49. Push the **Import** button
50. In the **Genesis Console Output** window, push the **Close** button

Post-Processing the Results (Mode Shape of all Design Cycles)

51. Push the **Up** button
52. Push the **Deform Mesh/Color Mesh** button
53. Push the **Filled Contours** Button
54. Animate the mode 8 for the last design cycle
55. Verify that the 8th mode is still the first torsional mode

    If mode 8 is still the torsional mode, we do not need to change anything.
56. Animate mode 8 for different design cycle
57. Push the **Up** button

Post-Processing the Results (Densities)

58. Push the **Deform Mesh/Color Mesh** button
59. Push the two **Clear** buttons
60. Push the **Filled Elements** buttons
61. Select the **Topology Results** for the last cycle
62. Push the **Options...** button
63. Push the **Hide Element With No Value** button

    The most important elements to keep are colored in red.

Quit Design Studio

64. From the main menu bar, select **File** → **Quit**
65. Push the **Don’t Save** button

3.6.3 Part 3

The purpose of this part of the example is to learn how to create a structure containing only the most important ribs discarding the less important ones. Also this example will be used to verify the topology optimization results of the last design cycle obtained in Part 2.

If, for any reason, you do not have the `TPDSG006_3.dat` file generated in Part 2, copy the file `TPDSG006_3_ref.dat` to `TPDSG006_3.dat`.

When you finish this example, you should have created a file named: `TPDSG006_4.dat.`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG006_3.dat`

Creating a New Group

You will create a new PBARL Group. This PBARL will be used to group together the bar elements that you select as the most important for reinforcement.

3. Select the **Display** tab
4. Push the **Manage Groups** button
5. Select the **PBARL** group
6. From the Edit menu toolbar, select the **Copy Group** button
7. From the Edit menu toolbar, select the **Paste Group** button
8. Select the new **PBARL** group
9. Push the **Modify Group** button from the Edit Menu toolbar
10. For Name enter `Reinforcement`
11. Change the comments from “`Automatically generated rib candidate elements`” to “`Topologically found ribs`”
12. Push the **Finish** button
13. Push the **Up** button

Optimize the Structure

If you completed Part 2, you can skip this step.

14. From the main menu bar, select **Genesis** → **Optimize**

Import the Post-Processing Files (Densities)

15. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
16. Select the `TPDSG006_3_dsgDENSxx.pch` file, where `xx` is the last design cycle
17. Push the **Open** button

Post-Processing the Results (Density Values)

18. Select the **Post** tab
19. Push the **Deform Mesh/Color Mesh** button
20. Select the Topology Result for the last design cycle
21. Push the **Options...** button
22. Select the checkbox for **Hide Elements With No Value**
23. Slide the lower cutoff slidebar to mask out element with low density values
24. Push the **Close** button

Add the Element to the Selection

Without pushing the **Up** button in the **Deform Mesh/Color Mesh** section:

25. Select the **Display** tab
26. Push the **Show/Hide Elements** button
27. Push the **Select All** button

    Verify that there are about 25-28 elements in the selection.

28. Push the **Up** button

Clear the Density Results

29. Select the **Post** tab
30. Push the **Up** button

Moving Elements with High Density to a New Group

Now we will move the <u>selected</u> elements to the reinforced group.

31. Select the **Analysis** tab
32. From the category chooser, select **Elements**

    Verify that there are approximately 25-28 elements in the selection.

33. Push the **Modify Elements** button from the Edit Menu toolbar
34. Note: Check that the **Change Element's group** radio button is selected
35. Push **Next>**
36. Select the `Reinforcement` group
37. Push the **Finish** button

Check the Groups

38. Select the **Display** tab
39. Push the **Show/Hide Groups** button
40. Push the **Hide All** button
41. Turn on and off the two PBARL groups

Note:

If you are satisfied with the solution, you are now almost ready to delete the non-important bars, otherwise you can manually change elements from the two bar groups as you deem appropriate; one reason might be to try to make the reinforcement mesh symmetric. Before deleting the non-important elements you will delete the topology optimization data.

Deleting the Topology Data

If you do not delete the topology regions, when you run Genesis again, the ribs will be used with the initial density value (0.08), not with the full density as they should.

42. Select the **Topology** tab
43. From the category chooser, select **Topology Regions**
44. Push **Select** button

    You have to select PBARL `Autorib` too, if you do not see it: Select uncheck **List only assembly**

45. Push **Delete Topology Design** button

    Verify that no group has the hammer next to its color.

46. From the category chooser, select **Topology Objectives**
47. Select the objective
48. Push the **Delete Topology Objective** button from the Edit Menu toolbar

49. From the category chooser, select **Topology Constraints**
50. Select the existing constraint
51. Push the **Delete Topology Constraint** button from the Edit Menu toolbar

Verify That There is no Topology Data

52. From the category chooser, select **Quick Setup Trails**

    Verify that there are 0 **Topology Regions Defined**, 0 **Objectives Defined** and 0 **Constraints Defined.**

Deleting the Elements with Low Density

Now we will delete the elements that are less important.

53. Select the **Display** tab
54. If necessary, push the **Show/Hide Groups** button
55. **Hide** all groups with the exception of the PBARL Autorib group
56. Select the **Analysis** tab
57. From the category chooser, select **Elements**
58. Push the **Select All** button to select all the visible bars
59. Push the **Delete Elements** button from the Edit Menu toolbar

    Now, all elements with low density have been deleted.

60. From the category chooser, select **Grids**
61. Push the **Select All** button
62. Push the **Delete Grids** button from the Edit Menu toolbar

    Now, all grids associated with the deleted elements have been deleted.

63. Select the **Display** tab
64. **Show All** the groups
65. Push the **Up** button
66. Push the **Manage Groups** button
67. Select the PBARL `Autorib` group (whose elements were all just deleted)
68. Push the **Delete Group** button from the Edit Menu toolbar

    Now, the original autorib group has been deleted.

Save the Design Studio Database File

69. From the main menu bar, select **File** → **Save As...**
70. Enter `TPDSG006_4` as the Filename and push **Save (**as a Design Studio File**)**

Analyze the Structure Using Genesis

71. From the main menu bar, select **Genesis** → **Single Analysis**

    Study the **Genesis Console Output**; when done, push the **Close** button.

Clean the Post-Processing Results

72. Select the **Post** tab
73. Push the **Manage Result Dataset** button
74. From the main menu bar, select **Edit** → **Select All**
75. Push the **Delete Result Set** button from the Edit Menu toolbar
76. Push the **Up** button

Import the Post-Processing Files

77. Using similar steps described above; import the `TPDSG006_4_dsg00.pch` results

Post-Processing the Results (Mode Shape)

78. Push the **Deform Mesh/Color Mesh** button
79. Push the **Filled Contours** Button
80. Push the **Options..** button and reset the Lower Cutoff slider to lowest value
81. Animate the modes

    What is the mode shape number of the torsional frequency?
    What is the torsional frequency?

82. Push the **Up** button

Export the Input File

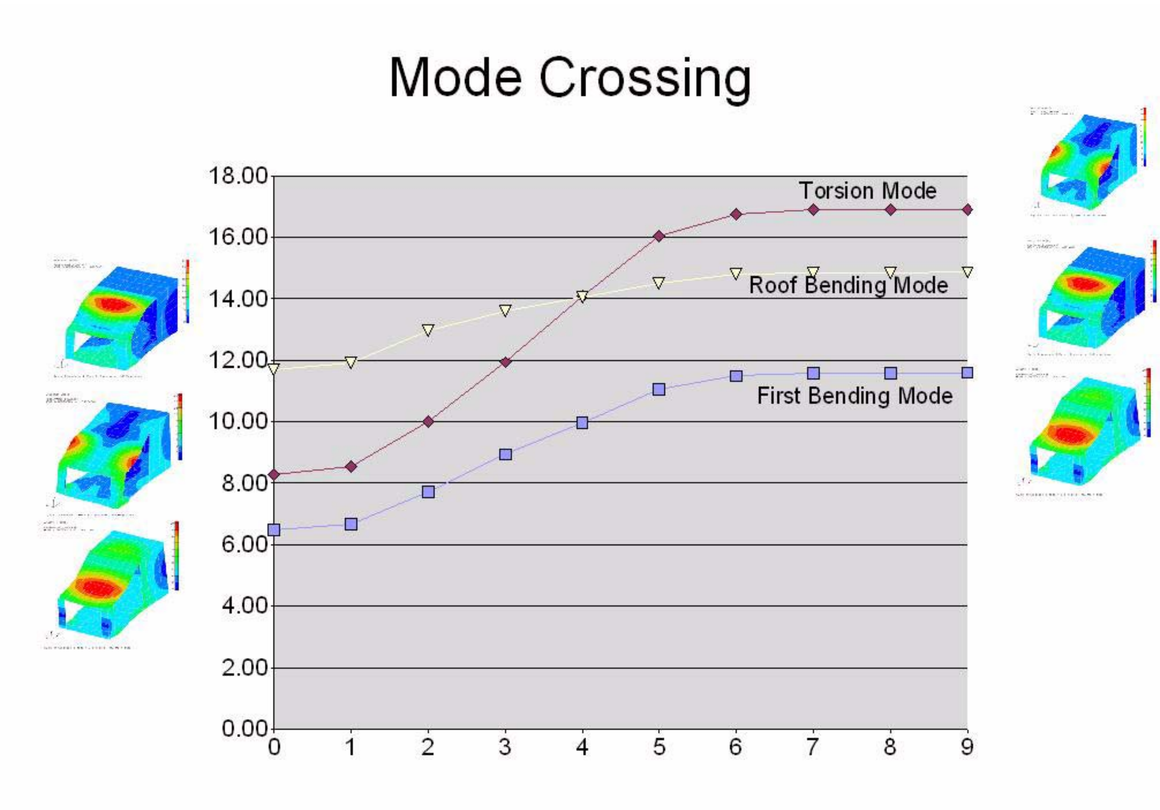
83. From the main menu bar, select **File** → **Export** → **Input Data...**
84. Enter `TPDSG006_4.dat`
85. Push the **Save** button

Quit Design Studio

86. From the main menu bar, select **File** → **Quit**
87. Push the **Don’t Save** button

Mode Tracking in Action

The following figure shows the crossing of the torsional mode and the roof bending mode. In design cycle 0 the torsional mode is number 8, in the last design cycle the torsional mode is 9. On the other hand, in design cycle 0 the roof bending mode is 9 and in the last design cycle is 8. The first bending mode does not change the number, it starts and ends as the 7th mode.



3.7 Reinforcing a Solid Part Using Topology Optimization on a Design Studio Generated Skin Layer

Introduction

The purpose of this exercise is to learn how to reinforce solid parts using shell skins. First the creation of the skin using Design Studio is demonstrated. Later topology optimization is performed to design the skin.

Problem Statement:

The design objective is to maximize the frequencies for the 1st & 2nd elastic natural modes. The objective function is to incorporate weighting factors for the first and second elastic frequencies of 3.0 and 1.0, respectively. A mass fraction ≤ 0.15 of the reinforcement region (added skin) is used as the design constraint. The design region is every shell element of the added skin.

During structural analysis the structure is free to vibrate and it has 6 rigid body modes. The problem is divided into two parts that will help you learn/review the following tasks:

1) How to generate a surface mesh of shell elements (skin)

2) How to create a topology optimization problem with natural frequency responses

The two parts of this exercise should be performed sequentially. However, if you want to skip the first part you can, as the needed results are provided.

Example ID

`TPDSG007`

Files Used in This problem

A list of the key files is provided and the ones that will be created during this exercise is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification

Part	File Name	Description
1	`TPDSG007_1.dat`	Provided: Contains the finite element mesh and the eigenvalue loadcase. There are no ribs in this model.
1,2	`TPDSG007_2.dat`	Generated in part 1 and used as a starting file for part 2. This file is the same as the `TPDSG007_1.dat` file but also contains the reinforcement skin data.
1,2	`TPDSG007_2_ref.dat`	Provided: Reference file same as `TPDSG007_2.dat` file

2	TPDSG007_3.dat	Generated in part 2.This is Genesis input data with the FE model of the structure ready to be optimized.
2	TPDSG007_3_ref.dat	Provided: Reference file same as TPDSG007_3.dat file

3.7.1 Part 1

The purpose of this part of the example is to learn how to create skins around solid parts. You will also study the natural frequency and the mode shapes of the structure. If you are familiar with these tasks, you can skip this part and go to Part 2.

When you finish this example, you should have created a file named: `TPDSG007_2.dat.`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG007_1.dat`

Analyze the Structure Using Genesis

3. From the main menu bar, select **Genesis** → **Single Analysis**

   Study the **Genesis Console Output**; when done, push the **Close** button.

Import the Post-Processing Files

4. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
5. Select the `TPDSG007_1_dsg00.pch` file
6. Push the **Open** button

Post-Processing the Results (Mode Shape)

7. Select the **Post** tab
8. Push the **Deform Mesh/Color Mesh** button
9. Push the **Filled Contour** Button, this button is in the **Color Mesh** frame
10. From the **Color Mesh** list, select Mode 1 result for design cycle 0
11. To animate, push the **Oscillate** radio button
12. Verify that the first 6 modes are rigid body modes
13. Inspect modes 7, 8 and some others
14. Push the **Up** button

Study the Output File

15. Start any text editor

16. In the text editor load the Genesis data file: `TPDSG007_1_dsg.out`
17. Study briefly the file
18. Using the output file complete the following table

Natural Frequency Number	**Natural Frequency Original Reference Solution (1)**	**Natural Frequency (2)**
7	291.4	
8	455.6	
9	644.2	
10	772.3	
Mass	263Kg	

(1) Result from baseline GENESIS analysis
(2) Result from your run, `TPDSG007_1_dsg.out`

Your values should be very close to the baseline reference values.

19. Close the output file

Create a New Group

20. Select the **Display** tab
21. Push the **Manage Groups** button
22. Push the **New Group** button in the Edit Menu toolbar
23. Enter Name: `Reinforcement_Skin`
24. Pick Type: PSHELL
25. Pick Red Color
26. Push the **Finish** button
27. Push the **Up** button

Assign Property Values to the New Group

28. Select the **Analysis** tab
29. From the category chooser, select **Group Properties**
30. Select the **PSHELL** you just created, `Reinforcement_Skin`

31. Push the **Modify Group Property** button from the Edit Menu toolbar
32. For thickness: Enter `4.0`
33. Push the **Finish** button

Create the Skin

34. From the category chooser, of the **Analysis** Tab, select **Elements**
35. Push the **New Elements** button in the Edit Menu toolbar
36. Select the **Surface elements from Selected Solid Elements** option
37. Push **Next>**
38. Push the **Select All** button

    Verify that there are 5442 elements selected.
39. Push **Next>**
40. Select the PSHELL group
41. Push the **Finish** button, to finish the creation of the skin

Visualize the Skin

42. Select the **Display** Tab
43. Push the **Show/Hide Groups** button
44. Hide the PSOLID groups

    Notice that the shell elements forming the skin are shown
45. Push the **Up** button
46. For the **Model Cutaway**, select the **Cut V.C.S, X axis, hide - side** option
47. Use the slider below to change the cutting plane

    Notice that only a layer of shells is created for the surface solid elements.
48. For the **Model Cutaway**, select the **None** option to view the entire model
49. Push the **Show/Hide Groups** button and select the Show All button to display all groups

Export the Input File

50. From the main menu bar, select **File** → **Export** → **Input Data...**
51. Enter `TPDSG007_2.dat`

52. Push the **Save** button

Quit Design Studio

53. From the main menu bar, select **File** → **Quit**
54. Push the **Don’t Save** button

3.7.2 Part 2

The purpose of this part of the example is to create the necessary topology data and to perform the topology optimization.

If, for any reason, you do not have the `TPDSG007_2.dat` file generated in part 1, copy the file `TPDSG007_2_ref.dat` to `TPDSG007_2.dat`.

When you finish this example, you should have created a file named: `TPDSG007_3.dat.`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG007_2.dat`

   Verify that there is a skin of shell elements.

Review the Skin

3. Select the **Display** Tab
4. Push the **Show/Hide Groups** button
5. Hide the PSOLID groups

   The skin is in red.

Create the Designable Region

6. Select the **Topology** tab
7. From the category chooser, select **Topology Regions**
8. Select PSHELL 5
9. Push the **Modify Topology Design** button
10. Use `0.15` for the initial mass fraction
11. Push the **Finish** button to complete the design data for the topology region

    Verify that there is the hammer icon next to the PSHELL 5.
    The hammer icon indicates that the group is being designed.

Defining the Design Objective

We will create two objectives that will use the first 2 elastic modes (modes 7 and 8).

12. From the category chooser, select **Topology Objectives**

13. Push the **New Topology Objective** button in the Edit Menu toolbar
14. For Name enter: `Freq_7`
15. Select **Frequency Mode Number** response type.
16. For the **Frequency Mode Number,** enter `7`
17. Select the **Max** Objective Definition switch
18. For **Weight** Factor, Enter `3.0`
19. Push **Next>**
20. Select the existing loadcase
21. Push the **Finish** button

    Verify that now there is one response in the objectives list.

22. Repeat the above steps to create an additional objective using the following information:

Objective Name	Mode Number	Objective Switch	Weight Factor
`Freq_8`	`8`	`Max`	`1.0`

Defining the Design Constraints

23. From the category chooser, select **Topology Constraints**
24. Push the **New Topology Constraint** button in the Edit Menu toolbar
25. Enter Name `Skin`
26. Select the **Mass Fraction** response
27. Enter `0.15` for the **Upper Bound**
28. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Request to Print Results for All Design Cycles

GENESIS, by default, prints the design results for all design cycles, but in the current input file the command APRINT=FLAST is used and that will cause Genesis to print the results for only the first and last design cycles. The following steps will change the APRINT command to APRINT=ALL.

29. From the main menu bar, select **Genesis** → **Options**
30. Select the **Output Control** tab
31. Change the **Analysis Output** to: **All Cycles**

32. Push the **Apply** button

Save the Design Studio Database File

33. From the main menu bar, select **File** → **Save As...**
34. Enter `TPDSG007_3` as the Filename and push **Save (**as a Design Studio File**)**

Export the Input File

35. From the main menu bar, select **File** → **Export** → **Input Data...**
36. Enter `TPDSG007_3.dat`
37. Push the **Save** button

Optimize the Structure

38. From the main menu bar, select **Genesis** → **Optimize**

Import the Post-Processing Files (Densities)

39. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
40. Select the `TPDSG007_3_dsgDENS00.pch` file and check the **Import Similar Results for All Design Cycles** checkbox
41. Push the **Open** button

Post-Processing the Results (Densities)

42. Select the **Post** tab
43. Push the **Deform Mesh/Color Mesh** button
44. Select the **Topology Results** for the last cycle
45. Push the **Options..** button
46. Check the **Hide Elements with No Value** checkbox
47. Push the **Close** button

Import the Post-Processing Files (Modes of All Design Cycles)

48. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**

49. Select the `TPDSG007_3_dsg00.pch` file and check the **Import Similar Results for All Design Cycles** checkbox
50. Push the **Open** button

Post-Processing the Results (Mode Shape of all Design Cycles)

51. Animate mode 7, 8 and 9 for design cycles 0 and last
52. Push the **Up** button

Quit Design Studio

53. From the main menu bar, select **File** → **Quit**
54. Push the **Don't Save** button

Study the Output File

55. Load in a text editor the file: `TPDSG007_3_dsg.out`
56. Complete the following table

Natural Frequency Number	Natural Frequency Original Reference Solution (1)	Natural Frequency Topology Reference Solution (2)	Natural Frequency Topology Results (3)
7	291.4	313.8	
8	455.6	472.4	
9	644.22	648.3	
10	772.31	817.7	
MASS	263	277Kg	

(1) Result from the baseline GENESIS analysis
(2) Result from reference GENESIS topology optimization
(3) Result from your run, `TPDSG007_3_dsg.out`

57. Study the design cycle history printed at the end of the file
58. Why the initial value is 4.0?

Hint. The objective function in this problem is constructed by Genesis using the following equation:
Objective=3.0*Freq_7(Design Cycle 0)/Freq_7+1.0*Freq_8(Design Cycle 0)/Freq_8
This equation corresponds to the Genesis DOPT parameter TINDEXM=0 or Normalized

Reciprocal.
Other formulations are possible. Here is the list of options:
TINDEXM=1 (Normalized Direct)
Objective=-3.0*Freq_7/Freq_7(Design Cycle 0)-1.0*Freq_8/Freq_8(Design Cycle 0)
TINDEXM=2 (Reciprocal)
Objective=3.0/Freq_7+1.0/Freq_8
TINDEXM=3(Direct)
Objective=-3.0*Freq_7-1.0*Freq_8

Note:

Design Studio can be used to pick any of these formulations. In the **Genesis** Menu, Select **Options...**, select **Design Control** Tab, push **Advanced...** button**, select Methods** Tab, pick your option in **Topology Index.**

3.8 Stiffness Maximization with Buckling Constraints

Introduction

The purpose of this exercise is to demonstrate how to set up a topology optimization problem with buckling constraints.

Problem Statement:

The design objective is to minimize the global strain energy subject to the mass fraction <= 0.65 for the designable region comprised of every element of the inner part. In addition to the mass fraction constraint, five buckling design constraints are also defined for this problem:

First Buckling Load factor Lama_1 >= 2.0
Second Buckling Load factor Lama_2 >= 2.5
Third Buckling Load factor Lama_3 >= 3.0
Fourth Buckling Load factor Lama_4 >= 3.5
Fifth Buckling Load factor Lama_5 >= 4.0

In the provided analysis problem, there is one static analysis loadcase, and one buckling loadcase. The structure is fixed in one end and loaded in the opposite end with compressive loads

The problem is divided into two parts that will help you learn the following tasks:

1) How to set up a topology optimization problem with buckling responses

2) How to create an input data file that represents the topology optimization solution

The two parts of this exercise should be done sequentially. However, if you want to skip the first part, the necessary files for part two are included.

Example ID

TPDSG008

Files Used in This problem

A list of the key files is provided and the ones that will be created during this exercise is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification

Part	File Name	Description
1	TPDSG008_1.dat	Provided: Contains the finite element mesh and the buckling loadcase. There are no ribs in this model.

1,2	TPDSG008_2.dat	Generated in part 1 and used as a starting file for part 2. This file is the same as the TPDSG008_1.dat file along with topology optimization data.
1,2	TPDSG008_2_ref.dat	Provided: Reference file same as TPDSG008_2.dat file
2	TPDSG008_3.dat	Generated in part 2.This is Genesis input data with the FE model of the structure updated based on the optimization results
2	TPDSG008_3_ref.dat	Provided: Reference file same as TPDSG008_3.dat file

3.8.1 Part 1

The purpose of this part of the example is to learn how to perform a topology optimization with buckling constraints.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG008_1.dat`

Create the Designable Region

3. Select the **Topology** tab
4. From the category chooser, select **Topology Regions**
5. Select center region

   You can either click on the part on the screen or select PSHELL 3 from the list.
6. Push the **Modify Topology Design** button
7. Use `0.65` for the initial mass fraction
8. Push **Next>**
9. Push the **Change** button and select the `Center_CS` coordinate system
10. Push **Next>**
11. For constraint 1, Select **MYZ: Mirror about YZ plane**
12. For constraint 2, Select **MZX: Mirror about XZ plane**
13. Push the **Finish** button to complete the design data for the topology region

    Verify that there is the hammer icon next to the PSHELL 3.
    You have created a designable region with two symmetry conditions. The structure that topology will create should be double symmetric. The symmetry conditions you picked will rely on the mesh being symmetric. If the mesh is not symmetric, you could use other manufacturing conditions to avoid the symmetry requirements on the mesh.

Defining the Design Objective

14. Select the **Topology** tab
15. From the category chooser, select **Topology Objectives**
16. Push the **New Topology Objective** button in the Edit Menu toolbar
17. Select **Strain Energy** response type

18. Push **Next>**
19. Select the listed Static loadcase
20. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Defining the First Buckling Constraints

21. From the category chooser, select **Topology Constraints**
22. Push the **New Topology Constraint** button in the Edit Menu toolbar
23. Enter Name `Lama_1`
24. Select the **Buckling Load Factor** response
25. On the Buckling Load Factor Box, enter `1`
26. Enter `2.0` for the **Lower Bound**
27. Push **Next>**
28. Select the listed Buckling loadcase
29. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Additional Buckling Constraints

30. Select the constraint you just created
31. From the Edit menu toolbar, select the **Copy Constraint** button
32. From the Edit menu toolbar, select the **Paste Constraint** button
33. Repeat the Paste operation three more times to obtain five constraints in total
34. Select the second constraint
35. Push the **Modify Constraint** button from the Edit Menu toolbar
36. Change the name to: `Lama_2`
37. On the **Buckling Load Factor** box, enter `2`
38. Enter `2.5` for the **Lower Bound**
39. Push the **Finish** button
40. Repeat the above steps to modify the last 3 constraints using the following information:

Constraint Name	Buckling Load Factor	Lower Bound

Lama_3	3	3.0
Lama_4	4	3.5
Lama_5	5	4.0

Defining the Mass Constraints

41. Push the **New Topology Constraint** button in the Edit Menu toolbar
42. Enter Name `MassFr`
43. Select the **Mass Fraction** response
44. Enter `0.65` for the **Upper Bound**
45. Push the **Finish** button

    Verify that now there are six responses in the constraint list.

Save the Design Studio Database File.

46. From the main menu bar, select **File** → **Save As...**
47. Enter `TPDSG008_2` as the Filename and push **Save (**as a Design Studio File**)**

Export the Input File

48. From the main menu bar, select **File** → **Export** → **Input Data...**
49. Enter `TPDSG008_2.dat`
50. Push the **Save** button

Optimize the Structure

51. From the main menu bar, select **Genesis** → **Optimize**

Import the Post-Processing Files (Densities)

52. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
53. Select the `TPDSG008_2_dsgDENS00.pch` file and check the **Import Similar Results for All Design Cycles** checkbox
54. Push the **Open** button

Post-Processing the Results (Densities)

55. Select the **Post** tab
56. Push the **Deform Mesh/Color Mesh** button
57. Push the **Filled Elements** buttons
58. Select the **Topology Results** for the last cycle

Import the Post-Processing Files (Modes of All Design Cycles)

59. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
60. Select the `TPDSG008_2_dsg00.pch` file and check the **Import Similar Results for All Design Cycles** checkbox
61. Push the **Open** button

Post-Processing the Results (Mode Shape of all Design Cycles)

62. Push the **Oscillate** button
63. From the **Deform Mesh**, select different modes for the last design cycle

    Now, you should be able to see simultaneously the mode and the final topology answers.

64. Push the **Filled Contours** button
65. From the **Color Mesh**, select different modes

    Now, you should be able to see the **XYZ Magnitud**e of deformation colored. Grids that move more are in red, grids that move less are in blue.

66. Push the **Up** button

Quit Design Studio

67. From the main menu bar, select **File** → **Quit**
68. Push the **Don’t Save** button

Study the Output File

69. Start any text editor
70. In a text editor load the Genesis data file: `TPDSG008_2_dsg.out`
71. Study briefly the file

72. Using the output file complete the following table:

Buckling Load Factor Number	Buckling Load Factor Value Original Reference Solution (1)	Buckling Load Factor Value Topology Reference Solution (2)	Buckling Load Factor Value (3)
1	3.3	2.01	
2	29.6	17.9	
3	82.8	49.1	
4	163.3	78.5	
5	202.2	79.0	
Mass	2.87	2.12	

(1) Result from baseline buckling analysis
(2) Result from baseline topology optimization
(3) Result from your run, `TPDSG008_2_dsg.out`

The values in this table and the ones you should get should be very similar.

73. What is the lowest calculated value of lama in your run?

74. Are all the constraints satisfied?

75. Why doesn’t Genesis print all constraints?

Hint: Constraint Screening.

76. Would the current design buckle?

77. If you scale the applied loads, what scale factor will make the structure buckle?

78. What is the mass of the optimized structure?

Reference answer: 2.12

79. Compare with the original structure

Reference answer: Change=(2.12-2.87)/2.87*100=-26%

80. Why the mass change is not 35%?

Answer: Mass fraction is applied to the designable parts only

3.8.2 Part 2

The purpose of this part of the example is to learn how to create a structure containing only the most important elements discarding the less important ones. Also this part of the example will be used to verify the topology optimization results of the last design cycle obtained in Part 1.

If you do not have the `TPDSG008_2.dat` file generated in part 2, copy the file `TPDSG008_2_ref.dat` to `TPDSG008_2.dat`.

When you finish this example, you should have created a file named: `TPDSG008_3.dat.`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG008_2.dat`

Creating a new Group

You will create a new PSHELL Group. This PSHELL will be used to group together the quadrilateral elements that you select as the most important to keep.

3. Select the **Display** tab
4. Push the **Manage Groups** button
5. Select the existing **PSHELL** 3 group
6. From the Edit menu toolbar, select the **Copy Group** button
7. From the Edit menu toolbar, select the **Paste Group** button
8. Select the PSHELL that you just created
9. Push the **Modify Group** button from the Edit Menu toolbar
10. For Name enter `FinalTopology`
11. Select Yellow for color
12. Push the **Finish** button
13. Push the **Up** button

Optimize the Structure

If you completed Part 1, you can skip this step.

14. From the main menu bar, select **Genesis** → **Optimize**

Import the Post-Processing Files (Densities)

15. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
16. Select the `TPDSG008_2_dsgDENSxx.pch` file, where `xx` is the last design cycle
17. Push the **Open** button

Post-Processing the Results (Density Values)

18. Select the **Post** tab
19. Push the **Deform Mesh/Color Mesh** button
20. Select the Topology Result for the last design cycle
21. Push the **Options...** button
22. Push the **Hide Elements With No Value** button
23. Slide the lower cutoff slidebar to mask out element with low density values
24. Push the **Close** button

Moving Elements with High Density to a new Group

<u>Without</u> pushing the **Up** button in the **Deform Mesh/Color Mesh** section. We will move the elements to the `FinalTopology` group.

25. Select the **Analysis** tab
26. From the category chooser, select **Elements**
27. Push the **Select All** button

    Verify that there are several (700 or so) elements in the selection.
28. Push the **Modify Elements** button from the Edit Menu toolbar
29. Push **Next>**
30. Select the `FinalTopology` group
31. Push the **Finish** button

Clear the Density Results

32. Select the **Post** tab
33. Push the **Up** button

Delete the Topology Data

Now we will delete all the topology data. This prevents re-evaluating the densities and unnecessary calculations.

34. Select the **Topology** tab
35. From the category chooser, select **Topology Regions**
36. Push the **Select** button
37. Push the **Delete Topology Design** button

    Check that there are no hammer icons in the list.
38. From the category chooser, select **Topology Objectives**
39. Select the objective in the list
40. Push the **Delete Topology Objective** button from the Edit Menu toolbar
41. From the category chooser, select **Topology Constraints**
42. Select all the constraints in the list
43. Push the **Delete Topology Constraint** button from the Edit Menu toolbar

Verify that there is no Topology Data

44. From the category chooser, select **Quick Setup Trails**

    Verify that there are 0 **Topology Regions Defined**, 0 **Objectives Defined** and 0 **Constraints Defined.**

Deleting the Element with Low Density

Now we will delete the elements that are less important.

45. Select the **Display** tab
46. Push the **Show/Hide Groups** button
47. **Hide** all groups except PSHELL 3
48. Push the **Up** button
49. Select the **Analysis** tab
50. Make sure that the category chooser is set to **Elements**
51. Push the **Select All** button to select all the visible quadrilateral elements
52. Push the **Delete Elements** button from the Edit Menu toolbar

    Now, all elements with low density have been deleted.

53. From the category chooser, select **Grids**
54. Push the **Select All** button
55. Push the **Delete Grids** button from the Edit Menu toolbar

    Now, all grids associated with the deleted elements have been deleted

56. Select the **Display** tab
57. Push the **Show/Hide Groups** button
58. Push the **Show All** button
59. Push the **Up** button
60. Push the **Manage Groups** button
61. Select only the PSHELL 3 group
62. Push the **Delete Group** button from the Edit Menu toolbar

    Now, the unused group Property ID has been removed

Updating the Boundary Conditions

Since we just deleted grids in the boundary conditions, we will need to modify or to re-create the boundary conditions. Next we will re-create them, but first we will delete the existing ones.

63. Select the **Analysis** tab
64. From the category chooser, select **Grid Component Sets**
65. Select the existing set
66. Observe the location of the boundary conditions as they have to be recreated on the updated model.
67. Push the **Delete Grid-Component Set** button from the Edit Menu toolbar
68. Push the **New Grid-Component Set** button in the Edit Menu toolbar
69. For Name: enter `11_Nodes`
70. Push **Next>**
71. Select the same end as the original boundary conditions were
72. Push **Set Components** button

    Check the bottom of the panel. It should say there are 66 dof.

73. Push the **Finish** button

Updating the Loadcase

We will now update the loadcase to use the new boundary conditions.

74. From the category chooser of the **Analysis** tab, choose **Loadcases**
75. Select the static loadcase
76. Push the **Modify Loadcase** button from the Edit Menu toolbar
77. Push **Next>**
78. For SPC: Select the set with the name `11_Nodes`
79. Push the **Finish** button

Save the Design Studio Database File

80. From the main menu bar, select **File** → **Save As...**
81. Enter `TPDSG008_3` as the Filename and push **Save (**as a Design Studio File**)**

Analyze the Structure Using Genesis

82. From the main menu bar, select **Genesis** → **Single Analysis**

    Study the **Genesis Console Output**; when done, push the **Close** button.

Clean the Post-Processing Results

83. Select the **Post** tab
84. Push the **Manage Result Dataset** button
85. From the main menu bar, select **Edit** → **Select All**
86. Push the **Delete Result Set** button from the Edit Menu toolbar
87. Push the **Up** button

Import the Post-Processing Files

88. Import the `TPDSG008_3_dsg00.pch` file

Post-Processing the Results (Mode Shape)

89. Using similar steps described above post-process the modes

Export the Input File

90. From the main menu bar, select **File** → **Export** → **Input Data...**
91. Enter `TPDSG008_3.dat`
92. Push the **Save** button

Quit Design Studio

93. From the main menu bar, select **File** → **Quit**

Study the Output File

If you did not open the output file above, now you should do that.

94. In a text editor load the Genesis data file: `TPDSG008_3_dsg.out`
95. Study briefly the file
96. Using the output file, complete the following table:

Buckling Load Factor Number	Buckling Load Factor Value Original Reference Solution (1)	Buckling Load Factor Value Topology Reference Solution (2)	Buckling Load Factor Value Final Reference Solution (3)	Buckling Load Factor Value (4)
1	3.27	2.01	2.26	
2	29.58	17.9	20.17	
3	82.76	49.1	55.10	
4	163.32	78.5	76.89	
5	202.16	79.0	81.37	
Mass	2.87	2.12	2.14	

(1) Result from baseline buckling analysis
(2) Result from baseline topology optimization
(3) Result from baseline final reference analysis
(4) Result from your run, `TPDSG008_3_dsg.out`

97. What is the lowest calculated value of lama in the final results?
98. Are all the constraints satisfied?
99. Would the current design buckle?
100. If you scale the applied loads, what value will make the structure buckle?

101. What is the mass of the optimized structure?

   Reference answer: 2.14.

102. Compare with the original structure

   Reference answer: Change=(2.14-2.87)/2.87*100=-25%

103. Compare with the topology results

   Reference answer: Change=(2.14-2.12)/2.12*100=+1%

104. Why are the topology results different from the final results?

3.9 Optimization of Reinforcement Member With Random Response

Introduction

The purpose of this exercise is to demonstrate how to set up a topology optimization problem with random response. The loading, boundary conditions, and random response bulk data entries are present in the provided file `TPDSG009_1.dat`. Details of these bulk data entries may be found in the GENESIS Analysis Manual.

Problem Statement:

A plate is reinforced with shell element ribs and is subjected to vertical cyclic tip loads in a random modal frequency response load case. The design objective is to minimize the tip displacement subject to the mass fraction $\leq$ 0.20 for the designable region comprised of the added ribs on the entire surface of the plate.

The structure is fixed at one end and loaded at the opposite end.

The problem is divided into two parts that will help you learn the following tasks:

1) How to set up a topology optimization problem with random response. The Anticheckerboard filter is set to the non-default value of OFF in this part

2) In this part, the Anticheckerboard filter is set to its default value of ON to study its effect on the topology results

The two parts of this exercise should be done sequentially. However, if you want to skip the first part, the necessary files for part two are included.

Example ID

`TPDSG009`

Files Used in This problem

A list of the key files is provided and the ones that will be created during this exercise is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification

Part	File Name	Description
1	`TPDSG009_1.dat`	Provided: Contains the finite element mesh and the random response loadcase.

1,2	TPDSG009_2.dat	Generated in part 1 and used as a starting file for part 2. This file is the same as the TPDSG009_1.dat file along with topology optimization data.
1,2	TPDSG009_2_ref.dat	Provided: Reference file same as TPDSG009_2.dat file
2	TPDSG009_3.dat	Generated in part 2. This file is similar to TPDSG009_2.dat but with the Anticheckerboard filter turned ON
2	TPDSG009_3_ref.dat	Provided: Reference file same as TPDSG009_3.dat file

3.9.1 Part 1

The purpose of this part of the example is to learn how to perform a topology optimization with random response.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG009_1.dat`

Create the Random Modal Frequency Response Load Case

3. Select the **Analysis** tab
4. From the category chooser, select **Loadcases**
5. Push the **New Loadcase** button in the Edit Menu toolbar
6. Change the name to: `Modal_Freq_Resp`
7. Push the **Modal Frequency Response** radio button
8. Push the **Next>** button
9. For **SPC:**, select `G-C Set 1`
10. Push the **Next>** button
11. For **Eigenvalue Method**, select `Method 1`
12. Push the **Next>** button
13. For **Dynamic Load Set:**, select `DLoad Set 10`
14. For **Loading Frequency Set:**, select `Frequency Set 40`
15. For **Modes Loadcase:**, select `Modes`
16. For **Modal Damping:**, select `Damping 20`
17. For **Random Set:**, select `RANDOM 30`
18. Push the **Next>** button
19. For **Displacement:**, select `Post` and `Grid Set 1`
20. Push the **Finish** button

Create the Designable Region

21. Select the **Topology** tab

22. From the category chooser, select **Topology Regions**
23. Select the ribs

    You can either click on the part on the screen or select PSHELL 2 from the list.
24. Push the **Modify Topology Design** button
25. Use `0.20` for the initial mass fraction
26. Push the **Finish** button to complete the design data for the topology region

    Verify that there is the hammer icon next to the PSHELL 2.

Defining the Design Objective

27. From the category chooser, select **Topology Objectives**
28. Push the **New Topology Objective** button in the Edit Menu toolbar
29. Select **Dynamic Displacement** response type, and
30. Select **Random RMS (RMSDISP)** from the pull down menu
31. Make sure the **Min** button is selected for the Objective Definition
32. Push **Next>**
33. For the grid selection type in Grid ID: `133`, and push the **Add** button

    Verify that `1 grid selected` is shown at the bottom of the viewport
34. Push the **Translation 3** radio button
35. Push the **Next>** button
36. Select **Modal_Freq_Resp** loadcase
37. Push the **Finish** button

Defining the Mass Constraints

38. From the category chooser, select **Topology Constraints**
39. Push the **New Topology Constraint** button in the Edit Menu toolbar
40. Enter Name `MassFr`
41. Select the **Mass Fraction** response
42. Enter `0.20` for the **Upper Bound**
43. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Set Output Control for the First and Last Design Cycles Only

44. From the main menu bar, select **Genesis** → **Options**
45. Select the **Output Control** tab
46. Check the **Analysis Output** box, and select **First & Last**
47. Check the **Design Output** box, and select **First & Last**
48. Push the **Apply** button

Set the Maximum Number of Design Cycles

49. From the main menu bar, select **Genesis** → **Options**
50. Select the **Design Control** tab
51. Check the **Maximum Design Cycles** box, and set to `40`
52. Push the **Apply** button

Set the Topology Optimization Minimum Move Limit

53. From the main menu bar, select **Genesis** → **Options**
54. Select the **Design Control** tab
55. Push the **Advanced** button
56. Select the **Move Limits** tab
57. Check the **Minimum Topology (DTMIN)** box, and set to **0.07**
58. Push the **Close** button
59. Push the **Apply** button

Set the Topology Optimization Anticheckerboard Filter

For rib topology optimization the DOPT parameter **FILTER** should be set to the non-default value of **Off** to turn off checkerboard filtering. If checkerboard filtering is not turned off, then the results will tend to make patches rather than directed ribs.

60. From the main menu bar, select **Genesis** → **Options**
61. Select the **Design Control** tab
62. Push the **Advanced** button
63. Select the **Misc.** tab

64. Check the **Anticheckerboard Filter (FILTER)** box, and select **Off** from the pull down menu
65. Push the **Close** button
66. Push the **Apply** button

Save the Design Studio Database File.

67. From the main menu bar, select **File** → **Save As...**
68. Enter `TPDSG009_2` as the Filename and push **Save (**as a Design Studio File**)**

Export the Input File

69. From the main menu bar, select **File** → **Export** → **Input Data...**
70. Enter `TPDSG009_2.dat`
71. Push the **Save** button

Optimize the Structure

72. From the main menu bar, select **Genesis** → **Optimize**

    Notice that the design objective went from **0.150** (at design cycle 0) to **0.116** (at the final design cycle. This means the maximum dynamic tip response has decreased by about 23% with the addition of the rib reinforcement.

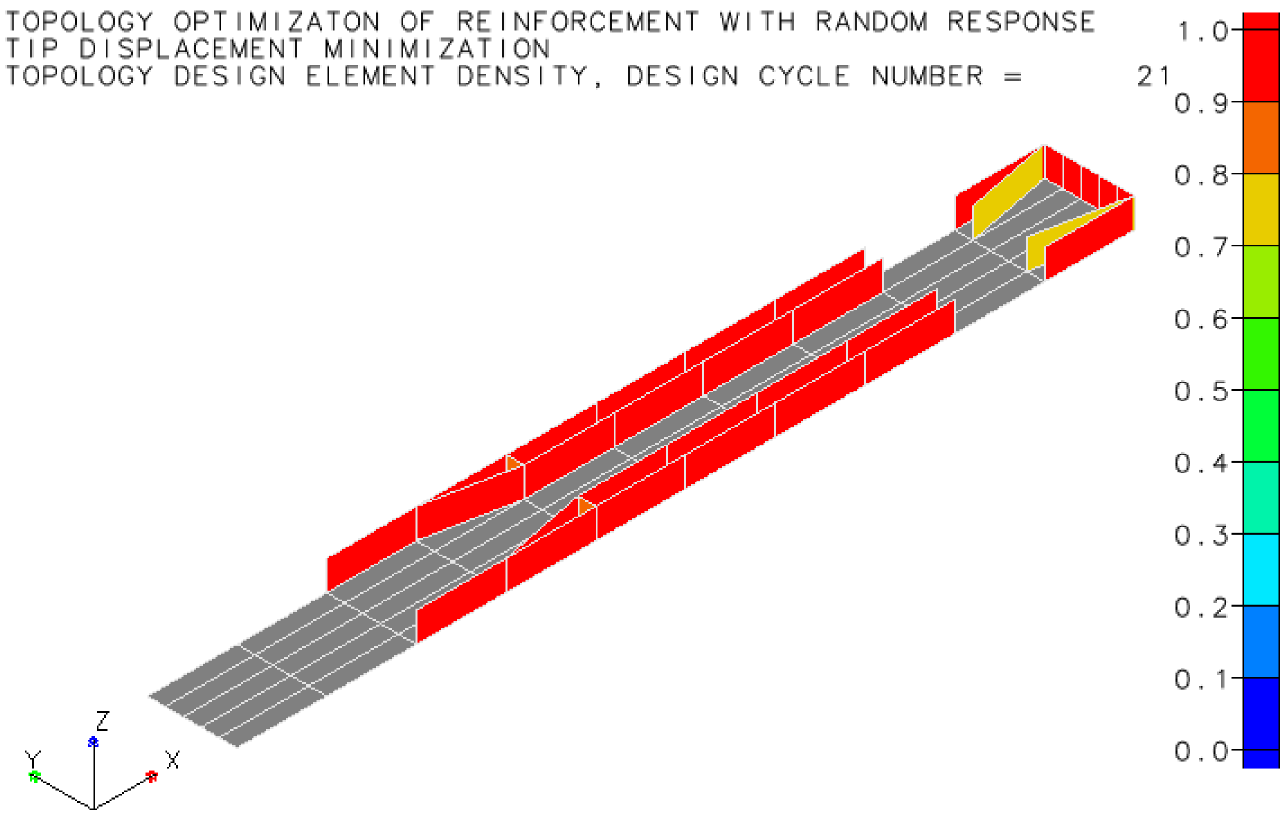
Import the Post-Processing Files (Densities)

73. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
74. Select the `TPDSG009_2_dsgDENS00.pch` file and check the **Import Similar Results for All Design Cycles** checkbox
75. Push the **Open** button

Post-Processing the Results (Densities)

76. Select the **Post** tab
77. Push the **Deform Mesh/Color Mesh** button
78. Push the **Filled Elements** buttons
79. Select the **Topology Results** for the last cycle

    With the anticheckerboard FILTER=OFF (Objective = 0.116) the optimum rail configuration looks something like the picture below:



Import the Post-Processing File (Displacements)

80. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
81. Select the `TPDSG009_2_dsg00.pch` file and check the **Import Similar Results for All Design Cycles** checkbox
82. Push the **Open** button

Post-Processing the Results (Create a Frequency Response Plot)

83. Select the **Post** tab
84. Push the **Freq. Resp. Plot** button
85. Push the **New Freq. Resp. Plot** button in the Edit Menu toolbar
86. Push the **Magnitude + Phase** radio button
87. Push **Next>**
88. Push the **+** button
89. Choose both the first and last design cycles to be plotted

90. Push **Next>**
91. Choose **Grid 133** to be plotted
92. Push the **Translation 3** radio button
93. Push **Next>**

The Z displacement frequency response plot for grid 133 looks like:

94. Push the **Finish** button

How to Save a Frequency Response Plot

95. Right click on the graph
96. Push the **Save Image...** button
97. Enter `G133Z_Disp`

Quit Design Studio

98. From the main menu bar, select **File** → **Quit**
99. Push the **Don't Save** button

3.9.2 Part 2

The purpose of this part of the example is see how the default anticheckerboard filter (FILTER = ON) affects the topology optimization results.

If you do not have the `TPDSG009_2.dat` file generated in part 1, copy the file `TPDSG009_2_ref.dat` to `TPDSG009_2.dat`.

When you finish this example, you should have created a file named: `TPDSG009_3.dat`.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG009_2.dat`.

Set the Topology Optimization Anticheckerboard Filter

3. From the main menu bar, select **Genesis** → **Options**
4. Select the **Design Control** tab
5. Push the **Advanced** button
6. Select the **Misc.** tab
7. Check the **Anticheckerboard Filter (FILTER)** box, and select **On** from the pull down menu
8. Push the **Close** button
9. Push the **Apply** button

Save the Design Studio Database File.

10. From the main menu bar, select **File** → **Save As...**
11. Enter `TPDSG009_3` as the Filename and push **Save (**as a Design Studio File**)**

Export the Input File

12. From the main menu bar, select **File** → **Export** → **Input Data...**
13. Enter `TPDSG009_3.dat`
14. Push the **Save** button

Optimize the Structure

15. From the main menu bar, select **Genesis** → **Optimize**

    Notice that the design objective went from **0.150** (at design cycle 0) to **0.122** (at the final design cycle. This means the maximum dynamic tip response has decreased by about 19% with the addition of the rib reinforcement.

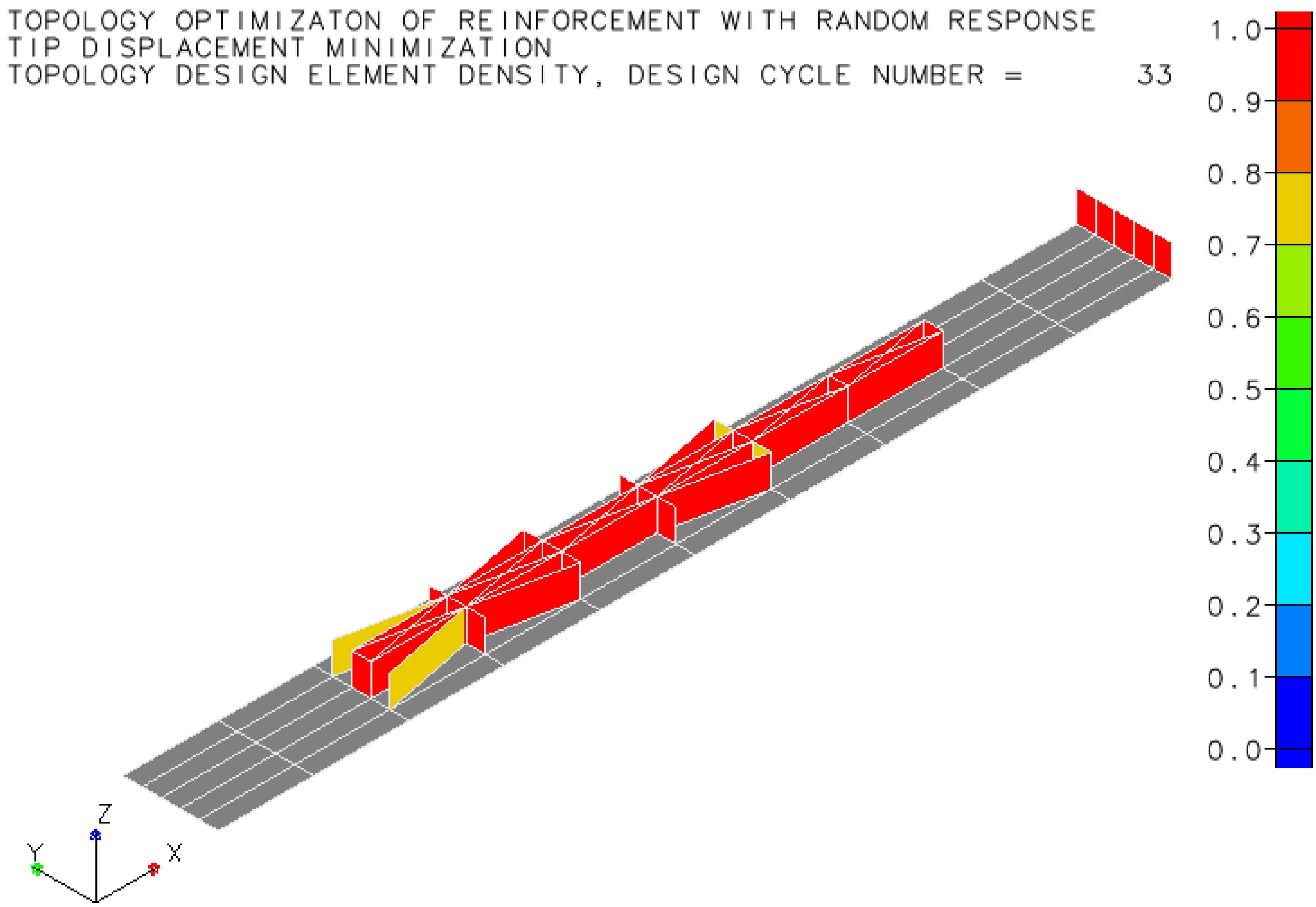
Import the Post-Processing Files (Densities)

16. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
17. Select the `TPDSG009_3_dsgDENS00.pch` file and check the **Import Similar Results for All Design Cycles** checkbox
18. Push the **Open** button

Post-Processing the Results (Densities)

19. Select the **Post** tab
20. Push the **Deform Mesh/Color Mesh** button
21. Push the **Filled Elements** buttons
22. Select the **Topology Results** for the last cycle

With the anticheckerboard FILTER=ON (Objective = 0.122) the optimum rail configuration looks something like the picture below:



Import the Post-Processing File (Displacements)

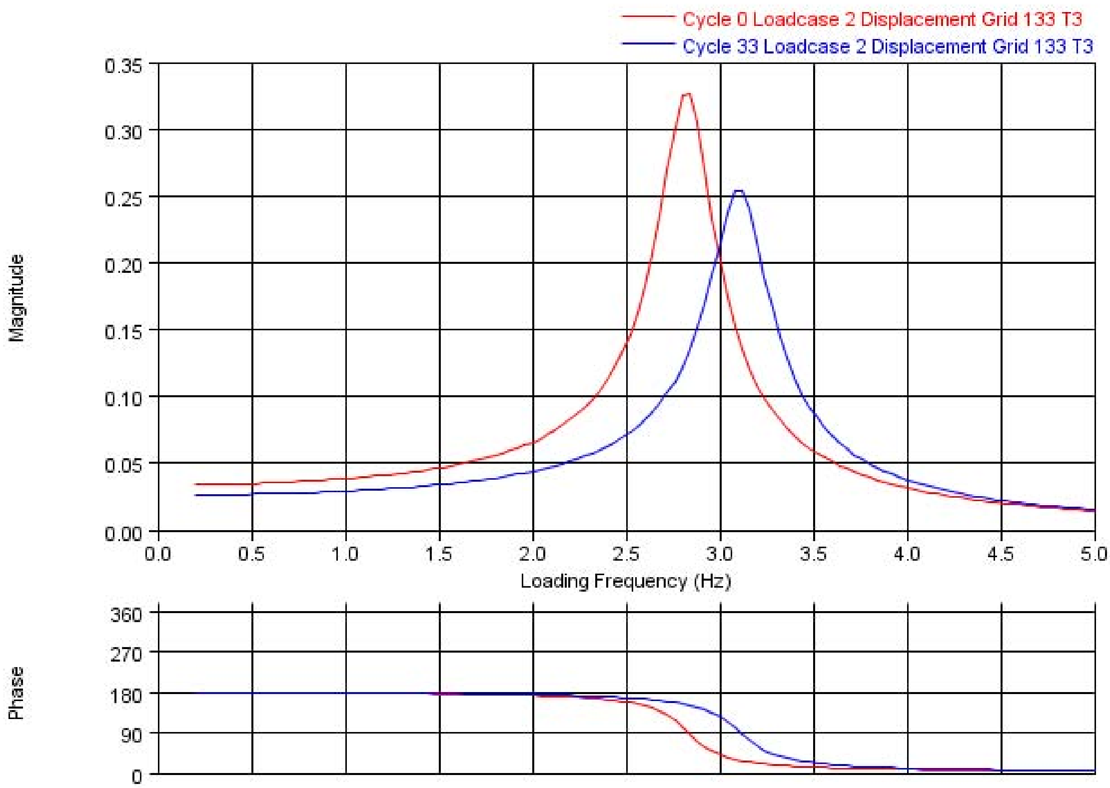
23. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
24. Select the `TPDSG009_3_dsg00.pch` file and check the **Import Similar Results for All Design Cycles** checkbox
25. Push the **Open** button

Post-Processing the Results (Create a Frequency Response Plot)

26. Select the **Post** tab
27. Push the **Freq. Resp. Plot** button
28. Push the **New Freq. Resp. Plot** button in the Edit Menu toolbar
29. Push the **Magnitude + Phase** radio button

30. Push **Next>**
31. Push the **+** button
32. Choose both the first and last design cycles to be plotted
33. Push **Next>**
34. Choose **Grid 133** to be plotted
35. Push the **Translation 3** radio button
36. Push **Next>**

The Z displacement frequency response plot for grid 133 looks like:



Compare these results (with FILTER = ON) to the results in the previous section (with FILTER = OFF). When the anticheckerboard filter is turned off there are less restrictions for keeping the ribs collectively during the topology optimization. This is why the objective function is better for the case with FILTER = OFF.

Quit Design Studio

37. From the main menu bar, select **File** → **Quit**
38. Push the **Don't Save** button

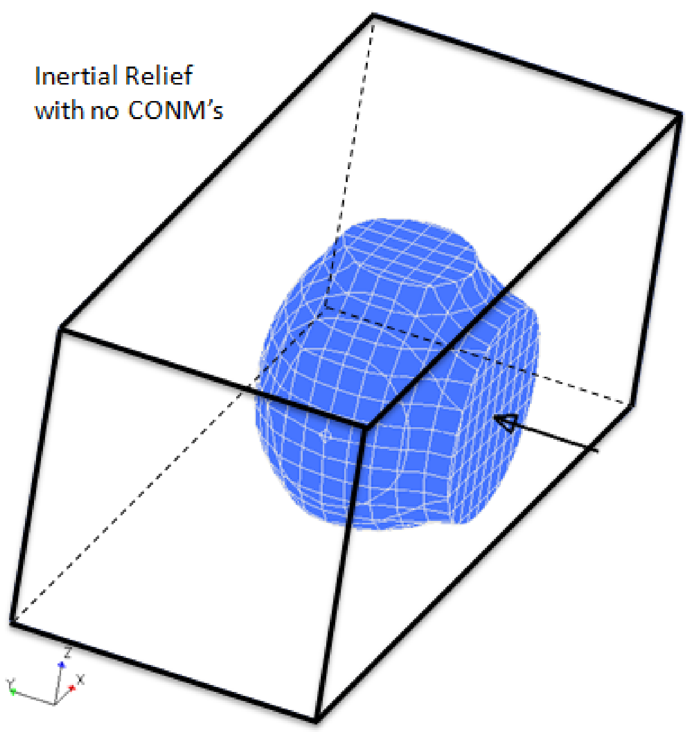
3.10 Differences in Topology Optimization Using Single Point Constraints and Inertial Relief

Introduction

The purpose of this exercise is to demonstrate how inertial relief and single point constraint (SPC) boundary conditions affect the topology optimization of a support.

During topology optimization an applied load on a support that uses SPC's to react the load will produce structure with the most efficient geometry for the specified mass retained. However, impact loading that is modeled as a static equivalent load in topology should not necessarily be used on a structure that is restrained with SPC's. Any loading on an unrestrained structure is reacted by the structures mass. For topology optimization the inertial relief method of static loading can be applied to a free structure without artificially restraining it.

There is an issue concerning topology blank uniform mass versus concentrated masses when using inertial relief static solutions. Topology optimization strives to produce a geometry that yields the stiffest collection of elements to resist the applied loading. For a single load on an unrestrained brick the topology result using inertial relief clusters the material in a sphere like shape opposing the load. This is the stiffest geometry under the circumstances and is shown below.



A blob of elements isn't the most elegant engineering design for our support. Inserting large concentrated masses (CONM's) at the four corners opposite the applied load provides inertial reactive loads that design the support without fixing the corners like SPC's would.

Problem Statement:

> A brick comprised of solid elements has a single load applied to one side. The load is offset from the center of the face so it alone would not produce a symmetric topology design. Manufacturing constraints will be used to force topology to produce a symmetric support. The design objective is to minimize the strain energy subject to the mass fraction <= 0.10 for the designable region comprised of the entire brick.

The problem is divided into two parts that will demonstrate topology optimization using static analysis and inertial relief static analysis:

1) How to set up a topology optimization problem on the brick with one offset static load and 4 corners of the brick restrained with SPC's

2) How to set up the same topology problem with inertial relief and concentrated masses

The two parts of this exercise should be done sequentially. Separate files for each part are provided.

Example ID

`TPDSG010`

Files Used in This problem

A list of the key files is provided and the ones that will be created during this exercise is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification

Part	File Name	Description
1	`TPDSG010_1.dat`	Provided for part 1: Contains the finite element mesh with loading and single point constraints.
1	`TPDSG010_1_ref.dat`	Reference file for part 1: Contains all the data in `TPDSG010_1.dat` along with the topology data created in part 1.
2	`TPDSG010_2.dat`	Provided for part 2: Contains the finite element mesh with concentrated masses and topology optimization data.
2	`TPDSG010_2_ref.dat`	Reference file for part 2: Contains all the data in `TPDSG010_2.dat` along with the data created in part 2.

3.10.1 Part 1

The purpose of this part of the example is to learn how to perform a topology optimization on the brick with SPC's and symmetry constraints.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG010_1.dat.`

Create the Static Load Case

3. Select the **Analysis** tab
4. From the category chooser, select **Loadcases**
5. Select **Loadcase 1**, and push the **Modify Loadcase** button from the Edit Menu toolbar
6. Change the name to: `Statics with SPC's`
7. Push the **Next>** button
8. For **SPC:**, select `G-C Set 2`
9. Push the **Next>** button
10. For **Load Set:**, select `Static Load 2`
11. Push the **Next>** button
12. For **Displacement:**, select `Post` and `All`
13. Push the **Finish** button

    Notice the force applied to one face of the brick is not dead center (it is offset), and the translational SPC's at the 4 corners opposite the applied face.

Create the Designable Region

14. Select the **Topology** tab
15. From the category chooser, select **Topology Regions**
16. Select the brick

    You can either click on the part on the screen or select PSHELL 6 from the list.
17. Push the **Modify Topology Design** button
18. Use `0.10` for the initial mass fraction

19. Push the **Next>** button
20. For **Coordinate System:** push the **Change** button
21. Select `Symmetry`, and push the **Next>** button
22. For **Constraint 1:**, select `MYZ` from the pull down menu
23. For **Constraint 2:**, select `MXY` from the pull down menu
24. Push the **Finish** button to complete the design data for the topology region

    Verify that there is the hammer icon next to the PSHELL 6.

Defining the Design Objective

25. Select the **Topology** tab
26. From the category chooser, select **Topology Objectives**
27. Push the **New Topology Objective** button in the Edit Menu toolbar
28. Select **Strain Energy** response type
29. Make sure the **Min** button is selected for the Objective Definition
30. Push **Next>**
31. Select `Statics with SPC's` loadcase
32. Push the **Finish** button

Defining the Mass Constraints

33. From the category chooser, select **Topology Constraints**
34. Push the **New Topology Constraint** button in the Edit Menu toolbar
35. Enter Name `MassFr`
36. Select the **Mass Fraction** response
37. Enter `0.10` for the **Upper Bound**
38. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Set the Maximum Number of Design Cycles

39. From the main menu bar, select **Genesis** → **Options**
40. Select the **Design Control** tab
41. Check the **Maximum Design Cycles** box, and set to `30`

42. Push the **Apply** button

Set the Topology Optimization Minimum Move Limit

43. From the main menu bar, select **Genesis** → **Options**
44. Select the **Design Control** tab
45. Push the **Advanced** button
46. Select the **Move Limits** tab
47. Check the **Minimum Topology (DTMIN)** box, and set to `0.05`
48. Push the **Close** button
49. Push the **Apply** button

Optimize the Structure

50. From the main menu bar, select **Genesis** → **Optimize**

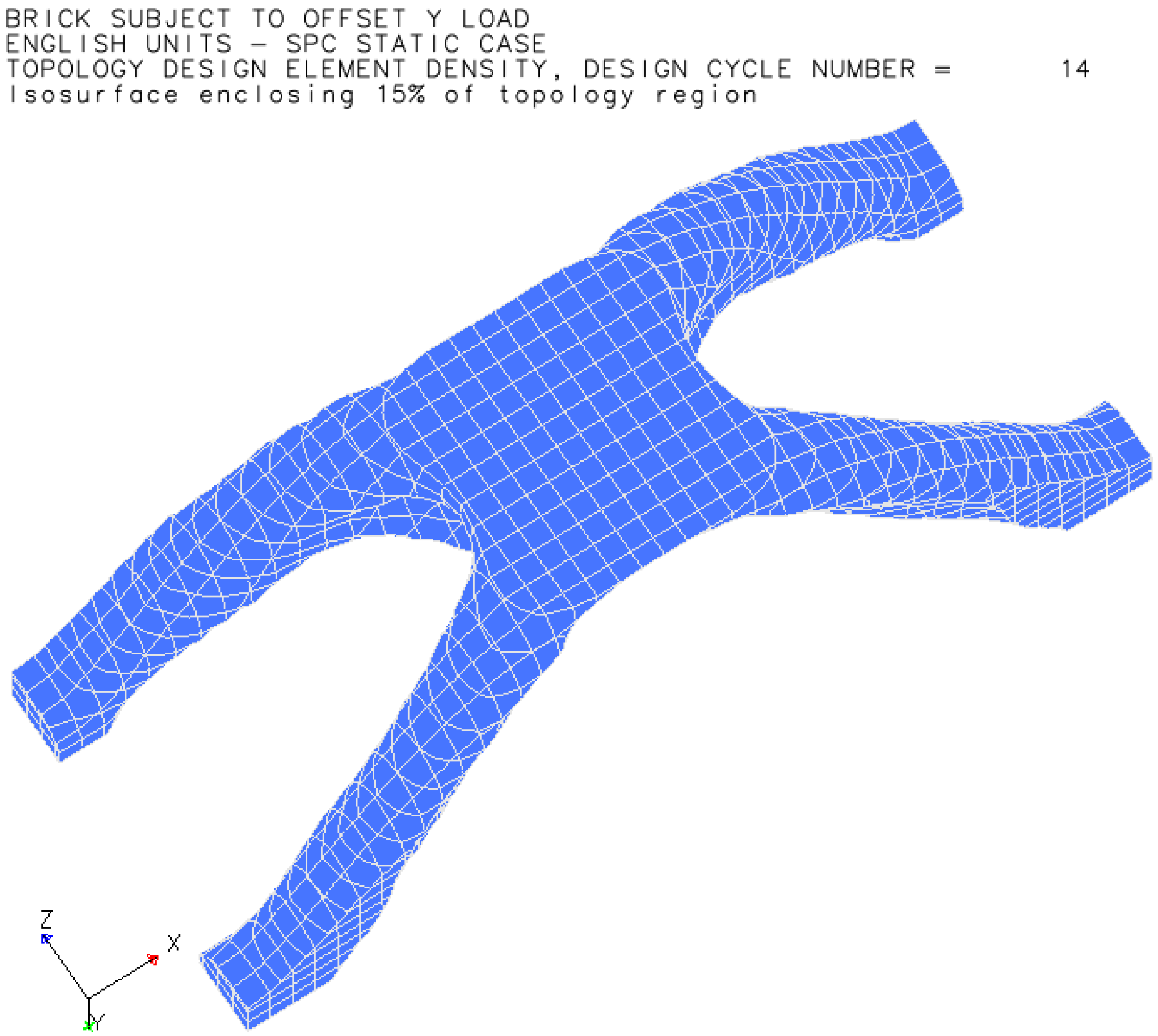
Import the Post-Processing Files (Densities)

51. From the **Genesis Console Output** window, select the **Import Post...** button
52. Select the `TPDSG010_1_dsgDENSxx.pch` file (where `xx` is the number of the final design cycle)
53. Push the **Import** button
54. In the **Genesis Console Output** window, push the **Close** button

Post-Processing the Results (Densities)

55. Select the **Post** tab
56. Push the **Density Isosurface** button
57. Select the **Topology Result** for the last cycle
58. Push the **Options...** button
59. Select the **Show Topology Region** checkbox to view the initial topology region

The optimization results will look like:



Quit Design Studio

60. From the main menu bar, select **File** → **Quit**
61. Push the **Save** button

    This action saves the file as `TPDSG010_1.dsg`

3.10.2 Part 2

This part of the example shows how the same brick problem in part one can be set up to run topology with an inertial relief loadcase.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG010_2.dat`

   This data file is the result of part 1 with two parts edited in. The subtitle has been changed and 4 CONM2 masses have been included at the SPC locations (SPC's will be removed in this task).

Modify the Static Loadcase for Inertial Relief

3. Select the **Analysis** tab
4. From the category chooser, select **Loadcases**,
5. Select **Statics with SPC's**, and push the **Modify Loadcase** button from the Edit Menu toolbar
6. Change the name to: `IR Statics with CONM's`
7. Push the **Next>** button
8. For **SPC:**, Select `None`
9. For **Inertia Relief Support Set:**, Select `Automatic`
10. Push the **Finish** button

Optimize the Structure

11. From the main menu bar, select **Genesis** → **Optimize**

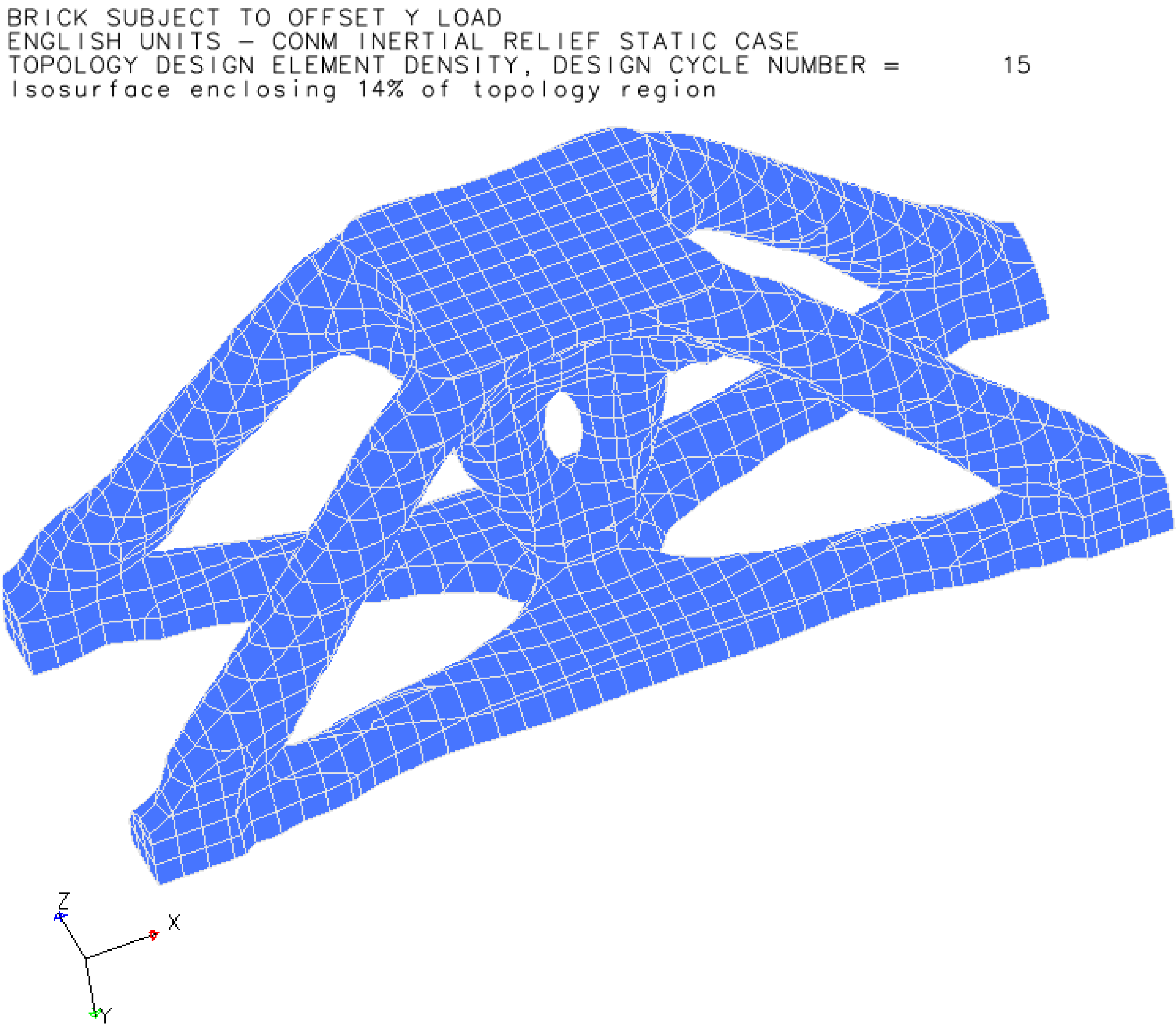
Import the Post-Processing Files (Densities)

12. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
13. Select the `TPDSG010_2_dsgDENSxx.pch` file (where `xx` is the number of the final design cycle)
14. Push the **Open** button

Post-Processing the Results (Densities)

15. Select the **Post** tab
16. Push the **Density Isosurface** button
17. Select the **Topology Result** for the last cycle

The optimization results will look like



The optimized support bracket with inertial relief looks very different than the results from part 1. Part 1 was run with SPC's at the 4 corners such that relative displacements between all 4 points is zero (i.e. they are welded in place).

The inertial relief case in part 2 was run on an unrestrained brick with concentrated masses at the 4 corners which provide inertial reactions. All four corners are free to move relative to each other. Notice that a center pedestal in the Z direction has formed along with the extra structure in the X-Y plane connecting the four corners. Topology puts this structure in this location to help make the 4 corners and the force application point as rigid as possible.

The inertial relief statics with concentrated mass method lends itself well to topology design of unrestrained automobile bodies subject to impact loads. Concentrated masses of all components like wheels, suspension, engine, transmission, batteries, fuel tanks, etc. can be accounted for in an inertial relief load case without artificial SPC restraints.

Quit Design Studio

18. From the main menu bar, select **File** → **Quit**
19. Push the **Don’t Save** button

3.11 Solid Block Subject to Torsional Loads with Stampable Sheet Constraints

Introduction

The purpose of this exercise is to demonstrate how to impose a stampable sheet fabrication constraint during topology optimization. The desired structure will be buildable with 2 stamped sheets of a specified thickness.

Problem Statement:

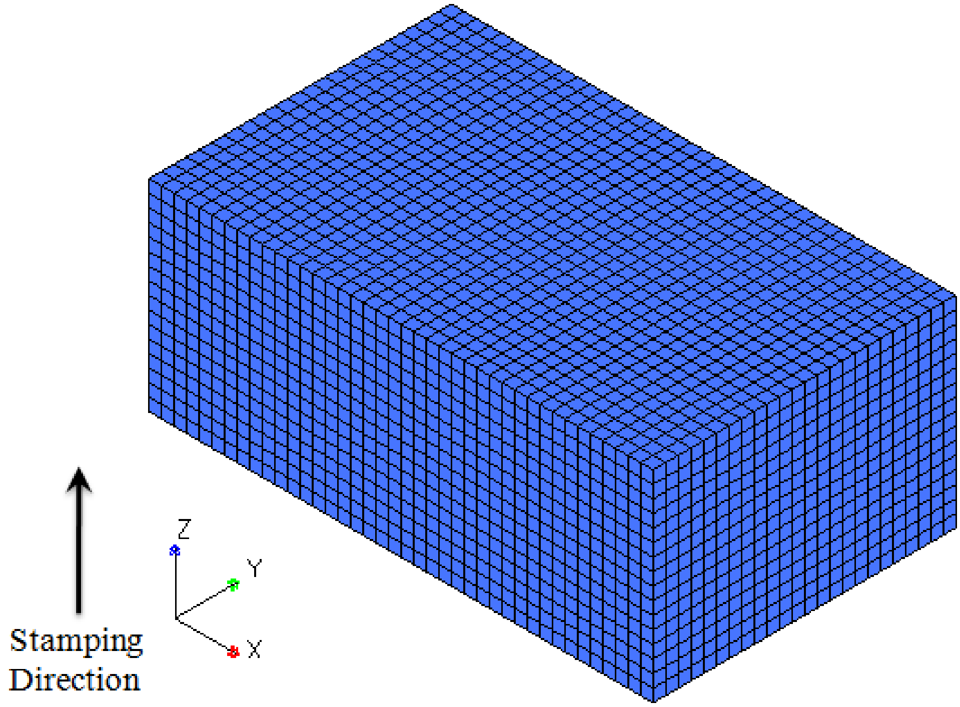
A brick shaped design space is comprised of solid elements that has two torsional loadcases applied to it. The first loadcase has a force couple applied to one end while the opposite end is fixed, and the second loadcase has both ends fixed while a force couple is applied at the center. Symmetry constraints are defined about a coordinate system at the center of the brick to enforce mirror symmetry about the XZ plane, and about the YZ plane (see picture below). A manufacturing constraint will be used to force topology to produce two stampable sheets with a separation plane oriented normal to the specified stamping direction. The stamping direction for this case is the Z direction as shown below. The separation plane for this case locates itself at the center of the brick in the XY plane normal to the Z axis. Depending on the loadcases used in Genesis the separation plane could be located anywhere along the Z axis during topology optimization. The design objective is to minimize the strain energy subject to a Mass Fraction <= 0.25 for the designable region comprised of the entire brick that will automatically be segregated into two stampable sheets.

The problem is divided into two parts that will demonstrate the stampable sheet fabrication constraint with, and without allowing punch holes:

1) How to set up the stampable sheet topology optimization problem on the brick without punch holes.

2) How to set up the stampable sheet topology optimization problem on the brick with punch holes.

The two parts of this exercise should be done sequentially. Separate files for each part are provided.



Example ID

TPDSG011

Files Used in This problem

A list of the key files is provided and the ones that will be created during this exercise is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification

Part	File Name	Description
1	TPDSG011_1.dat	Provided: Contains the finite element mesh and the static loadcase.
1,2	TPDSG011_1_dsg.dat	Generated in part 1 and the starting file for part 2. This file is the same as the TPDSG011_1.dat file along with topology optimization data.
1,2	TPDSG011_1_ref.dat	Provided: Reference file same as TPDSG011_1_dsg.dat file
2	TPDSG011_2.dat	Exported at the end of part 1. Used as starting file for part 2. This file is similar to TPDSG011_1_dsg.dat
2	TPDSG011_2_ref.dat	Provided: Reference file includes the data in TPDSG011_2.dat file along with changes made in part 2

3.11.1 Part 1

The purpose of this part of the example is to learn how to perform a topology optimization on the brick with stampable sheet fabrication constraints, and with through holes not allowed.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG011_1.dat`

Examine the Static Load Cases

3. Select the **Analysis** tab
4. From the category chooser, select **Loadcases**

   Click on the loadcases separately to examine the applied loads.

Create the Designable Region

5. Select the **Topology** tab
6. From the category chooser, select **Topology Regions**
7. Select the brick

   You can either click on the part on the screen or select PSOLID 6 from the list.
8. Push the **Modify Topology Design** button
9. Use `0.25` for the initial mass fraction
10. Push the **Next>** button
11. For **Coordinate System:** push the **Change** button
12. Select `Brick_Center`, and push the **Next>** button
13. For **Constraint 1:**, select `S2Z` from the pull down menu
14. For **Constraint 2:**, select `MYZ` from the pull down menu
15. For **Constraint 3:**, select `MZX` from the pull down menu
16. For **Minimum Size:**, enter `0.5`
17. For **Spread Fraction:**, enter `0.0`
18. For **Sheet Thickness:**, enter `0.5`
19. For **Allow Through Holes:**, click the **No** radio button

20. For **Void Value:**, enter `0.001`
21. For **Start Offset:**, enter `0.0`
22. Push the **Finish** button to complete the design data for the topology region

    Verify that there is the hammer icon next to the PSOLID 6.

Defining the Design Objective

23. Select the **Topology** tab
24. From the category chooser, select **Topology Objectives**
25. Push the **New Topology Objective** button in the Edit Menu toolbar
26. For Name:, enter `Strain_Energy`
27. Select **Strain Energy** response type
28. Make sure the **Min** button is selected for the Objective Definition
29. Push **Next>**
30. Select both the existing loadcases

    One can select multiple items from the list by holding the **Ctrl** button on the keyboard
31. Push the **Finish** button

    Observe that there are 2 objectives defined - one for each loadcase.

Defining the Mass Constraints

32. From the category chooser, select **Topology Constraints**
33. Push the **New Topology Constraint** button in the Edit Menu toolbar
34. Enter Name `MassFr`
35. Select the **Mass Fraction** response
36. Enter `0.25` for the **Upper Bound**
37. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Set the Maximum Number of Design Cycles

38. From the main menu bar, select **Genesis** → **Options**
39. Select the **Design Control** tab
40. Check the **Maximum Design Cycles** box, and set to `30`

41. Push the **Apply** button

Set the Topology Optimization Convergence Criteria

42. From the main menu bar, select **Genesis** → **Options**
43. Select the **Design Control** tab
44. Push the **Advanced** button
45. Select the **Convergence** tab
46. Check the **Hard Absolute (CONV2)** box, and set to `1.E-6`
47. Check the **Soft Variable (CONVDV)** box, and set to `1.E-6`
48. Push the **Close** button
49. Push the **Apply** button

Optimize the Structure

50. From the main menu bar, select **Genesis** → **Optimize**

    Study the console output and the history plot. Push the **Close** button when done

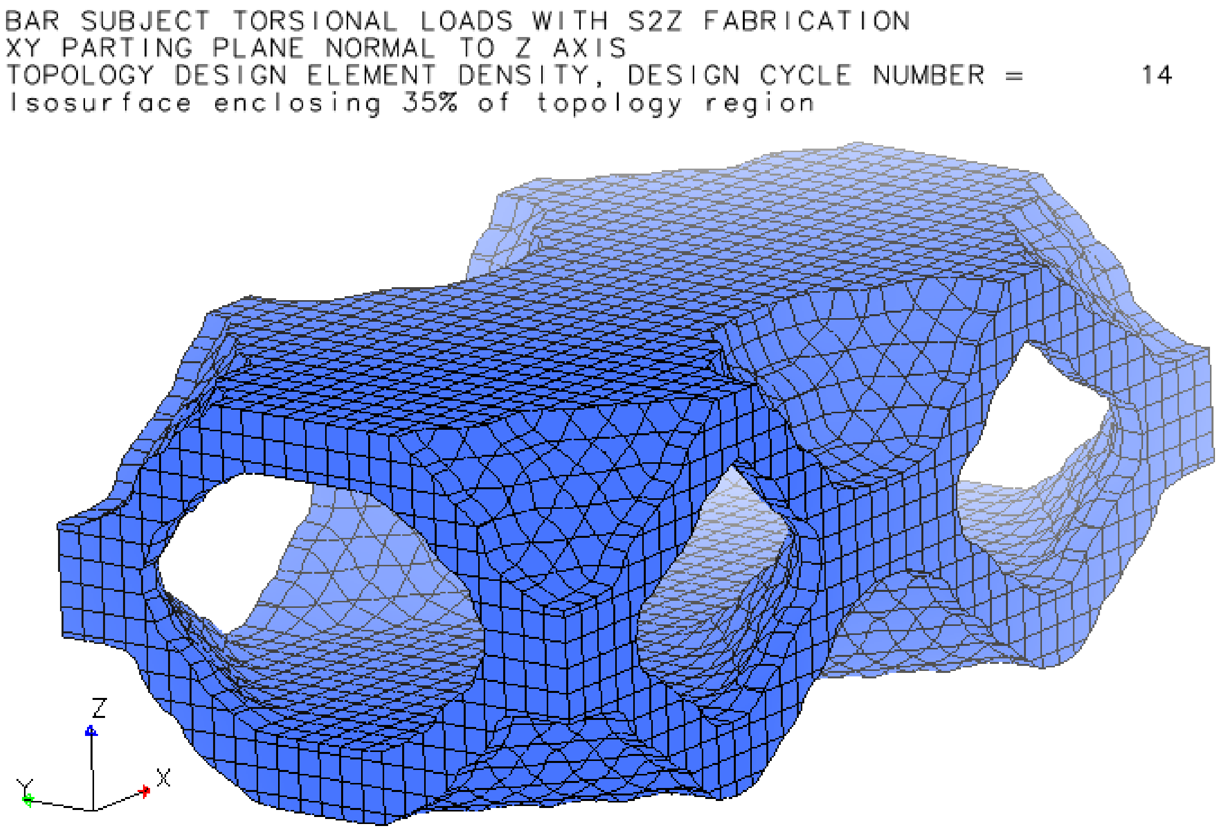
Import the Post-Processing Files (Densities)

51. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
52. Select the `TPDSG011_1_dsgDENS00.pch` file
53. Check the **Import Similar Results for All Design Cycles** box
54. Push the **Open** button

Post-Processing the Results (Densities)

55. Select the **Post** tab
56. Push the **Density Isosurface** button
57. Select the **Topology Result** for the last cycle

The optimization results will look like:



Export the Input File

58. From the main menu bar, select **File** → **Export** → **Input Data...**
59. Enter `TPDSG011_2.dat`
60. Push the Save button

Quit Design Studio

61. From the main menu bar, select **File** → **Quit**
62. Push the **Save** button

    This action saves the file as `TPDSG011_1.dsg`

3.11.2 Part 2

The purpose of this part of the example is to learn how to perform a topology optimization on the brick with stampable sheet fabrication constraints, and with through holes allowed.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG011_2.dat`

   This data file is the result of part 1. If part 1 has not been done then copy `TPDSG011_2_ref.dat` to `TPDSG011_2.dat`.

Modify the Designable Region

3. Select the **Topology** tab
4. From the category chooser, select **Topology Regions**,
5. Select the brick

   You can either click on the part on the screen or select PSOLID 6 from the list.

6. Push the **Modify Topology Design** button
7. Use `0.20` for the initial mass fraction
8. Push the **Next>** button
9. For **Allow Through Holes:** select the **Yes** radio button
10. Push the **Finish** button to complete the design data for the topology region

Modify the Mass Constraint

11. From the category chooser, select **Topology Constraints**
12. Select the `MassFr` constraint, and push the **Modify Constraint** button from the Edit Menu toolbar
13. Enter `0.20` for the **Upper Bound**
14. Push the **Finish** button

Optimize the Structure

15. From the main menu bar, select **Genesis** → **Optimize**

    Study the console output and the history plot. Push the **Close** button when done

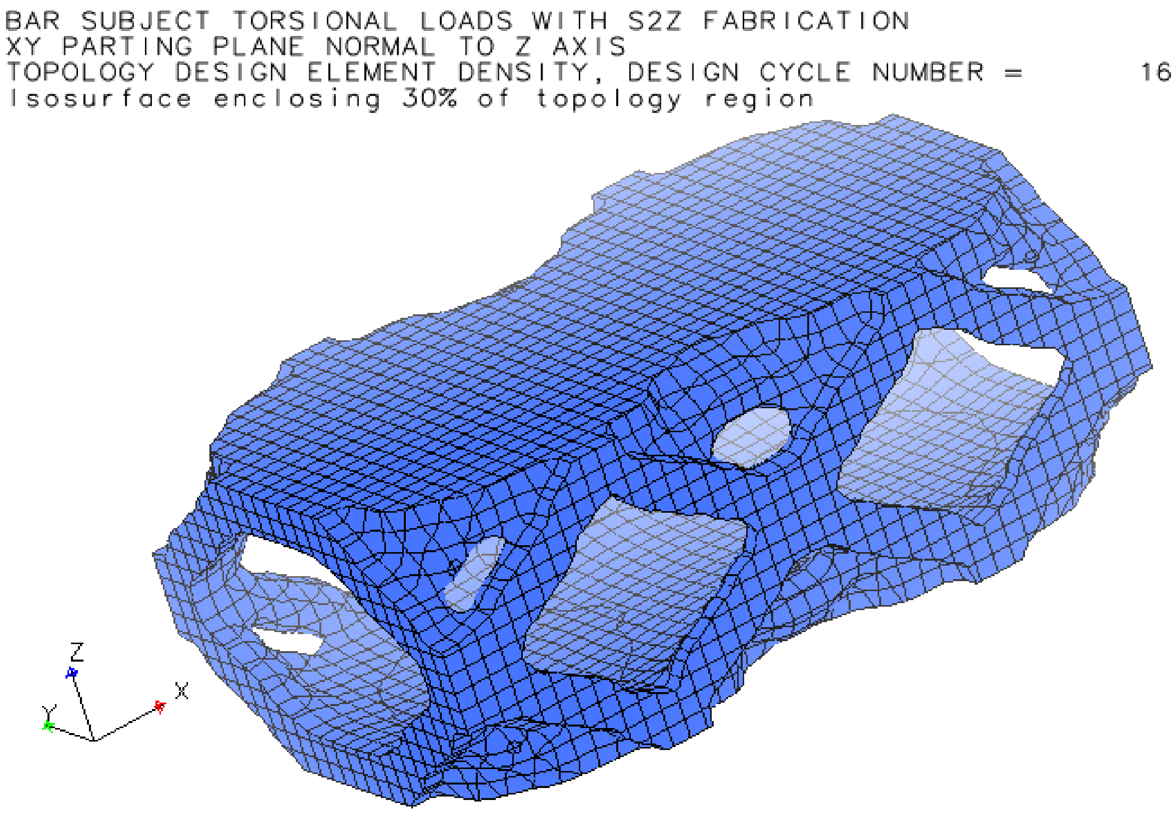
Import the Post-Processing Files (Densities)

16. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
17. Select the `TPDSG011_2_dsgDENSxx.pch` file (where `xx` is the number of the final design cycle)
18. Push the **Open** button

Post-Processing the Results (Densities)

19. Select the **Post** tab
20. Push the **Density Isosurface** button
21. Select the **Topology Result** for the last cycle

The optimization results will look like



Quit Design Studio

22. From the main menu bar, select **File** → **Quit**
23. Push the **Don't Save** button

3.12 Optimal Bonding Location for Thin Sheets Subject to Shear Loads

Introduction

The purpose of this exercise is to demonstrate how to use topology optimization to determine the optimal location for applying adhesive that bonds two thin sheets together.

Problem Statement:

Two thin sheets are to be bonded together where they overlap with a fixed amount of adhesive. The adhesive is modeled as solid elements over the entire overlap area. One end of the bonded flexure is fixed at 2 corners while the opposite end has shear loads applied. Topology optimization is used to determine where a small amount of adhesive should be applied in the overlap area that provides the stiffest bond for the two sheets. The design objective is to minimize the strain energy subject to the Mass Fraction Constraint $<=$ 0.02 for the designable region comprised of the solid element adhesive.

Example ID

`TPDSG012`

Files Used in This problem

A list of the key files is provided and the ones that will be created during this exercise is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification

File Name	Description
`TPDSG012.dat`	Provided: Contains the finite element mesh along with the loading.
`TPDSG012_dsg.dat`	Generated: This file is the same as the `TPDSG012.dat` file along with topology optimization data.
`TPDSG012_ref.dat`	Provided: Reference file same as `TPDSG012_dsg.dat` file

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG012.dat`

Examine the Static Load Cases

3. Select the **Analysis** tab
4. From the category chooser, select **Loadcases**

   Click on the loadcase to examine the applied loads and boundary conditions.

Examine the Property Groups

5. Select the **Display** tab
6. Push the **Show/Hide Groups** button

   Click on the different groups to see the solid adhesive elements.

7. Push the **Up** button

Create the Designable Region

8. Select the **Topology** tab
9. From the category chooser, select **Topology Regions**
10. Select the adhesive

    You can either click on the part on the screen or select PSOLID 4 from the list.

11. Push the **Modify Topology Design** button
12. Use `0.02` for the initial mass fraction
13. Push the **Finish** button to complete the design data for the topology region

    Verify that there is the hammer icon next to the PSOLID 4.

Defining the Design Objective

14. Select the **Topology** tab
15. From the category chooser, select **Topology Objectives**
16. Push the **New Topology Objective** button in the Edit Menu toolbar
17. Select **Strain Energy** response type
18. Make sure the **Min** button is selected for the Objective Definition
19. Push **Next>**
20. Select **STAT ID=1** loadcase
21. Push the **Finish** button

Defining the Mass Constraints

22. From the category chooser, select **Topology Constraints**
23. Push the **New Topology Constraint** button in the Edit Menu toolbar
24. Enter Name `MassFr`
25. Select the **Mass Fraction** response
26. Enter `0.02` for the **Upper Bound**
27. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Set the Maximum Number of Design Cycles

28. From the main menu bar, select **Genesis** → **Options**
29. Select the **Design Control** tab
30. Check the **Maximum Design Cycles** box, and set to `20`
31. Push the **Apply** button

Set the Topology Optimization Move Limits

Since the mass fraction for this problem is so small the following move limits should be set.

32. From the main menu bar, select **Genesis** → **Options**
33. Select the **Design Control** tab
34. Push the **Advanced** button
35. Select the **Move Limits** tab
36. Check the **Fractional Topology (DELT)** box, and set to `0.5`

    The default value for DELT is 1.e-6
37. Check the **Minimum Topology (DTMIN)** box, and set to `0.0066667`

    The default value for DTMIN is 0.2
38. Push the **Close** button
39. Push the **Apply** button

Optimize the Structure

40. From the main menu bar, select **Genesis** → **Optimize**

Import the Post-Processing Files

41. From the **Genesis Console Output** window select the **Import Post...** button
42. Select the `TPDSG012_dsgDENSxx.pch` file (where `xx` designates the last design cycle)
43. Use the **Ctrl** button and select all the analysis results files: `TPDSG012_dsgxx.pch` (where `xx` designates each design cycle)
44. Push the **Import** button
45. In the **Genesis Console Output** window, push the **Close** button

Post-Processing the Results (Displacements)

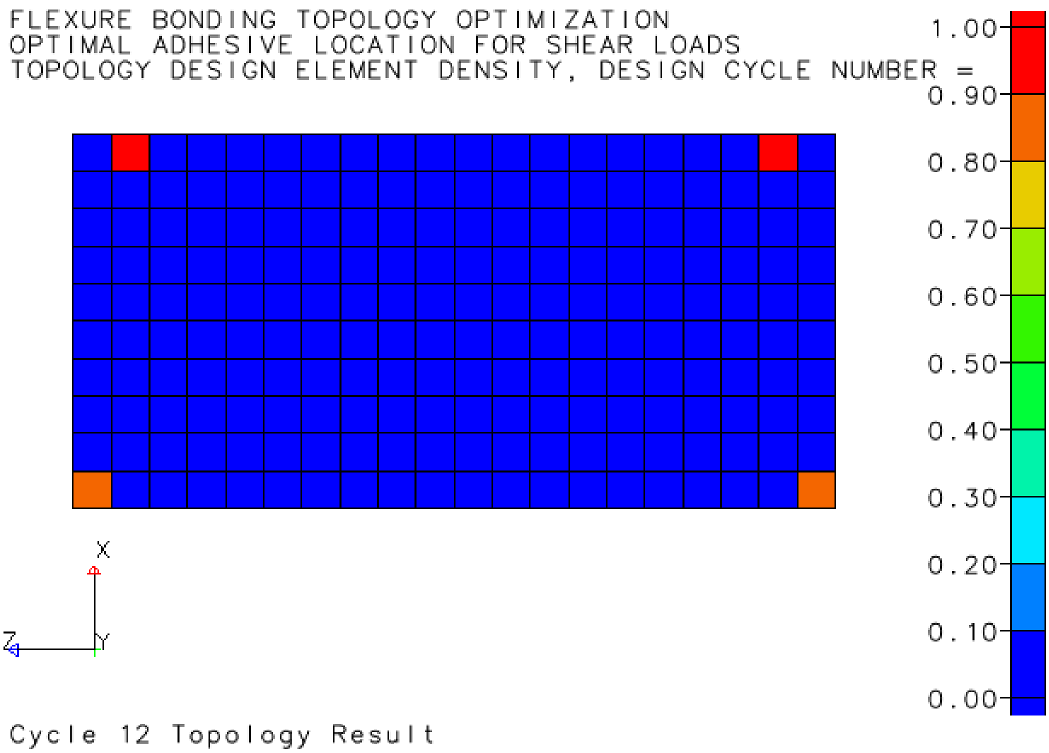
46. Select the **Post** tab
47. Push the **Deform Mesh/Color Mesh** button
48. Select the **Displacement** for the first cycle
49. Select the **Displacement** for the last cycle

Post-Processing the Results (Densities)

50. Select the **Post** tab
51. Push the **Deform Mesh/Color Mesh** button
52. Select the **Topology Result** for the last cycle

53. Push the **Options** button, and click on the **Hide Elements With No Value** box.

    The optimization results will look like:



54. Push the Close button

Quit Design Studio

55. From the main menu bar, select **File** → **Quit**
56. Push the **Save** button

    This action saves the file as `TPDSG012.dsg`

3.13 Topology Optimization of a Compliant Mechanism

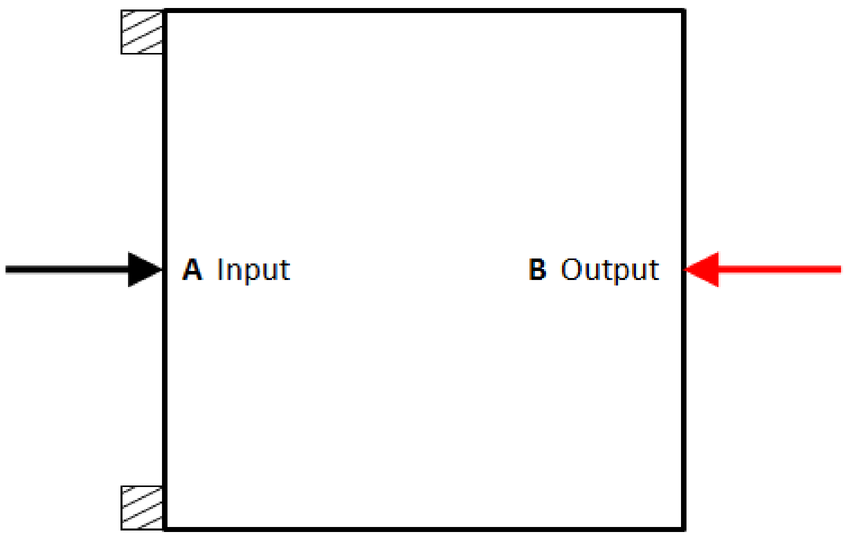
Introduction

A typical topology optimization is formulated to find the optimal structural configuration or the optimal material distribution of structural reinforcement that stiffens the structure. For these types of problems, the optimization is formulated to minimize strain energy and/or to maximize the frequencies which are associated with specific vibration modes.

For design of compliant mechanisms, on the other hand, the objective is to find the structural configuration that deforms to a prescribed direction instead of making it as stiff as possible.

Problem Statement:

Consider the 2D topology area below comprised of shell elements. The upper and



lower corners on the left side are fixed. Input point A, and output point B are also shown. The design goal is to find the structural configuration that deforms as much as possible in the horizontal direction at point B due to a load applied at point A. This example is called a Force/Displacement Inverter (or crunching mechanism) because it is desirable that as point A is moved to the right (black arrow) point B will move to the left (red arrow). The geometry of the compliant mechanism must have structural members between the fixed corners and both points A and B. The design of the total structure should exhibit compliance that is flexible, but not too flexible. The ratio of the output reaction force and the input force F_B/F_A is called the mechanical advantage of the compliant mechanism.

The optimization formulation for this problem will include enforcing a unit displacement at point A as the input with an SPCD. At point B we constrain all but the horizontal motion with SPC's, and apply a fictitious reaction force that artificially restrains it's horizontal displacement to zero.The design objective will be to maximize the reaction force at point B subject to a topology mass fraction design constraint of 40%.

In the current version of Genesis, the reaction force is not supported as a built in design response. However, in this example the reaction force is modeled to be equivalent to the displacement of an SPOINT using DMIG bulk data to modify the stiffness matrix. The DMIG is set up to add an artificial stiffness of -1.0 at the column of the added SPOINT, and the row of point B (Grid 861 comp 1). Using DMIG to alter the stiffness matrix in this way allows the reaction force (i.e. SPOINT displacement) at point B to be designable. It also applies this reaction force to point B as an internal load via the stiffness matrix which "force balances" grid 861 yielding zero displacements just like an SPC would.

The DMIG bulk data for this example looks like:

```
$
$ DMIG data
$
SPOINT      9999
DMIG    RF             0       6       2
DMIG*   RF                          9999
*                    861               1-1.0000000000000
```

Example ID:

TPDSG013

Special Features:

This problem goes through the formulation involved in designing a compliant mechanism using topology optimization. It also describes the formulation for designing the reaction force at a point which is not a built in design response.

Files Used in This problem

A list of the key files is provided and the ones that will be created during this exercise is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification

File Name	Description
TPDSG013.dat	Provided: Contains the finite element mesh along with the loading.
TPDSG013_dmig.bulk	Provided: DMIG bulk data to model the reaction force as a displacement at a scalar point.

TPDSG013_dsg.dat	Generated: This file is the same as the TPDSG013.dat file along with topology optimization data.
TPDSG013_CM.dat	Provided: Coarsened surface of the optimization result along with loading and boundary conditions ready for analysis

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG013.dat`

Examine the Static Load Case

3. Select the **Analysis** tab
4. Change the view by pushing the **X-Y (Top)** icon at the bottom of the main viewport.
5. From the category chooser, select **Loadcases**

   Click on the loadcase to examine the boundary conditions. Notice there are no applied loads shown.

6. From the category chooser, select **Static Loads**

   Click on the load set. Notice that nothing shows on the screen yet.

7. Push the **Modify Load Set** button from the Edit Menu toolbar

   An enforced displacement (SPCD) of 1.0 is now visible on the structure. This enforced displacement is the input loading for this loadcase.

8. Push the **Next>** button

   At the bottom of the right viewport is a message that includes “1 SPCD on 1 grids”.

9. Push the **Cancel** button

Create the Designable Region

10. Select the **Topology** tab
11. From the category chooser, select **Topology Regions**
12. Select the plate

    You can either click on the part on the screen or select PSHELL 5 from the list.

13. Push the **Modify Topology Design** button
14. Use `0.4` for the initial mass fraction
15. Push the **Finish** button

    Verify that there is the hammer icon next to the PSHELL 5.

Defining the Design Objective

16. Select the **Topology** tab
17. From the category chooser, select **Topology Objectives**
18. Push the **New Topology Objective** button in the Edit Menu toolbar
19. For Name, enter `Reaction Force`
20. Select **Displacement** response type
21. Select the **Max** radio button for the Objective Definition

    For this type of compliant mechanism problem we always want to maximize the magnitude of the reaction force. However, the SPOINT displacement placeholder for the reaction force can be either positive or negative depending on the problem set up. For this problem, the reaction force that enforces zero displacement at point B is positive so we will maximize it during the optimization.

22. Push **Next>**
23. For Grid ID, type in the SPOINT ID of `9999` and push the **Add** button
24. Make sure the `Translation 1` radio button is selected
25. Push **Next>**
26. Select **STAT ID=1** loadcase
27. Push the **Finish** button

Defining the Mass Constraints

28. From the category chooser, select **Topology Constraints**
29. Push the **New Topology Constraint** button in the Edit Menu toolbar
30. Enter Name `MassFr`
31. Select the **Mass Fraction** response
32. Enter `0.4` for the **Upper Bound**
33. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Set the Maximum Number of Design Cycles

34. From the main menu bar, select **Genesis** → **Options**
35. Select the **Design Control** tab
36. Check the **Maximum Design Cycles** box, and set to `30`

37. Push the **Apply** button

Optimize the Structure

38. From the main menu bar, select **Genesis** → **Optimize**
39. Inspect the **Genesis Console Output** window

    Look at the DESIGN CYCLE HISTORY table. Notice the objective (which is the reaction force at point B) goes from a negative value to a positive maximum. For each design cycle this is due to the unit displacement at point A and how the current design configuration reacts to it.

Study the Output

40. From the **Genesis Console Output**, select **View Output File**
41. Scroll to the bottom of the file.

    The objective for the last design cycle is listed. It's value should be equal and opposite to the sum of the component 1 REACTION FORCES for SPC'd grids 1, 2, 821, 861, 1600, and 1641. These values for the last design cycle are printed in a table a few pages before the end of the file.
42. Close the output file, the Genesis Console Output viewport, and the Design History viewport.

Import the Post-Processing Files (Densities)

43. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
44. Select the `TPDSG013_dsgDENS00.pch` file
45. Check the **Import Similar Results for All Design Cycles** box
46. Push the **Open** button

Import the Post-Processing Files (Displacements)

47. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
48. Select the `TPDSG013_dsg00.pch` file
49. Check the **Import Similar Results for All Design Cycles** box
50. Push the **Open** button

Post-Processing the Results

51. Select the **Post** tab

52. Push the **Deform Mesh/Color Mesh** button
53. Select the **Displacement Result** for the first cycle
54. Select the **Topology Result** for the first cycle
55. Scroll through the results by pushing the down arrow on the keyboard.
56. Push the **Up** button

Export the Shell Data for the Last Design Cycle

57. Select the **Post** tab
58. Push the **Density Isosurface** button
59. Select the **Topology Result** for the last cycle
60. From the main menu bar, select **File** → **Export** → **Coarsened Surface...**
61. Make sure the Surface File Format radio button is set to **Bulk Data**
62. Slide the Surface Mesh control about 90% of the way to Fine
63. Push the **Save** button

Quit Design Studio

64. From the main menu bar, select **File** → **Quit**
65. Push the **Don't Save** button

Start Design Studio

66. Start Design Studio
67. Import the Genesis data file: `TPDSG013_CM.dat`

    This data file is similar to the coarsened surface file saved in the previous step, but with loads and boundary conditions inserted so that a simple static analysis may be performed on the optimized configuration.

Examine the Static Load Case

68. Select the **Analysis** tab
69. Change the view by pushing the **X-Y (Top)** button at the bottom of the main viewport.
70. From the category chooser, select **Loadcases**

    Click on the loadcase to examine the boundary conditions.

Analyze the Structure

71. From the main menu bar, select **Genesis** → **Single Analysis**

    Close the Genesis Console Output viewport

Import the Post-Processing Files (Displacements)

72. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
73. Select the `TPDSG013_CM_dsg00.pch` file
74. Push the **Open** button

Post-Processing the Results (Displacements)

75. Select the **Post** tab
76. Push the **Deform Mesh/Color Mesh** button
77. Select the **Displacement Result**
78. Push the **Ramp** radio button

    Notice that as point A is displaced to the right (+X), point B moves to the left (-X). This is the type of compliant mechanism that is desired.

Quit Design Studio

79. From the main menu bar, select **File** → **Quit**
80. Push the **Don’t Save** button

3.14 Topology Optimization of a Solid Element Compliant Mechanism

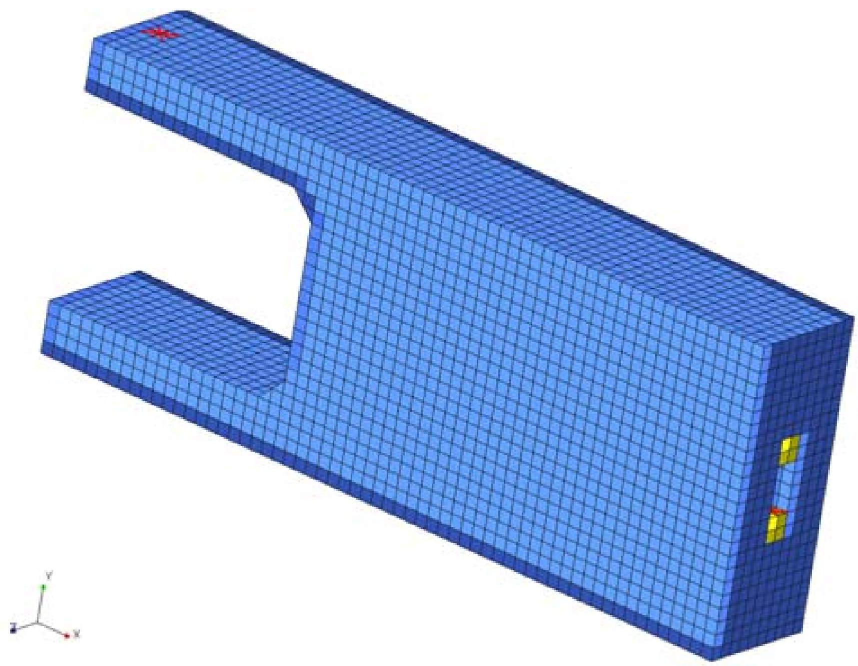
Introduction

A typical topology optimization is formulated to find the optimal structural configuration or the optimal material distribution of structural reinforcement that stiffens the structure. For these types of problems, the optimization is formulated to minimize strain energy and/or to maximize the frequencies which are associated with specific vibration modes.

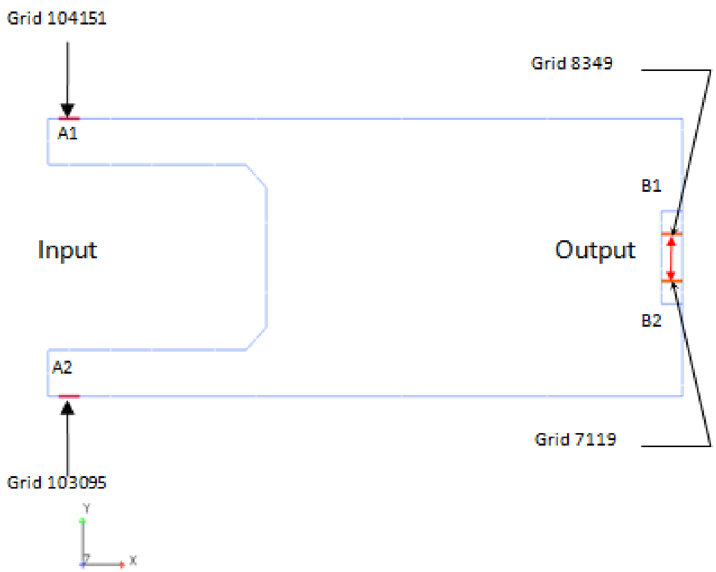
For design of compliant mechanisms, on the other hand, the objective is to find the structural configuration that deforms to a prescribed direction instead of making it as stiff as possible. It is recommended to study the example "**Topology Optimization of a Compliant Mechanism**" prior to running this one.

Problem Statement:

Consider the 3D topology area below. This solid element topology blank will be used



to design a crunching type of compliant mechanism. Handles are already semi formed so that the final design will exhibit a mechanical advantage based on the cantilevered handle geometry.



The upper and lower corners on the left side show input points A1, and A2. As these two points are squeezed together the final design will crunch output points B1 and B2 together. The ratio of the output reaction forces and the input forces F_B/F_A is called the mechanical advantage of the compliant mechanism.

The optimization formulation for this problem will include enforcing a given displacement at point A1 and A2 as the input with SPCD's. At points B1 and B2 we constrain all but the vertical motion with SPC's, and apply fictitious reaction forces that artificially restrains the vertical gap displacement to zero.The design objective will be to maximize the reaction forces at point B1 and B2 subject to a topology mass fraction design constraint of 20%. Mirror symmetry will be enforced about the XZ and XY planes.

In the current version of Genesis, the reaction force is not supported as a built in design response. However, in this example the reaction force is modeled to be equivalent to the displacement of SPOINT's using DMIG bulk data to modify the stiffness matrix. The DMIG data concept is similar to the previous example and it's extension to this example is shown here.

SPOINT definitions:

SPOINT, 98349 is the positive reaction force at point B1 for crunching.
SPOINT, 97119 is the negative reaction force at point B2 for crunching.
SPOINT, 99999 sums the two reaction forces such that a positive value crunches, and a negative value releases. This is done with the Multi Point Constraint (MPC) shown below.

Reaction Forces at points B1 and B2 via DMIG:

```
$
$DMIG data
$
DMIG    RF              0       6       2
DMIG*   RF                              97119
*                    7119                   2-1.0000000000000
DMIG*   RF                              98349
*                    8349                   2-1.0000000000000
```

MPC used for total reaction force:

```
MPC            1    99999       01.000000   97119       01.000000
+                   98349       0-1.00000
```

Example ID:

TPDSG014

Special Features:

This problem goes through the formulation involved in designing a solid element compliant mechanism using topology optimization. It also describes the formulation for designing the reaction forces at two points which is not a built in design response.

Files Used in This problem

A list of the key files is provided and the ones that will be created during this exercise is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification

File Name	Description
TPDSG014.dat	Provided: Contains the finite element mesh along with the loading.
TPDSG014_dmig.bulk	Provided: DMIG bulk data to model the reaction forces as a displacements at scalar points.
TPDSG014_dsg.dat	Generated: This file is the same as the TPDSG014.dat file along with topology optimization data.
TPDSG014_dsg_dmig.bulk	Generated: DMIG bulk data file for TPDSG014_dsg.dat.
TPDSG014_ref.dat	Provided: Reference file same as TPDSG014_dsg.dat
TPDSG014_ref_dmig.bulk	Provided: DMIG bulk data for TPDSG014_ref.dat.

Start Design Studio

1. Start Design Studio

2. Import the Genesis data file: `TPDSG014.dat`

Examine the Static Load Case

3. Select the **Analysis** tab
4. From the category chooser, select **Loadcases**

   Click on the loadcase to examine the boundary conditions. Notice there are no applied loads shown.

5. From the category chooser, select **Static Loads**

   Click on the load set. Notice that nothing shows on the screen yet.

6. Push the **Modify Load Set** button from the Edit Menu toolbar

   An enforced displacement (SPCD) at points A1 and A2 are now visible on the structure. This enforced displacement is the input loading for this loadcase.

7. Push the **Next>** button

   At the bottom of the right viewport is a message that includes “2 SPCDs on 2 grids”.

8. Push the **Cancel** button

Create the Designable Region

9. Select the **Topology** tab
10. From the category chooser, select **Topology Regions**
11. Select the solid element group for topology

    You can either click on the part on the screen or select PSOLID 6 from the list.

12. Push the **Modify Topology Design** button
13. Use `0.2` for the initial mass fraction
14. Push the **Next>** button
15. For Coordinate System:, push the **Change** button
16. Select the `SYM` coordinate system and push the **Next>** button
17. For Constraint 1:, select **MZX: Mirror about XZ plane** from the drop down menu
18. For Constraint 2:, select **MXY: Mirror about XY plane** from the drop down menu
19. Push the **Finish** button

    Verify that there is the hammer icon next to the PSOLID 6.

Defining the Design Objective

20. Select the **Topology** tab
21. From the category chooser, select **Topology Objectives**
22. Push the **New Topology Objective** button in the Edit Menu toolbar
23. For Name, enter `Crunching Forces`
24. Select **Displacement** response type
25. Select the **Max** radio button for the Objective Definition

    For this type of compliant mechanism problem we always want to maximize the magnitude of the reaction forces. However, the SPOINT displacement placeholders for the reaction forces can be either positive or negative depending on the problem set up. For this problem, the reaction forces are summed using an MPC. The reactive crunching forces at points B1 and B2 that enforces zero displacement at the gap is positive so we will maximize it during the optimization.

26. Push **Next>**
27. For Grid ID, type in the SPOINT ID of `99999` and push the **Add** button
28. Make sure the `Translation 1` radio button is selected
29. Push **Next>**
30. Select **STAT ID=1** loadcase
31. Push the **Finish** button

Defining the Mass Constraints

32. From the category chooser, select **Topology Constraints**
33. Push the **New Topology Constraint** button in the Edit Menu toolbar
34. Enter Name `MassFr`
35. Select the **Mass Fraction** response
36. Enter `0.2` for the **Upper Bound**
37. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Set the Maximum Number of Design Cycles

38. From the main menu bar, select **Genesis** → **Options**
39. Select the **Design Control** tab
40. Check the **Maximum Design Cycles** box, and set to `30`
41. Push the **Apply** button

Optimize the Structure

42. From the main menu bar, select **Genesis** → **Optimize**
43. Inspect the Genesis Console Output viewport

    Look at the DESIGN CYCLE HISTORY table. Notice the objective (which is the sum of the reaction forces at points B1 and B2) goes to a positive maximum. For each design cycle this is due to the enforced displacement at points A1 & A2 and how the current design configuration reacts to it.

Study the Output

44. From the Genesis Console Output viewport, select **View Output File**
45. Scroll to the bottom of the file.

    The objective for the last design cycle is listed. It’s value is equal to the sum of the reaction forces at output points B1 and B2. For this example the maximum objective is approximately 1300 Newtons.
    The enforced displacements at the input points A1 and A2 produce reaction forces at grids 104151 and 103095 respectively. Scroll up to see these reaction forces printed in a table. The Y component force magnitudes sum to 130 Newtons for this example.

    The mechanical advantage F_B/F_A is thus 1300/130 = 10 for this configuration.

46. Close the output file, the Genesis Console Output viewport, and the Design History viewport.

Import the Post-Processing Files (Densities)

47. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
48. Select the `TPDSG014_dsgDENSxx.pch` file

    Where `xx` corresponds to the last design cycle.

49. Push the **Open** button

Import the Post-Processing Files (Displacements)

50. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
51. Select the `TPDSG014_dsgxx.pch` file

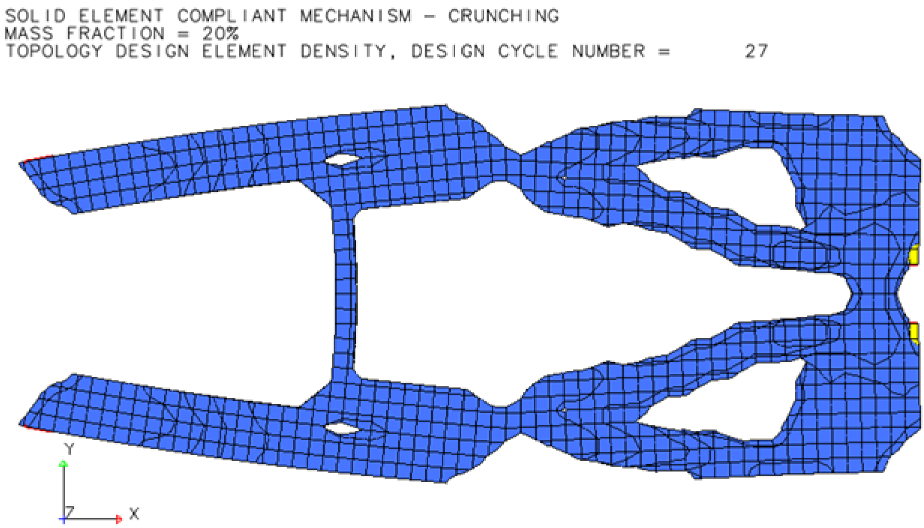
    Where `xx` corresponds to the last design cycle.

52. Push the **Open** button

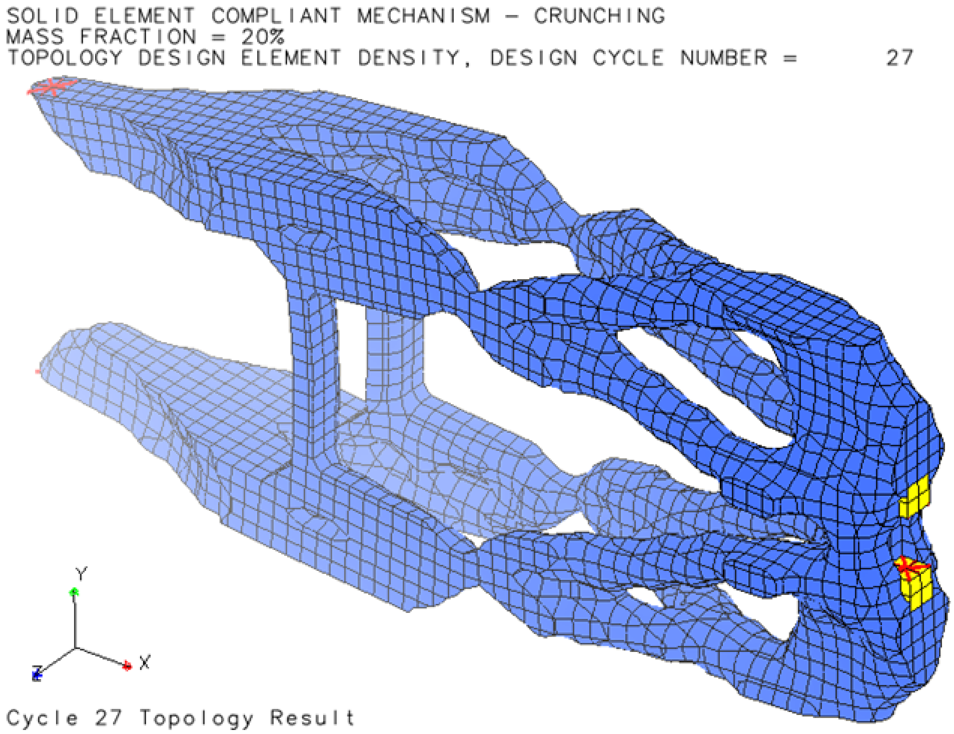
Post-Processing the Results

53. Select the **Post** tab
54. Push the **Density Isosurface** button
55. Select the **Displacement Result** for the last cycle
56. Select the **Topology Result** for the last cycle
57. Push the **Ramp** radio button to animate the static load.

    The following pictures show the expected design:



Cycle 27 Topology Result



58. Push the **Up** button

Quit Design Studio

59. From the main menu bar, select **File** → **Quit**
60. Push the **Don't Save** button

3.15 Study the Effects of Anti-Checkerboard Filter

Introduction

The purpose of this exercise is to demonstrate how the Anti-Checkerboard filter influences final result of the topology optimization of the structure.

Problem Statement:

A Plate comprised of shell elements has a single load applied along the free edge. All the grids along the other edge of the plate are fixed. The design objective is to minimize the strain energy subject to the mass fraction <= 0.25 for the designable region comprised of the entire plate.

The problem is divided into two parts that will demonstrate topology optimization with and without activating the checkerboard filters.

1) How to perform topology optimization with the Anticheckerboard filter **"On"** (Default)

2) How to perform topology optimization with the Anticheckerboard filter **"Off"** and comparison of the final results from both the parts

Example ID

`TPDSG015`

Files Used in This problem

A list of the key files is provided and the ones that will be created during this exercise is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification

File Name	Description
`TPDSG015.dat`	Provided: Contains the finite element mesh along with the loading.
`TPDSG015_1.dat`	Generated: File exported from Design Studio and is the same as the `TPDSG015.dat` file.with anti checkerboard filter set to ON
`TPDSG015_2.dat`	Generated: File exported from Design Studio and is the same as the `TPDSG015.dat` file.with anti checkerboard filter set to OFF

3.15.1 Part 1

The purpose of this part of the example is to learn how to perform a topology optimization on the plate with the Anticheckerboard filter “ON”.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG015.dat`

Set the Topology Optimization Anticheckerboard Filter

3. From the main menu bar, select **Genesis** → **Options**
4. Select the **Design Control** tab
5. Push the **Advanced** button
6. Select the **Misc.** tab
7. Check the **Anticheckerboard Filter (FILTER)** box, and select **On** from the pull down menu
8. Push the **Close** button
9. Push the **Apply** button

Save the Design Studio Database File

10. From the main menu bar, select **File** → **Save As...**
11. Enter `TPDSG015_1` as the Filename and push **Save (**as a Design Studio File**)**

Export the Input File

12. From the main menu bar, select **File** → **Export** → **Input Data...**
13. Enter `TPDSG015_1.dat`
14. Push the **Save** button

Optimize the Structure

15. From the main menu bar, select **Genesis** → **Optimize**

Import the Post-Processing Files (Densities)

16. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
17. Select the `TPDSG015_1_dsgDENS00.pch` file and check the **Import Similar Results for All Design Cycles** checkbox
18. Push the **Open** button

Post-Processing the Results (Densities)

19. Select the **Post** tab
20. Push the **Deform Mesh/Color Mesh** button
21. Push the **Filled Elements** buttons
22. Select the **Topology Results** for the last cycle

Quit Design Studio

23. From the main menu bar, select **File** → **Quit**
24. Push the **Don’t Save** button

3.15.2 Part 2

The purpose of this part of the example is to learn how to perform a topology optimization on the Plate with the Anticheckerboard filter “OFF”.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TPDSG015.dat`

Set the Topology Optimization Anticheckerboard Filter

3. From the main menu bar, select **Genesis** → **Options**
4. Select the **Design Control** tab
5. Push the **Advanced** button
6. Select the **Misc.** tab
7. Check the **Anticheckerboard Filter (FILTER)** box, and select **Off** from the pull down menu
8. Push the **Close** button
9. Push the **Apply** button

Save the Design Studio Database File.

10. From the main menu bar, select **File** → **Save As...**
11. Enter `TPDSG015_2` as the Filename and push **Save (**as a Design Studio File**)**

Export the Input File

12. From the main menu bar, select **File** → **Export** → **Input Data...**
13. Enter `TPDSG015_2.dat`
14. Push the **Save** button

Optimize the Structure

15. From the main menu bar, select **Genesis** → **Optimize**

Import the Post-Processing Files (Densities)

16. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
17. Select the `TPDSG015_2_dsgDENS00.pch` file and check the **Import Similar Results for All Design Cycles** checkbox
18. Push the **Open** button

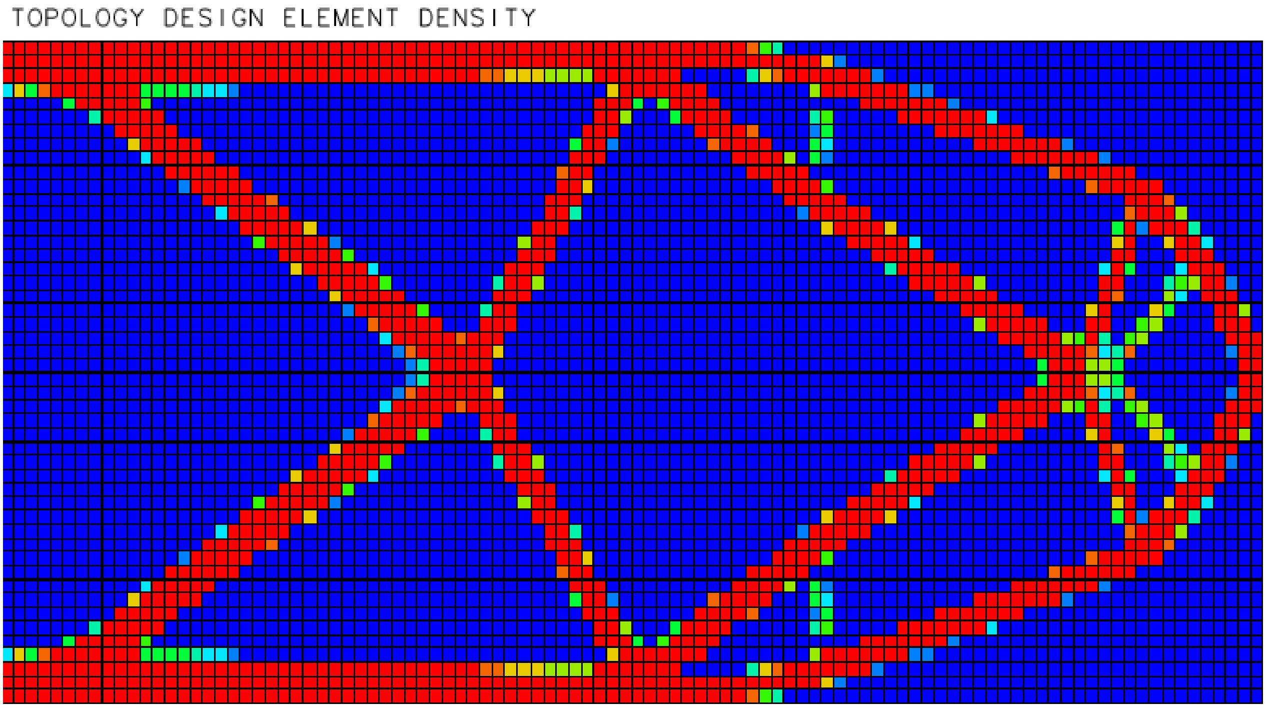
Post-Processing the Results (Densities)

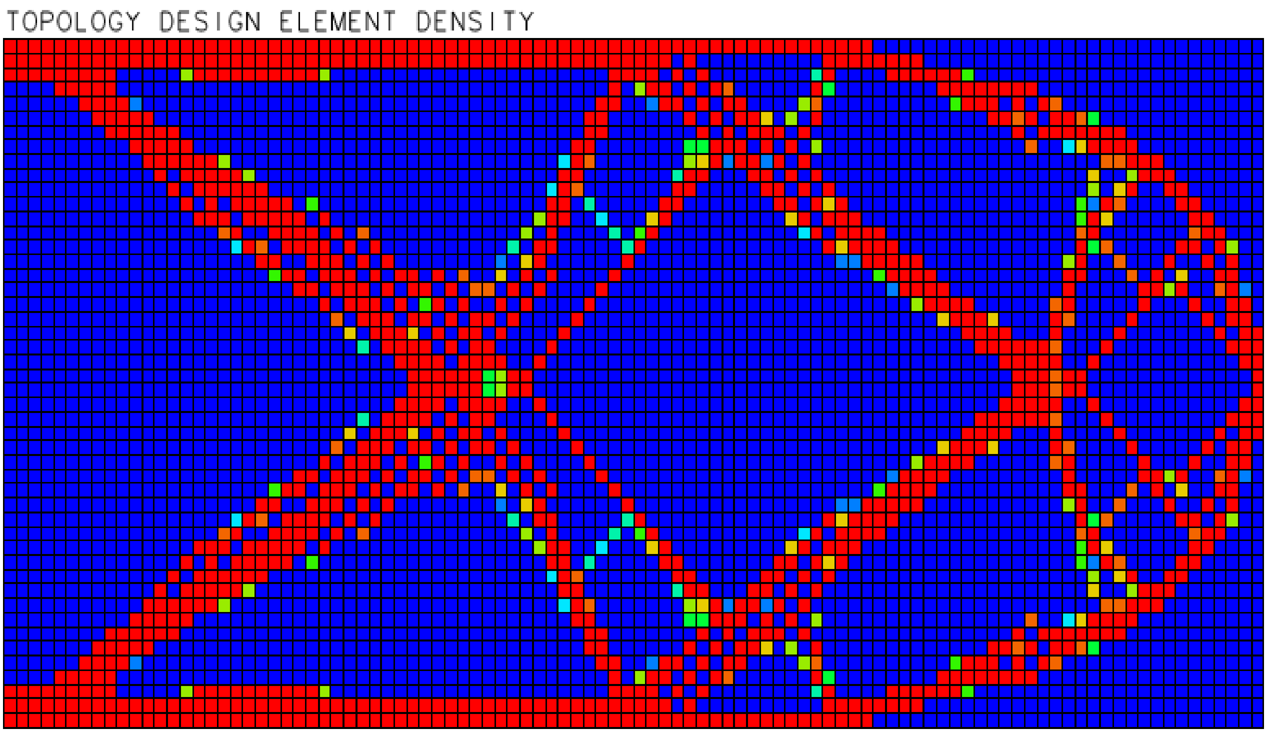
19. Select the **Post** tab
20. Push the **Deform Mesh/Color Mesh** button
21. Push the **Filled Elements** buttons
22. Select the **Topology Results** for the last cycle

Quit Design Studio

23. From the main menu bar, select **File** → **Quit**
24. Push the **Don’t Save** button

Comparison: Anticheckerboarding = ”On” vs. “Off”





3.16 Creating and Using Synthetic Responses (TRESP2)

Introduction

The purpose of this exercise is to get familiar with the creation and usage of synthetic responses in the topology optimization problem. This problem also goes through the steps on how to post process topology optimization results.You will also learn how to visualize Element Strain Energies (ESE) and calculate total strain energy for a loadcase using Design Studio.

In this problem, the objective is to minimize the strain energies for each of the loadcases. As this problem has three loadcases and since the objective function needs to be a single value response, the objective is defined as a normalized sum of the strain energies for each of the loadcases. This approach normally works well for a wide variety of problems. But in some problems, there is a possibility that the strain energy of one of the loadcases might not be minimized.

To minimize all the strain energies, one can use a methodology to solve min-max (minimizing the maximum response) problem called the *Beta method*. This methodology uses synthetic responses to create objective and constraint equations. In this method, an artificial design variable called *Beta* is introduced and additional constraint equations are created using this *Beta* value. The objective function is set to minimize *Beta* and the scaled responses are constrained to be less than *Beta*. If the value of *Beta* is reduced, the maximum value of the response must reduce in order to satisfy the additional *Beta* constraints.

The following optimization problem will be created and solved:

Minimize Beta

Subject to:

Mass Fraction ≤ 0.30
The structure should be castable in the Z direction
((Strain Energy - Loadcase 1)/(Initial Strain Energy - Loadcase 1)) - Beta ≤ 0.0
((Strain Energy - Loadcase 2)/(Initial Strain Energy - Loadcase 2)) - Beta ≤ 0.0
((Strain Energy - Loadcase 3)/(Initial Strain Energy - Loadcase 3)) - Beta ≤ 0.0

Designable region:

All solid elements in the structure

Example ID

TPDSG016

Files Used in This Problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
TPDSG016.dat	Input data	Provided: This file is the GENESIS input data containing the FE mesh along with the loadcase definitions
TPDSG016_dsg.dat	Input data	Created: This file is created by Design Studio to run GENESIS. This contains all the data in TPDSG016.dat along with the optimization data created in the example
TPDSG016_ref.dat	Input data	Provided: This file is provided as a reference file ready to be optimized using GENESIS. It is similar to the TPDSG016_dsg.dat file
TPDSG016_dsgDENSxx.pch	Punch file	Created: Punch file containing the topology optimization results for the xx design cycle
TPDSG016_dsgDENSxx.pch	Punch file	Created: Punch file containing the analysis results (strain energies) for the xx design cycle

Start Design Studio

1. Start Design Studio
2. Load the Genesis data file: TPDSG016.dat

Review the existing Loadcases

3. Select the **Analysis** Tab
4. From the category chooser, select **Loadcases**
5. Select the first loadcase from the list
6. Study the loading and boundary conditions in the Viewport
7. Repeat the process to study the other two loadcases
8. From the main menu, select **Edit** → **Deselect All** to clear any loadcase selection

Create a Designable Region

9. Select the **Topology** tab

10. From the category chooser, select **Topology Regions**
11. Select all the existing PSOLID property groups
12. Push the **Modify Topology Design** button
13. Accept the default of `0.3` for the initial mass fraction
14. Push the **Next>** button
15. Push the **Change** button to change the coordinate system
16. Select `Coord_11`
17. Push the **Next>** button
18. For **Constraint 1**: Select **FGZ**
19. For **Minimum Size** enter `0.5`
20. For **Spread Fraction** delete the existing default value
21. Push the **Finish** button to complete the design data for the topology region

    Verify that there is the hammer icon next to each PSOLID label.
    The hammer icon indicates that the group is being designed.

Create the Beta Extra Variables

22. Select the **Topology** tab
23. From the **Topology** category chooser, select **Extra Variables**
24. Push the **New Topology Extra Variable** button from the Edit Menu toolbar
25. Enter `Beta` in the **Name**
26. Make sure the **Independent Topology Variable (TVAR)** radio button is selected
27. Push the **Next>** button
28. Enter `1.0` for **Initial Value**, `0.0` for **Lower Bound**, and `1.0` for **Upper Bound**
29. Push the **Finish** button

Create Synthetic Responses for Constraints

30. From the **Topology** category chooser, select **Synthetic Responses**
31. Push the **New Topology Synthetic Response** button from the Edit Menu toolbar
32. Enter `Constraint_LC1` for the **Name**
33. Make sure the **User Function (TRESP2)** radio button is selected
34. Push **Next>**

35. Push the **+** button

    The ‘+’ button is used to add arguments to the equation. Notice that everytime you push the ‘+’ button a trail is launched to add a argument. One can delete arguments from the equation by using the ‘-’ button adjacent to the argument.

36. Make sure the **Fundamental Response** radio button is selected
37. Push **Next>**
38. Select the **Strain Energy** radio button as Response Type
39. Push **Next>**
40. Select the `TORSION END 1` loadcase from the list
41. Push **Next**>
42. Push the **+** button
43. Select the **Extra Variable** radio button

    This will activate the design variable menu.

44. Select `1 Beta` from the design variable menu

    Now we have the two arguments required to state the synthetic response. You are free to change those argument names anyway you like.

45. Push **Next>**
46. For the equation, enter `F= (Arg1/607)-Arg2`

    The value of 607 is the strain energy value of the loadcase for the initial design with the density of all the elements at 0.3.

47. Push the **Finish** button
48. Select the created Synthetic response
49. From the Edit menu toolbar, select the **Copy Topology Synthetic Response** button
50. From the Edit menu toolbar, select the **Paste Topology Synthetic Response** button
51. Select the `Copy of Constraint_LC1` synthetic response
52. Push the **Modify Topology Synthetic Response** button from the Edit Menu toolbar
53. Change **Name** to `Constraint_LC2`
54. Push **Next>**
55. Push the **Modify** button adjacent to the first argument `Arg1`
56. Push **Next>**
57. Push **Next>**
58. Select the `TORSION END 2` loadcase from the list

    The scale factor value of 607 was not modified in the equation as the strain energy value is the same as the first loadcase.

59. Push the **Finish** button
60. From the Edit menu toolbar, select the **Paste Topology Synthetic Response** button
61. Select the `Copy of Constraint_LC1` synthetic response
62. Push the **Modify Topology Synthetic Response** button from the Edit Menu toolbar
63. Change **Name** to `Constraint_LC3`
64. Push **Next>**
65. Push the **Modify** button adjacent to the first argument `Arg1`
66. Push **Next>**
67. Push **Next>**
68. Select the `TORSION MIDDLE` loadcase from the list
69. In the equation, change the scale factor value from `607` to `1.59`
70. Push the **Finish** button

Create Synthetic Responses for Objective

71. From the **Topology** category chooser, select **Synthetic Responses**
72. Push the **New Topology Synthetic Response** button from the Edit Menu toolbar
73. Enter `Objective` for the name
74. Make sure the **User Function (TRESP2)** radio button is selected
75. Push **Next>**
76. Push the `+` button
77. Select the **Extra Variable** radio button
78. Select `1 Beta` from the design variable menu
79. Push **Next>**
80. For the equation, enter `F= Arg1`
81. Push the **Finish** button

Defining Design Objective

82. From the category chooser, select **Topology Objectives**
83. Push the **New Topology Objective** button in the Edit Menu toolbar
84. Select the **Synthetic Response** radio button for the response type

85. Make sure **Min** is selected for Objective Definition switch
86. Push **Next>**
87. Select the `Objective` Synthetic Response from the list
88. Push the **Finish** button

Defining the Mass Fraction Constraints

89. From the category chooser, select **Topology Constraints**
90. Push the **New Topology Constraint** button in the Edit Menu toolbar
91. Select the **Mass Fraction** response and make sure **All Designed Groups** option is selected
92. Enter `0.3` for the **Upper Bound**
93. Push the **Finish** button

    Verify that now there is 1 response in the constraint list

Defining the Synthetic Constraints

94. From the category chooser, select **Topology Constraints**
95. Push the **New Topology Constraint** button in the Edit Menu toolbar
96. Select the **Synthetic Response** radio button for the response type
97. Enter `0.0` for the **Upper Bound**
98. Push **Next>**
99. Select the `Constraint_LC1` Synthetic Response from the list
100. Push **Next>**
101. Select the `TORSION END 1` loadcase
102. Push the **Finish** button
103. Push the **New Topology Constraint** button in the Edit Menu toolbar
104. Select the **Synthetic Response** radio button for the response type
105. Enter `0.0` for the **Upper Bound**
106. Push **Next>**
107. Select the `Constraint_LC2` Synthetic Response from the list
108. Push **Next>**
109. Select the `TORSION END 2` loadcase

110. Push the **Finish** button
111. Push the **New Topology Constraint** button in the Edit Menu toolbar
112. Select the **Synthetic Response** radio button for the response type
113. Enter `0.0` for the **Upper Bound**
114. Push **Next>**
115. Select the `Constraint_LC3` Synthetic Response from the list
116. Push **Next>**
117. Select the `TORSION MIDDLE` loadcase
118. Push the **Finish** button

Set Output Control for the First and Last Design Cycles Only

119. From the main menu bar, select **Genesis** → **Options**
120. Select the **Output Control** tab
121. Check the **Analysis Output** box, and select **First & Last**
122. Check the **Design Output** box, and select **First & Last**
123. Push the **Apply** button

Set the Maximum Number of Design Cycles

124. From the main menu bar, select **Genesis** → **Options**
125. Select the **Design Control** tab
126. Check the **Maximum Design Cycles** box, and set to `35`
127. Push the **Apply** button

Optimize the Structure Using Genesis

128. From the main menu bar, select **Genesis** → **Optimize**
129. Study the **Design History** charts, when done push the Close button
130. Study the **Genesis Console Output** window

Import the Post Processing Files

131. From the **Genesis Console Output** window, select **Import Post...** button
132. Select all the post-processing files

133. Push the **Import** button
134. From the **Genesis Console Output** window, select **Close** button

Postprocessing the Results (Density Isosurfaces)

135. Select the **Post** tab
136. Push the **Density Isosurface** button
137. Select a Topology Result for last design cycle

    Identify the created cavities. Is this structure castable?

138. Push the **Options...** button. Slide the bar to display different isosurface values
139. Select the **Show Topology Region** checkbox to view the initial topology space
140. Push the **Close** button
141. Push the **Up** button

Postprocessing the Results (Density Values)

142. Select the **Post** tab
143. Push the **Deform Mesh/Color Mesh** button
144. Select a Topology Result for any design cycle

    Study the results.

145. Push the **Options...** button. Slide the lower cutoff slidebar to mask out element with low density values
146. Push the **Close** button
147. Push the **Up** button

Postprocessing the Strain Energy Results

148. Select the **Post** tab
149. Push the **Deform Mesh/Color Mesh** button
150. Select a ESE Result for a loadcase in the initial design cycle
151. Select a ESE Result for a loadcase in the final design cycle
152. Study the strain energy distribution in each of the loadcases
153. Push the **Up** button

Calculate the Total Strain Energy Results

154. Select the **Post** tab
155. Push the **Sum ESE Results** button
156. Select a ESE Result for a loadcase 1 in the initial design cycle
157. Select **Element Strain Energy** in the options listbox below
158. Push the **Inspect** button
159. Make sure the **Sum ESE for Selected Groups** radio button is selected
160. Push **Next>**
161. Select the designed PSOLID group
162. Push **Finish**

    The total strain energy value of 607 is printed in the **Design Studio Messages** window.

163. Select a ESE Result for a loadcase 1 in the final design cycle
164. Select **Element Strain Energy** in the options listbox below
165. Push the **Inspect** button
166. Make sure the **Sum ESE for Selected Groups** radio button is selected
167. Push **Next>**
168. Select the designed PSOLID group
169. Push **Finish**

    The total strain energy value for the final design is printed in the **Design Studio Messages** window. Notice the decrease in Strain Energy from the initial to the final design.

170. Repeat the above steps to compare the strain energies between the initial and final designs for the other two loadcases

Quit Design Studio

171. From the main menu bar, select **File** → **Quit**
172. Push the **Don't’ Save** button

CHAPTER 4

Sizing Optimization Examples

- **Simple Sizing Optimization Setup**
- **Using One Variable to Design Multiple Regions**
- **Quick Sizing - Designing ROD elements**
- **Defining Non-linear Relationship between Property and Design Variable**
- **Using the Design Element Library**
- **Design of a Square Plate with Volume Heat Generation and Enforced Temperature**
- **Design Using Multidisciplinary Load Cases - Three Rod Truss**
- **Quick Sizing Shell Elements - Design of a Pallet Lifter**
- **Discrete Optimization - Three Rod Truss**
- **Design Based on Random Response**

4.1 Simple Sizing Optimization Setup

Introduction

The purpose of this exercise is to get familiar with the creation of basic sizing optimization data and to learn how to post process simple sizing optimization results. A design variable is created and linked to the property that is to be designed.

The following optimization problem will be created and solved:

Minimize Mass

Subject to:

Von mises Stress $\leq$ 300 MPa

Designable region:

The thickness of the structure

Example ID

`SZDSG001`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SZDSG001.dat`	`Input data`	Provided: Contains the finite element mesh of a bracket with applied load and boundary conditions.
`SZDSG001_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `SZDSG001.dat`.
`SZDSG001_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SZDSG001_dsgxx.pch`	`Punch Output data`	Generated using Genesis within Design Studio. This file contains the analysis outputs requested for a design cycle during the optimization
`SZDSG001_ref.dat`	`Input data`	Provided. Reference result input file. Should be nearly identical to `SZDSG001_dsg.dat`. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SZDSG001.dat`

   The imported model will be shown in Viewport Window area.

Define Design Variable

3. From the **Design** category chooser, select **Design Variables**
4. Push the **New Design Variable** button from the Edit Menu toolbar
5. Enter `Thickness` in the name
6. Select **Independent Design Variable** radio button
7. Push the **Next>** button.
8. Enter `1.0` for **Initial Value**, `0.1` for **Lower Bound**, and `4.0` for **Upper Bound**.
9. Push **Finish**

   Notice that there is an asterisk in front of the design variable, which indicates that this design variable is not being used.

Defining Sizing Optimization

10. From the **Design** category chooser, select **Sizing**
11. Select PSHELL 2 from the list
12. Push the **Modify Sizing Design** button to add design attributes.
13. Select `Thickness` design variable created earlier in the **Thickness** dropdown listbox.

    As the **Nonstructural Mass** is not designed, no design variable is assigned for this.

14. Push **Finish**

    You will see the hammer icon next to the group in the list. This indicates that the group is being size-designed.

Defining Design Objective

15. From the **Design** category chooser, select **Objectives**
16. Push the **New Objective** button from the Edit menu toolbar to create a new objective.
17. Enter `Mass` in the Name field.
18. Select **Mass** as response type.

    Since this model has only one group property, there is no need to select specific groups or materials for the mass. Leave **Entire Model** in the menu.

19. Since objective is to minimize mass, select **Min** as Objective Definition Switch.
20. Push **Finish** button.

Defining the Design Constraints

21. From the **Design** category chooser, select **Constraints**
22. Push the **New Constraint** button from the Edit menu toolbar to create a new constraint.
23. Enter `vonMises` in the name field.
24. Select **Stress** as response type and select **Selected Groups** from the menu.
25. Enter `300` in the Upper Bound.

    Since von Mises stress is always positive, there is no need to specify lower bound.

26. Push the **Next>** button.
27. Pick the structure from the Viewport Window or select PSHELL 2 from the property list.
28. Push the **Next>** button.
29. Select von Mises Top & Bottom from the menu as the stress component.
30. Push the **Next>** button.
31. Select all the loadcases listed in the panel.

    Either selecting one by one while holding down the **Ctrl** key or in one shot with the **Shift** key.

32. Push the **Finish** button.

Optimize the Structure Using Genesis

33. From the main menu bar, select **Genesis** → **Optimize**

    Genesis will start optimizing the model. The Genesis Console Output dialog and Design History dialog will be shown and you can monitor the optimization progress.

Importing the Results

34. From the Genesis Console Output window, select **Import Post...** button
35. Select all the files to be imported: `SZDSG001_dsgxx.pch` where `xx` is the design cycle number.
36. Push the **Import** button.
37. From the Genesis Console Output window, select the **Close** button

Post-processing the Results

38. Select the **Post** tab.
39. Push the **Animation** button.
40. Select **Shell Stress/Strain** from the Color Result Type chooser.
41. Push **Filled Contours** (depending on your viewing preference)
42. Push the **Next>** button.
43. Press and hold the **Ctrl** key and select cycles 0, 1, and 2 for "`Loadcase 1 Shell Stress`".

    This will let you view how the von Mises stress from loadcase 1 has changed over the design cycles. (You can also try to view loadcases 2 and 3.)
44. Push the **Next>** button.

    There ia a slide bar for the animation speed and three other buttons. Change the slide bar if the animation speed is too fast or too slow. If you wish to save the animation as a "gif" file for later presentation, push **Save Animation**. You can also save individual frames by pushing **Save Animation Frames...**. If you want to change the upper/lower cutoffs, scaling factor, and other view related options, push **Options...**.
45. Push the **Finish** button after you finish animation.

Quit Design Studio

46. From the main menu bar, select **File** → **Quit**
47. Push the **Don't Save** button

Study the Results

48. Open the output file `SZDSG001_dsg.out` in a text editor. Study the values of the objective and the constraint and complete the following table

Type	Initial Value	Final Value	change%
Objective			
Maximum Constraint Violations			

One should notice that the objective function value increases despite the fact that we are trying to minimize the objective. This is because the optimization algorithm tries to first satisfy the constraints and achieve a feasible design which has more priority.

4.2 Using One Variable to Design Multiple Regions

Introduction

The purpose of this exercise is to solve a sizing optimization problem.Two independent design variables are used to design the thicknesses of three regions.

The following optimization problem will be created and solved:

Minimize Mass

Subject to:

Von mises Stress $\leq$ 300 MPa

Designable region:

Three regions of the structure using two independent design variables

Example ID

SZDSG002

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SZDSG002.dat	Input data	Provided: Contains the finite element mesh of a bracket with applied load and boundary conditions.
SZDSG002_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in SZDSG002.dat.
SZDSG002_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
SZDSG002_dsgxx.pch	Punch Output data	Generated using Genesis within Design Studio. This file contains the analysis outputs requested for a design cycle during the optimization
SZDSG002_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SZDSG002_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio

2. Load the Genesis data file: `SZDSG002.dat`

   The imported model will be shown in Viewport Window.

Define Two Design Variables

3. From the **Design** category chooser, select **Design Variables**
4. Push the **New Design Variable** button from the Edit menu toolbar
5. Enter `Back` in the name
6. Select **Independent Design Variable** radio button
7. Push the **Next>** button.
8. Enter `1.0` for **Initial Value**, `0.1` for **Lower Bound**, and `4.0` for **Upper Bound**.
9. Push **Finish**

   Notice that there is an asterisk in front of the design variable, which indicates that this design variable is not being used.
10. Select the design variable created from the list
11. From the Edit Menu toolbar, select **Copy Design Variable** button
12. From the Edit Menu toolbar, select **Paste Design Variable** button
13. Push the **Modify Design Variable** button
14. Enter `Side` in the name
15. Push **Finish**

    As the initial value, lower and upper bounds are the same as the other variable, only the name needs to be modified.

Defining Sizing Optimization

16. Select **Design** tab
17. From the **Design** category chooser, select **Sizing**
18. Select PSHELL 2 from the list

    The back of the bracket will be highlighted.
19. Push the **Modify Sizing Design** button to add design attributes.
20. Select `Back` in the **Thickness** menu.
21. Push **Finish**

    You will see the hammer icon next to the group in the list. This indicates that the group is being size-designed.

22. Select PSHELL 4 and 6 from the list in the panel. (Hold the Control key and select them both.)

    Make sure to de-select PSHELL 2. Both sides of the bracket are now highlighted.

23. Push **Modify Sizing Design** button to add design attributes.
24. Check the box to the left of the **Thickness** menu.

    The check box is there because multiple groups were selected to start the trail. Only items with the check box checked will be used to update all selected groups. Items with the box unchecked will be left as is in each group.

25. Select `Side` in the **Thickness** menu.
26. Push **Finish**.

    You will now see the hammer icon next to all the groups in the list. This indicates that they are all being size-designed.

Defining Design Objective

27. From the **Design** category chooser, select **Objectives**
28. Push **New Objective** button from the Edit menu toolbar to create a new objective.
29. Enter `Mass` in the Name field.
30. Select **Mass** as response type.

    Since this model has only one group property, no need to select specific groups or materials for the mass. Leave **Entire Model** in the menu.

31. Since the objective is to minimize mass, select **Min** as Objective Definition Switch.
32. Push **Finish**

Defining the Design Constraints

33. From the **Design** category chooser, select **Constraints**
34. Push **New Constraint** button from the Edit menu toolbar to create a new constraint.
35. Enter `vonMises` in the name field.
36. Select **Stress** as response type and select **Selected Groups** from the menu.
37. Enter `300` in the Upper Bound.

    Since von Mises stress is always positive, no need to specify lower bound.

38. Push **Next>**
39. Select PSHELL 2, 4, and 6 from the property list.
40. Push **Next>**

41. Select **von Mises Top & Bottom** from the menu as stress component.
42. Push **Next>**
43. Select all the loadcases listed in the panel.
44. Push **Finish**

Optimize the Structure Using Genesis

45. From the main menu bar, select **Genesis** → **Optimize**

    Genesis will start optimizing the model. The Genesis Console Output dialog and Design History dialog will be shown and you can monitor the optimization progress. Once the optimization is finished, you can view the output file by pushing the **View Output File** button or you can close the dialogs by pushing the **Close** button.

Importing the Results

46. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
47. Select `SZDSG002_dsg00.pch` from the file browser.
48. Check the **Import Similar Results for All Design Cycles** check box, then push the **Open** button.

    This will allow all similar files to import in one shot.

Viewing the Stress Results

49. Select the **Post** tab.
50. Push **Deform Mesh/Color Mesh...**
51. For the **Color Mesh** list, select a Shell Stress result to plot.
52. Exit Design Studio.

Study the Results

53. Open the output file. Study the values of the objective and the constraint and complete the following table

Type	Initial Value	Final Value	change%
Objective			
Maximum Constraint Violations			

4.3 Quick Sizing - Designing ROD elements

Introduction

The purpose of this exercise is to create and solve a sizing optimization problem.The quick sizing setup option in Design Studio is used to define the sizing design of multiple properties at the same time.

The analysis model is a ten-rod truss model with static loading. The design data for all the ten ROD elements in the truss is created at the same time.

The following optimization problem will be created and solved:

Minimize Mass

Subject to:

Stress in the ROD elements $\leq$ 25000 MPa

Designable region:

Ten ROD elements using ten independent design variables

Example ID

SZDSG003

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SZDSG003.dat	Input data	Provided: Contains the finite element mesh of a ten-rod truss with applied load and boundary conditions.
SZDSG003_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in SZDSG003.dat.
SZDSG003_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
SZDSG003_dsgxx.pch	Punch Output data	Generated using Genesis within Design Studio. This file contains the analysis outputs requested for a design cycle during the optimization
SZDSG003_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SZDSG003_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Load the Genesis data file: `SZDSG003.dat`
3. Select the XY icon in the Viewport to change to Top view

   The imported model will be shown in Viewport Window.

Design the ten ROD elements

4. From the **Design** category chooser, select **Quick Setup Trails**

   Notice a summary of the Design data defined. There is no design data defined.

5. Push the **Quick Sizing Setup** button
6. Push the **Select** button to select all the properties in the list
7. Push **Next>**

   Now you will be asked to enter parameters to define the initial value, the lower and the upper bounds of the design variables. All three values are determined by the area value on the property card (PROD) multiplied by Mx (multiplier) plus Cx (constant). For example, if you are designing rod area which is defined as 1.0 in the PROD group property, the initial area A0, the lower and the upper bounds AL and AU are calculated by A0=1.0*M1+C1, AL=1.0*M2+C2, and AU=1.0*M3+C3 respectively. MinLB and MaxUB are absolute minimum and maximum regardless of the current group property values.

8. Accept defaults of `0.0` in **C1** and `1.0` in **M1** fields

   This will make the initial value to be the same, as specified by the analysis data.

9. Enter `0.001` in **C2** field and `0.0` in **M2** field

   This will make the lower bound value to be 0.001 for all designable areas.

10. Enter `0.0` in **C3** field and `10.0` in **M3** field

    This will make the upper bound value to be 10 times the analysis data areas.

11. Push the **Finish** button

    Summaries are shown in the panel. Now there are 10 design variables defined and linking with 10 sizing regions.
    To modify some of the defined design variables or sizing regions, the user has to go to the **Design Variables** or **Sizing** categories respectively in the **Design** Category Chooser.

Defining Design Objective

12. From the **Design** category chooser, select **Objectives**
13. Push **New Objective** button from the Edit menu toolbar to create a new objective.

14. Enter `Mass` in the Name field.
15. Select **Mass** as response type.

    Since this model has only one group property, no need to select specific groups or materials for the mass. Leave **Entire Model** in the menu.
16. Since the objective is to minimize mass, select **Min** as Objective Definition Switch.
17. Push **Finish**

Defining the Design Constraints

18. From the **Design** category chooser, select **Constraints**
19. Push **New Constraint** button from the Edit menu toolbar to create a new constraint.
20. Enter `RodStress` in the name field.
21. Select **Stress** as response type and select **Selected Groups** from the menu.
22. Enter `-25000.0` for the **Lower bound** and `25000.0` for the **Upper Bound**.

    Since the rod might be in tension or compression, one needs to specify both the lower and upper bounds.
23. Push **Next>**
24. Select all the existing properties from the list by pushing the **Select** button.
25. Push **Next>**
26. Select **Stress End A** from the **PROD Stress** menu as stress component.
27. Push **Next>**
28. Select the existing loadcase listed in the panel.
29. Push **Finish**

Request the Element Sizing Data File to be Output

30. From the main menu bar, select **Genesis** → **Options...**
31. Select the **File Control** tab
32. For **Element Sizing File**, make sure that the **Create** option is selected
33. Push the **Apply** button

Optimize the Structure Using Genesis

34. From the main menu bar, select **Genesis** → **Optimize**

    Genesis will start optimizing the model. The Genesis Console Output dialog and Design History dialog will be shown and you can monitor the optimization progress. Once the optimization is finished, you can view the output file by pushing the **View Output File** button or you can close the dialogs by pushing the **Close** button.

Importing the Stress Results

35. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
36. Select `SZDSG003_dsg00.pch` from the file browser.
37. Check the **Import Similar Results for All Design Cycles** check box, then push the **Open** button.

    This will allow all similar files to import in one shot.

Viewing the Stress Results

38. Select the **Post** tab.
39. Push **Deform Mesh/Color Mesh...**
40. For the **Color Mesh** list, select a Rod Stress result to plot.

    Compare the stress results from the first to the last design cycle. Some rods violate the stress constraint in the initial design cycle.
41. Push the **Up** button

Importing the Sizing Results

42. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
43. Select `SZDSG003_dsgOPOST00.pch` from the file browser.
44. Check the **Import Similar Results for All Design Cycles** check box, then push the **Open** button.

    This will allow all similar files to import in one shot.

Viewing the Sizing Results

45. Select the **Post** tab.
46. Push **Deform Mesh/Color Mesh...** button
47. For the **Color Mesh** list, select a PROD Area result to plot.

    Compare the areas from the first to the last design cycle. Initially, all the rod have the same area of 5.0 while in the final design the areas are different for each rod.
48. Click on an element in the Viewport to see the value of the area in the Design Studio messages window.

Quit Design Studio

49. From the main menu bar, select **File** → **Quit**
50. Push the **Don’t Save** button

Study the Output File

51. Start any text editor
52. In the text editor load the Genesis data file: `SZDSG003_dsg.out`
53. Navigate to the end of the file for a listing of the objective values in each design cycle
54. Study the design variables values changes in each design cycle
55. Close the file

4.4 Defining Non-linear Relationship between Property and Design Variable

Introduction

The purpose of this exercise is to create and solve a sizing optimization problem by designing the properties of a BAR element. As the properties of the BAR are nonlinear functions of the dimensions equations are used to define this nonlinear relationship. The analysis model is a cantilever beam with three BAR elements.

The following optimization problem will be created and solved:

Minimize Mass

Subject to:

Stress in the BAR elements ≤ 45000.0

End Displacement ≤ 2.0

First Fundamental Frequency ≥ 30.0

Designable region:

Properties of the BAR elements - Area, Moment of Inertia and Stress Recovery Locations

Example ID

`SZDSG004`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SZDSG004.dat`	`Input data`	Provided: Contains the finite element mesh of a cantilever beam with applied load and boundary conditions.
`SZDSG004_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `SZDSG004.dat.`
`SZDSG004_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SZDSG004_dsgxx.pch`	`Punch Output data`	Generated using Genesis within Design Studio. This file contains the analysis outputs requested for a design cycle during the optimization

SZDSG004_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SZDSG004_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Load the Genesis data file: `SZDSG004.dat`
3. Select the XY icon in the Viewport to change to Top view

Define Two Independent Design Variables

4. From the **Design** category chooser, select **Design Variables**
5. Push the **New Design Variable** button from the Edit menu toolbar
6. Enter `B` in the name
7. Select **Independent Design Variable** radio button
8. Push the **Next>** button.
9. Enter `0.5` for **Initial Value**, `0.1` for **Lower Bound**, and `1.0` for **Upper Bound**.
10. Push **Finish**
11. Push the **New Design Variable** button from the Edit menu toolbar
12. Enter `H` in the name
13. Select **Independent Design Variable** radio button
14. Push the **Next>** button.
15. Enter `1.0` for **Initial Value**, `0.1` for **Lower Bound**, and `10.0` for **Upper Bound**.
16. Push **Finish**

    Notice that there is an asterisk in front of both the design variable, which indicates that this design variable is not being used.

Define Three Equation Design Variables

17. From the **Design** category chooser, select **Design Variables**
18. Push the **New Design Variable** button from the Edit menu toolbar
19. Enter `Area` in the name
20. Select **Equation Design Variable** radio button
21. Push the **Next>** button.

22. Select the design variable `B` for the first argument
23. Push the + button
24. Select the design variable `H` from the dropdown listbox for the second argument
25. In the panel below the + button, enter `F=Arg1*Arg2` to define the equation.
26. Push **Finish**

    Notice that the asterisk in front is not present for the independent variables as they are being used in the equation. Note the asterisk in front of the equation variable which indicates that this one is not being used.

27. Push the **New Design Variable** button from the Edit menu toolbar
28. Enter `Moment of Inertia` in the name
29. Select **Equation Design Variable** radio button
30. Push the **Next>** button.
31. Select the design variable `B` for the first argument
32. Push the + button
33. Select the design variable `H` from the dropdown listbox for the second argument
34. In the panel below the + button, enter `F=Arg1*Arg2**3/12` to define the equation.
35. Push **Finish**
36. Push the **New Design Variable** button from the Edit menu toolbar
37. Enter `Stress Location` in the name
38. Select **Equation Design Variable** radio button
39. Push the **Next>** button.
40. Select the design variable `H` for the first argument
41. In the panel below the + button, enter `F=Arg1/2` to define the equation.
42. Push **Finish**.

Defining Sizing Optimization

43. From the **Design** category chooser, select **Sizing**
44. Select the PBAR properties in the list
45. Push the **Modify Sizing Design** button to add design attributes.
46. Select the checkbox for **Area** and select `Area` in the menu
47. Select the checkbox for **I1** and select `Moment of Inertia` in the menu

48. Select the checkbox for **C1** and select `Stress Location` in the menu

    The checkbox needs to be checked while defining data for multiple selections in Design Studio. Only the properties with the selected checkbox are updated for all the selections.

49. Push **Finish**

    You will see the hammer icon next to the group in the list. This indicates that the group is being size-designed.

Defining Design Objective

50. From the **Design** category chooser, select **Objectives**
51. Push **New Objective** button from the Edit menu toolbar to create a new objective.
52. Enter `Mass` in the Name field.
53. Select **Mass** as response type.
54. Select **Min** as Objective Definition Switch.
55. Push **Finish**

Defining the Stress Design Constraints

56. From the **Design** category chooser, select **Constraints**
57. Push **New Constraint** button from the Edit menu toolbar to create a new constraint.
58. Enter `Stress-EndA` in the name field.
59. Select **Stress** as response type and select **Selected Groups** from the menu.
60. Enter `-45000.0` for the Lower bound and `45000.0` for the Upper Bound.
61. Push **Next>**
62. Select all the existing properties from the list by pushing the **Select** button.
63. Push **Next>**
64. Select **Stress Point C at End A** from the menu as stress component.
65. Push **Next>**
66. Select the existing loadcase listed in the panel.
67. Push **Finish**
68. Push **New Constraint** button from the Edit menu toolbar
69. Enter `Stress-EndB` in the name field.
70. Select **Stress** as response type and select **Selected Groups** from the menu.
71. Enter `-45000.0` for the Lower bound and `45000.0` for the Upper Bound.

72. Push **Next>**
73. Select all the existing properties from the list by pushing the **Select** button.
74. Push **Next>**
75. Select **Stress Point C at End B** from the menu as stress component.
76. Push **Next>**
77. Select the existing loadcase listed in the panel.
78. Push **Finish**

Defining the Displacement Design Constraints

79. From the **Design** category chooser, select **Constraints**
80. Push **New Constraint** button from the Edit menu toolbar to create a new constraint.
81. Enter `End Displacement` in the name field.
82. Select **Displacement** as response type.
83. Enter `-2.0` for the Lower bound and `2.0` for the Upper Bound.
84. Push **Next>**
85. Select the end grid (grid id 4) from the Viewport or type `4` and select Add for selecting the grid.
86. Select the **Translation 2** radio button
87. Push **Next>**
88. Select the existing loadcase listed in the panel.
89. Push **Finish**

Defining the Frequency Design Constraints

90. From the **Design** category chooser, select **Constraints**
91. Push **New Constraint** button from the Edit menu toolbar
92. Enter `1st Frequency` in the name field.
93. Select **Frequency Mode Number** as response type and enter `1` for the mode number
94. Enter `30.0` for the Lower bound.
95. Push **Next>**
96. Select the existing frequency loadcase listed in the panel.
97. Push **Finish**

Optimize the Structure Using Genesis

98. From the main menu bar, select **Genesis** → **Optimize**

    Genesis will start optimizing the model. The Genesis Console Output dialog and Design History dialog will be shown and you can monitor the optimization progress. Once the optimization is finished, you can view the output file by pushing the **View Output File** button or you can close the dialogs by pushing the **Close** button.

Quit Design Studio

99. From the main menu bar, select **File** → **Quit**
100. Push the **Don't Save** button

Study the Output File

101. Start any text editor
102. In the text editor load the Genesis data file: `SZDSG004_dsg.out`
103. Navigate to the end of the file for a listing of the objective values in each design cycle
104. Study the design variables values changes in each design cycle
105. Close the file

4.5 Using the Design Element Library

Introduction

The purpose of this exercise is to create and solve a sizing optimization problem by designing the properties of a BAR element. A portal frame modeled with BAR elements is used for the analysis. The BAR elements are defined with a rail cross-section. The dimensions of this rail cross-section are directly designed using the design element library. The nonlinear relationship between the dimensions and the properties of the BAR is taken into account in the design library.

The following optimization problem will be created and solved:

Minimize Volume

Subject to:

Stress in the BAR elements ≤ 20000.0

Translation Displacement at loading ≤ 4.0

Rotational Displacement at loading ≤ 0.015

Designable region:

Cross-sectional dimensions of the BAR elements

Example ID

SZDSG005

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SZDSG005.dat	Input data	Provided: Contains the finite element mesh of a portal frame with applied load and boundary conditions.
SZDSG005_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in SZDSG005.dat.
SZDSG005_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
SZDSG005_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SZDSG005_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Load the Genesis data file: `SZDSG005.dat`
3. Select the XY icon in the Viewport to change to Top view

Review the Group Properties

4. Select the **Analysis** Tab
5. From the **Analysis** category chooser, select **Group Properties**
6. Select the existing PBARL group from the list
7. Push the **Modify Group Property** button in the Edit menu toolbar
8. Study the cross-section definition of the PBARL using the element library

   The six dimensions defining the rail section will be designed using the design element library.
9. Push **Cancel** button

Review the Existing Loadcase

10. From the **Analysis** category chooser, select **Loadcases**
11. Select the first existing loadcase and study its loading in the viewport
12. Select the second existing loadcase and study its loading in the viewport

Review the Six Independent Design Variables

13. From the **Design** category chooser, select **Design Variables**
14. Study the six design variables that control the dimensions of the rail.

    Notice that there is an asterisk in front of all the design variable, which indicates that this design variable is not being used.
    The **"I"** after the asterisk indicates that it is an independent design variable.
    The initial value and the bounds of the variable are also displayed.

Defining Sizing Optimization

15. From the **Design** category chooser, select **Sizing**
16. Select the PBARL property in the list
17. Push the **Modify Sizing Design** button to add design attributes.
18. Check that **Rail** is selected for the **Section** option

19. Select the design variable `d1` for **d1**
20. Similarly, select the design variables that are used to design the corresponding dimension of the rail.
21. Push **Finish**

    You will see the hammer icon next to the group in the list. This indicates that the group is being size-designed.

Defining Design Objective

22. From the **Design** category chooser, select **Objectives**
23. Push **New Objective** button from the Edit menu toolbar to create a new objective.
24. Enter `Volume` in the Name field.
25. Select **Volume** as response type.
26. Select **Min** as Objective Definition Switch.
27. Push **Finish**

Defining the Stress Design Constraints

28. From the **Design** category chooser, select **Constraints**
29. Push **New Constraint** button from the Edit menu toolbar to create a new constraint.
30. Enter `Stress1` in the name field.
31. Select **Stress** as response type and select **Selected Groups** from the menu.
32. Enter `-20000.0` for the Lower bound and `20000.0` for the Upper Bound.
33. Push **Next>**
34. Select the existing PBARL property from the list.
35. Push **Next>**
36. Select **Stress Location 1** from the **PBARL Stress** menu as stress component.
37. Push **Next>**
38. Select all the existing loadcases listed in the panel.
39. Push **Finish**
40. Select the existing constraint (`Stress1`) defined previously
41. From the Edit Menu toolbar, select **Copy Constraint** button
42. From the Edit Menu toolbar, select **Paste Constraint** button

43. Select the `Copy of Stress1` constraint and push the **Modify Constraint** button in the Edit menu toolbar
44. For the name, enter `Stress2`
45. Push **Next>**
46. Push **Next>**
47. Select **Stress Location 2** from the **PBARL Stress** menu as stress component.
48. Push **Finish**
49. From the Edit Menu toolbar, select **Paste Constraint** button
50. Select the `Copy of Stress1` constraint and push the **Modify Constraint** button in the Edit menu toolbar
51. For the name, enter `Stress8`
52. Push **Next>**
53. Push **Next>**
54. Select **Stress Location 8** from the **PBARL Stress** menu as stress component.
55. Push **Finish**
56. From the Edit Menu toolbar, select **Paste Constraint** button
57. Select the `Copy of Stress1` constraint and push the **Modify Constraint** button in the Edit menu toolbar
58. For the name, enter `Stress9`
59. Push **Next>**
60. Push **Next>**
61. Select **Stress Location 9** from the **PBARL Stress** menu as stress component.
62. Push **Finish**

Defining the Displacement Design Constraints

63. Push **New Constraint** button from the Edit menu toolbar
64. Enter `Translational Displacement` in the name field.
65. Select **Displacement** as response type.
66. Enter `-4.0` for the Lower bound and `4.0` for the Upper Bound.
67. Push **Next>**
68. Select the grid (grid id 3) where the loading is applied from the Viewport or type `3` and select Add for selecting the grid.

69. Select the **Translation 1** radio button
70. Push **Next>**
71. Select all the existing loadcases listed in the panel.
72. Push **Finish**
73. Select the displacement constraint (`Translational Displacement`)
74. From the Edit Menu toolbar, select **Copy Constraint** button
75. From the Edit Menu toolbar, select **Paste Constraint** button
76. Select the `Copy of Translational Displacement` constraint and push the **Modify Constraint** button in the Edit menu toolbar
77. For the name, enter `Rotational Displacement`
78. Enter `-0.015` for the Lower bound and `0.015` for the Upper Bound.
79. Push **Next>**
80. Select the **Rotation 3** radio button
81. Push **Finish**

Optimize the Structure Using Genesis

82. From the main menu bar, select **Genesis** → **Optimize**

    Genesis will start optimizing the model. The Genesis Console Output dialog and Design History dialog will be shown and you can monitor the optimization progress. Once the optimization is finished, you can view the output file by pushing the **View Output File** button or you can close the dialogs by pushing the **Close** button.

Quit Design Studio

83. From the main menu bar, select **File** → **Quit**
84. Push the **Don't Save** button

Study the Output File

85. Start any text editor
86. In the text editor load the Genesis data file: `SZDSG005_dsg.out`
87. Navigate to the end of the file for a listing of the objective values in each design cycle
88. Study the design variables values changes in each design cycle
89. Close the file

4.6 Design of a Square Plate with Volume Heat Generation and Enforced Temperature

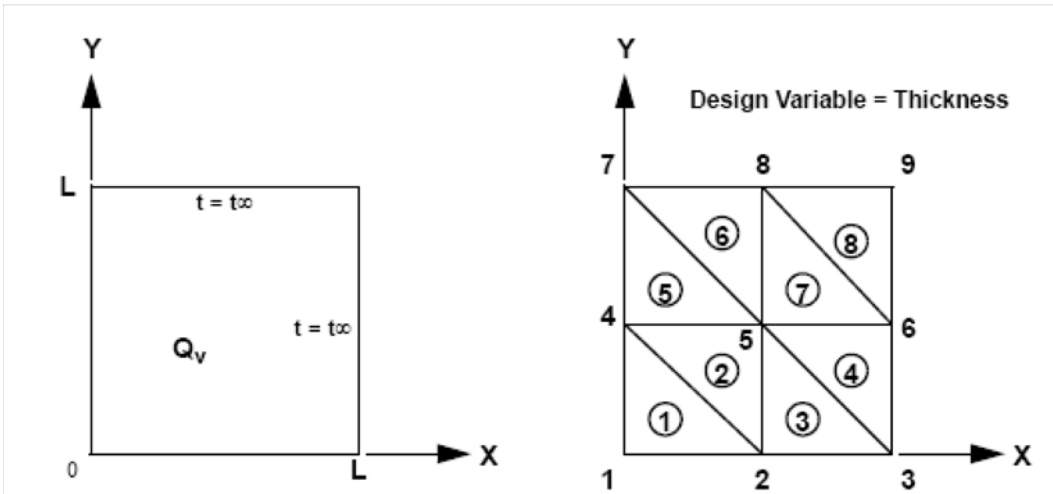
Introduction

The purpose of this exercise is to setup an optimization problem with heat transfer analysis.

Problem Statement:

Minimize the volume of the square plate subjected to temperature constraints.

Consider that there is also convection through one of the faces of the plate and through the lines defined by (0,Y) and (X,0). Assume that the ambient temperature is 50.0 K and a convection coefficient of 5.0 Watt/cm2- K.



Design Problem:

Minimize volume.

1. One design variable (thickness of the plate).
2. One set of temperature constraints.
3. One heat transfer load case.

Problem Description:

1. The structure consists of 8 TRIA3 elements subjected to a volumetric heat generation load and an enforced temperature at the root. There are also 8 CHBDYAREA3 elements to model convection through one face and 4 CHBDYLINE elements to model the convection through the (0,Y) and (X,0) boundaries.

2. The temperatures at grids 1,2,4 and 5 are constrained to be between 40 K and 100 K.

3. The set of plate element properties will be controlled by a single design variable. Note that this will change the eight TRIA3 element thickness at the same time. Also, the area factor AF of the CHBDYLINE elements is controlled by this design variable.

Example ID

`SZDSG006`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SZDSG006.dat`	`Input data`	Provided: Contains the finite element mesh of the structure
`SZDSG006_ref.dat`	`Input data`	Provided: Reference input file ready to be optimized
`SZDSG006_dsgxx.pch`	`DSG file`	Created: Punch file containing the temperature results for each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SZDSG006.dat`

Create a Heat Transfer Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select `Loadcase 1`
5. From the Edit Menu toolbar, select **Delete Loadcase** button
6. Push the **New Loadcase** button from the Edit menu toolbar
7. Enter `HEAT_TRANSFER` in the name field
8. Check the **Heat Transfer** radio button
9. Push **Next>**
10. From the **SPC** category chooser, select `22 SPC Set 22`

11. Push **Next>**
12. From the **Heat Transfer Load Set** category chooser, select `200 Heat Set 200`
13. Push **Next>**
14. From the first **Temperature** category chooser, select **Post**
15. From the second **Temperature** category chooser, select **All**
16. Push the **Finish** button

Define the Design Objective

17. From the **Design** category chooser, select **Objectives**
18. Push the **New Objective** button from the Edit menu toolbar
19. Enter `OBJ_FUN` in the name field
20. Select the **Volume** radio button as Response Type
21. Select the **Min** radio button as Objective Definition Switch
22. Push the **Finish** button

Define the Constraints

23. From the **Design** category chooser, select **Constraints**
24. Push the **New Constraint** button from the Edit menu toolbar
25. Enter `TEMPI` in the name field
26. Select the **More Response Types** radio button as the Response Type
27. Enter `40.0` as **Lower Bound** and `100.0` as **Upper Bound**
28. Push **Next>**
29. Select the **Temperature** radio button in Additional Responses
30. Push **Next>**
31. Enter `1,2,4,5` in the **Select by Grid ID** field
32. Push the **Add** button
33. Push **Next>**
34. Select `HEAT_TRANSFER` loadcase
35. Push the **Finish** button

Create an Independent Design Variable for the thickness

36. From the **Design** category chooser, select **Design Variables**
37. Push the **New Design Variable** button from the Edit menu toolbar
38. Enter `THICKNS` in the name field
39. Select the **Independent Design Variable** radio button
40. Push **Next>**
41. Enter `1.0` as **Initial Value**, `0.1` as **Lower Bound**, and `10.0` as **Upper Bound**
42. Push the **Finish** button

    Notice that there is an asterisk in front of the independent design variable, it indicates that this design variable is not being used. The "**I**" indicates this is an **I**ndependent design variable.

Define the Sizing Data

43. From the **Design** category chooser, select **Sizing**
44. Select PSHELL 22 group
45. Push the **Modify Sizing Design** button
46. From the **Thickness** category chooser, select `1 THICKNS`
47. Push the **Finish** button
48. Select PHBDY 23 group
49. Push the **Modify Sizing Design** button
50. From the **Area Factor** category chooser, select `1 THICKNS`
51. Push the **Finish** button

Save the Design Studio file

52. From the main menu bar, select **File** → **Save As...**
53. Enter `SZDSG006` as the Filename and push **Save (**as a Design Studio File)

Optimize the structure using Genesis

54. From the main menu bar, select **Genesis** →**Optimize**
55. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

56. From the main menu bar, select **File** →**Import** →**Punch/Output2 Results...**

57. Select the `SZDSG006_dsg00.pch` file
58. Check the **Import Similar Results for All Design Cycles** check box

    This will allow all similar files, to import in one shot.
59. Push the **Open** button

Post-Process the Temperature Results

60. Select the **Post** tab
61. Push the **Deform Mesh/Color Mesh**
62. In the **Color Mesh** window, select **Filled Contours**.
63. Select each cycle from the **Color Mesh** window and study it
64. Right-click in the viewport, select **List Top Ten**

    The list is printed in the Messages window.
65. Push **Up** when finished

Study the Output File

66. Start any text editor
67. In the text editor load the Genesis data file: `SZDSG006_dsg.out`
68. Study the file by listing the objective values
69. Study the design variables values
70. Close the file

Quit Design Studio

71. From the main menu bar, select **File** →**Quit**

4.7 Design Using Multidisciplinary Load Cases - Three Rod Truss

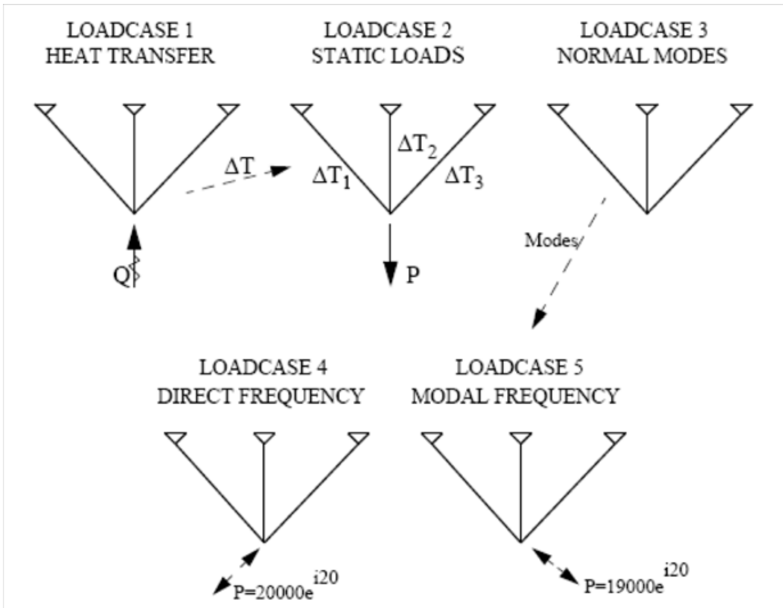
Introduction

The purpose of this exercise is to setup an optimization problem that includes heat transfer, normal modes, static, and frequency response analyses.

Problem Statement:

Considering the analysis problem, minimize the mass of the structure shown. Also, consider that

- 1. For the heat transfer analysis the temperature of grid 4 should be between -36 and 36 degrees.
- 2. For the frequency calculation analysis the first natural frequency should be above 75 hz.
- 3. For the static analysis the displacement on grid 4 should be less than 0.2 and the stress in each element should be between -1500 and 2000.
- 4. For the direct dynamic response analysis the magnitude of the displacement at grid 4 should be smaller than 0.2
- 5. For the modal dynamic response analysis the real part of the displacement on grid 4 should be smaller than 0.2.



The three design variables are the rod areas.

Design Problem:

Minimize mass

1. 3 sizing design variables
2. 5 load cases
3. temperature constraints in heat transfer load case.
4. static displacement constraints in static load case.
5. static stress constraints in static load case.
6. natural frequency constraints in frequency calculation load case.
7. dynamic displacement constraints in direct and modal dynamic response load cases.

Example ID

`SZDSG007`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SZDSG007.dat`	`Input data`	Provided: Contains the finite element mesh of the structure
`SZDSG007_ref.dat`	`Input data`	Provided: Reference input file ready to be optimized
`SZDSG007_dsgxx.pch`	`DSG file`	Created: Punch file containing the analysis results for each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SZDSG007.dat`

Create a Frequency Response Loadcase (Normal Mode)

3. From the **Analysis** category chooser, select **Loadcases**
4. Push the **New Loadcase** button from the Edit menu toolbar
5. Enter `NAT_FREQ` in the name field
6. Check the **Normal Modes** radio button
7. Push **Next>**

8. From the **SPC** category chooser, select `100 SPC Set 100`
9. Push **Next>**
10. From the **Eigenvalue Method** category chooser, select `300 Method 300`
11. Push the **Finish** button

Create a Direct Frequency Response Loadcase using Direct Dynamic Analysis

12. From the **Analysis** category chooser, select **Loadcases**
13. Push the **New Loadcase** button from the Edit menu toolbar
14. Enter `DirFreqResp` in the name field
15. Check the **Direct Frequency Response** radio button
16. Push **Next>**
17. From the **SPC** category chooser, select `100 SPC Set 100`
18. Push **Next>**
19. From the **Dynamic Load Set** category chooser, select `400 DLoad Set 400`
20. From the **Loading Frequency Set** category chooser, select `200 Frequency Set 200`
21. Push **Next>**
22. From the first **Displacement** category chooser, select **Both**
23. From the second **Displacement** category chooser, select **All**
24. Push the **Finish** button

Create a Modal Frequency Response Loadcase using Modal Dynamic Analysis

25. From the **Analysis** category chooser, select **Loadcases**
26. Push the **New Loadcase** button from the Edit menu toolbar
27. Enter `ModFreqResp` in the name field
28. Select the **Modal Frequency Response** radio button
29. Push **Next>**
30. Push **Next>**
31. Push **Next>**

32. From the **Dynamic Load Set** category chooser, select `500 DLoad Set 500`
33. From the **Loading Frequency Set** category chooser, select `200 Frequency Set 200`
34. From the **Modes Loadcase** category chooser, select `NAT_FREQ`
35. From the **Modal Damping** category chooser, select `500 DLoad Set 500`
36. Push **Next>**
37. From the first **Displacement** category chooser, select **Both**
38. From the second **Displacement** category chooser, select **All**
39. Push the **Finish** button

Define the Design Objective

40. From the **Design** category chooser, select **Objectives**
41. Push the **New Objective** button from the Edit menu toolbar
42. Enter `MASS` in the name field
43. Select the **Mass** radio button as Response Type
44. Check the **Min** radio button as Objective Definition Switch
45. Push the **Finish** button

Define the Temperature Constraints

46. From the **Design** category chooser, select **Constraints**
47. Push the **New Constraint** button from the Edit menu toolbar
48. Enter `TEMP` in the name field
49. Select the **More Response Types** radio button as the Response Type
50. Enter `-36.0` as **Lower Bound** and `36.0` as **Upper Bound**
51. Push **Next>**
52. Select the **Temperature** radio button in Additional Responses
53. Push **Next>**
54. Enter `4` in the **Select by Grid ID** field
55. Push the **Add** button
56. Push **Next>**
57. Select `HEAT_TRANSFER` loadcase

58. Push the **Finish** button

Define the Natural Frequency Constraint

59. From the **Design** category chooser, select **Constraints**
60. Push the **New Constraint** button from the Edit menu toolbar
61. Enter `FreqMode1` in the name field
62. Select the **Frequency Mode Number** radio button as the Response Type, and ensure that mode number 1 is entered
63. Enter `75` as **Lower Bound**
64. Push **Next>**
65. Select `NAT_FREQ` loadcase
66. Push the **Finish** button

Define the Static Analysis Displacement and Stress Constraints

67. From the **Design** category chooser, select **Constraints**
68. Push the **New Constraint** button from the Edit menu toolbar
69. Enter `STAT_U4` in the name field
70. Select the **Displacement** radio button in Response Type
71. Enter `-0.2` as **Lower Bound**
72. Enter `0.2` as **Upper Bound**
73. Push **Next>**
74. Enter `4` in the **Select by Grid ID** field
75. Push the **Add** button
76. Select the **Translation 1** radio button in Component
77. Push **Next>**
78. Select `STAT_LOAD` loadcase
79. Push the **Finish** button
80. Push the **New Constraint** button from the Edit menu toolbar
81. Enter `STAT_V4` in the name field
82. Select the **Displacement** radio button in Response Type
83. Enter `-0.2` as **Lower Bound**

84. Enter `0.2` as **Upper Bound**
85. Push **Next>**
86. Enter `4` in the **Select by Grid ID** field
87. Push the **Add** button
88. Select the **Translation 2** radio button in Component
89. Push **Next>**
90. Select `STAT_LOAD` loadcase
91. Push the **Finish** button
92. Push the **New Constraint** button from the Edit menu toolbar
93. Enter `STRESS_1` in the name field
94. Select the **Stress** radio button in Response Type
95. Enter `-1500.0` as **Lower Bound**
96. Enter `2000.0` as **Upper Bound**
97. Push **Next>**
98. Select PROD 11 group
99. Push **Next>**
100. Under **Item Code Information**, for the **PROD Stress** rod end chooser, select **Stress End B**
101. Push **Next>**
102. Select `STAT_LOAD` loadcase
103. Push the **Finish** button
104. Highlight the `STRESS_1` constraint just created
105. From the Edit Menu toolbar, select **Copy Constraint** button
106. From the Edit Menu toolbar, select **Paste Constraint** button

    A copy of the Stress_1 constraint now appears.

107. Highlight the `Copy of STRESS_1` constraint just created
108. Push the **Modify Constraint** button in the Edit menu toolbar
109. Enter `STRESS_2` in the name field
110. Push **Next>**
111. Select PROD 12 group
112. Push **Finish>**

113. Highlight the `STRESS_2` constraint just created
114. From the Edit Menu toolbar, select **Copy Constraint** button
115. From the Edit Menu toolbar, select **Paste Constraint** button

    A copy of the Stress_2 constraint now appears.

116. Highlight the `Copy of STRESS_2` constraint just created
117. Push the **Modify Constraint** button in the Edit menu toolbar
118. Enter `STRESS_3` in the name field
119. Push **Next>**
120. Select PROD 13 group
121. Push **Finish>**

Define the Direct Dynamic Displacement Constraints

122. From the **Design** category chooser, select **Constraints**
123. Push the **New Constraint** button from the Edit menu toolbar
124. Enter `MAG_U4` in the name field
125. Select the **More Response Types** radio button as the Response Type
126. Enter `0.2` as **Upper Bound**
127. Push **Next>**
128. Select the **Dynamic Displacement** radio button in Additional Responses
129. From the **Dynamic Displacement** pull down menu, select **From Direct (DDISP)**
130. Select the **Magnitude** radio button in Dynamic Component
131. Push **Next>**
132. Push the **Select None** button
133. Enter `4` in the **Select by Grid ID** field
134. Push the **Add** button
135. Select the **Translation 1** radio button in Component
136. Push **Next>**
137. Select `DirFreqResp` loadcase
138. Push the **Finish** button
139. Highlight the `MAG_U4` constraint just created
140. From the Edit Menu toolbar, select **Copy Constraint** button

141. From the Edit Menu toolbar, select **Paste Constraint** button
142. Highlight the `Copy of MAG_U4` constraint just created
143. Push the **Modify Constraint** button in the Edit menu toolbar
144. Change the name to `MAG_V4`
145. Push **Next>**
146. Push **Next>**
147. Select the **Translation 2** radio button in Component
148. Push the **Finish** button

Define the Modal Dynamic Displacement Constraints

149. Push the **New Constraint** button from the Edit menu toolbar
150. Enter `REA_U4` in the name field
151. Select the **More Response Type** radio button as the Response Type
152. Enter `0.2` as **Upper Bound**
153. Push **Next>**
154. Select the **Dynamic Displacement** radio button in Additional Responses
155. From the **Dynamic Displacement** pull down menu, select **From Modal (MDISP)**
156. Select the **Real** radio button in Dynamic Component
157. Push **Next>**
158. Push the **Select None** button
159. Enter `4` in the **Select by Grid ID** field
160. Push the **Add** button
161. Select the **Translation 1** radio button in Component
162. Push **Next>**
163. Select `ModFreqResp` loadcase
164. Push the **Finish** button
165. Highlight the `REA_U4` constraint just created
166. From the Edit Menu toolbar, select **Copy Constraint** button
167. From the Edit Menu toolbar, select **Paste Constraint** button
168. Highlight the `Copy of REA_U4` constraint just created

169. Push the **Modify Constraint** button in the Edit menu toolbar
170. Change the name to `REA_V4`
171. Push **Next>**
172. Push **Next>**
173. Select the **Translation 2** radio button in Component
174. Push the **Finish** button

Create Independent Design Variables for the rod areas

175. From the **Design** category chooser, select **Design Variables**
176. Push the **New Design Variable** button from the Edit menu toolbar
177. Enter `A1` in the name field
178. Select the **Independent Design Variable** radio button
179. Push **Next>**
180. Enter `1.0` as **Initial Value**, `0.1` as **Lower Bound**, and `100.0` as **Upper Bound**
181. Push the **Finish** button

    Notice that there is an asterisk in front of the independent design variable, it indicates that this design variable is not being used. The "**I**" indicates this is an **I**ndependent design variable.

Create 2 more Design Variables using the following table

Design Variable Name	Initial Value	Lower Bound	Upper Bound
A2	2.0	0.1	100.0
A3	1.0	0.1	100.0

Define the Sizing Data

182. From the **Design** category chooser, select **Sizing**
183. Select PROD 11 group
184. Push the **Modify Sizing Design** button
185. From the **Area** category chooser, select `1 A1`
186. Push the **Finish** button
187. Select PROD 12 group
188. Push the **Modify Sizing Design** button

189. From the **Area** category chooser, select `2 A2`
190. Push the **Finish** button
191. Select PROD 13 group
192. Push the **Modify Sizing Design** button
193. From the **Area** category chooser, select `3 A3`
194. Push the **Finish** button

Save the Design Studio file

195. From the main menu bar, select **File** → **Save As...**
196. Enter `SZDSG007` as the Filename and push **Save (**as a Design Studio File)

Optimize the structure using Genesis

197. From the main menu bar, select **Genesis** →**Optimize**
198. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

199. From the main menu bar, select **File** →**Import** →**Punch/Output2 Results...**
200. Select the `SZDSG007_dsg00.pch` file
201. Check the **Import Similar Results for All Design Cycles** check box

    This will allow all similar files, to import in one shot.
202. Push the **Open** button

Post-Process the Temperature Results

203. Select the **Post** tab
204. Push the **Deform Mesh/Color Mesh**
205. In the **Color Mesh** window, select **Filled Contours**
206. Select each cycle from the **Color Mesh** window and study it
207. Right-click in the viewport, select **List Top Ten**

    The list is printed in the Messages window.
208. Push **Up** when finished

Study the Output File

209. Start any text editor
210. In the text editor load the Genesis data file: `SZDSG007_dsg.out`
211. Study the file by listing the objective values
212. Study the design variables values
213. Close the file

Quit Design Studio

214. From the main menu bar, select **File** → **Quit**
215. Push the **Don’t Save** button

4.8 Quick Sizing Shell Elements - Design of a Pallet Lifter

Introduction

The purpose of this example is to introduce the basic steps to create and solve a simple sizing optimization problem. This example will first show how to create a pressure load set and add it to an existing loadcase. Second, this example will show how to make sizing regions designable, as well as how to create an objective function and constraints. Finally, this example will show how to visualize sizing results.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

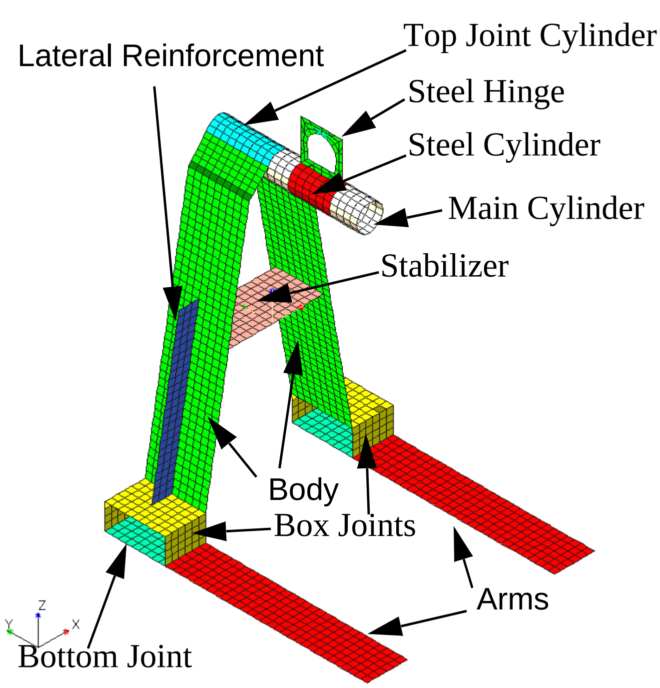
Stress $\leq$ 20,000 psi (all elements)

Magnitude of tip displacements $\leq$ 0.1 (4 tip grids: 13579, 13765, 14055 & 14235)

Designable region:

All aluminum members

The following picture shows the pallet lifter structure and its components.



Example ID

SZDSG008

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SZDSG008.dat	Input data	Provided: Contains the finite element mesh of a pallet lifter.
SZDSG008_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in SZDSG008.dat.
SZDSG008_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
SZDSG008_dsg.HIS	History file	Generated by a Genesis run within Design Studio. This file contains the objective and design variable histories. Design Studio uses this file for plots.
SZDSG008_dsgxy.pch	Punch file	Generated using Genesis within Design Studio. This file contains FE post-processing results for analysis. In this case, the FE results are displacement and stresses.
SZDSG008_dsgOPOSTxy.pch	Punch file	Multiple files generated using Genesis within Design Studio. These files contain the thicknesses for all shell elements for post-processing all design cycles.
SZDSG008_dsgUPDATExx.dat	Input data	Generated using Genesis within Design Studio. This file contains the updated PSHELL properties for the last design cycle xx.
SZDSG008_dsgSSOLxx.dat	Input data	Generated using Genesis within Design Studio. This file contains solid elements that represent the shell elements. This file is for visualization purpose and not intended for analysis. This file is for the last design cycle xx.
SZDSG008_ref.dat	Input data	Provided. Result input file. Should be nearly identical to SZDSG008_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: SZDSG008.dat

Check the Element Norms

3. Select the **Analysis** tab
4. From the category chooser, select **Elements**
5. From the Viewport, select all the displayed elements

6. Push the **Generate Orientation Vectors** button

   Verify that the orientation vectors of the elements in the arms of the lifter point upward.

Clearing the Selection

7. Right-click the Viewport, select **Clear** → **All**

Create the Static Loads

You will create pressure loads on the arms of the structure.

8. From the **Analysis** category chooser, select **Static Loads**
9. Push the **New Load Set** button from the Edit menu toolbar
10. Enter Name `Pressure_Load_Set`
11. Push **Next>**
12. Push **Next>**
13. From the Viewport, select all the elements of the arms

    Pay attention to the label that indicates the number of elements selected. Now it should be 360; if not, push the **Select None** button and repeat the element selection until only the desired number of elements is selected.
    Another way to select the elements in the arms is by hiding all the other groups except for the group named `Arms`. Select the **Display** tab and push the **Show/Hide Groups** and hide the other groups except the one named `Arms`. Select Analysis tab and select Static Loads from the category chooser to return back to the static load creation trail. Push the button **Select All** to select the all the elements that are displayed in the viewport.

14. Enter `-1.42` in the **Pressure** field

    Here you use negative pressure loads to make the load point downward which is opposite from the element orientation.

15. Push the **Add Pressure** button

    Pay attention to the label that indicates the number of pressures on the elements. It should say **360 pressures on 360 elements;** if not, push the **Cancel** button and repeat the steps above to start adding the element pressures once again.

16. Push the **Finish** button

    Verify that now there is one static load set in the **Static Loads** list.

Modify the Default Loadcase

The load set you just created is not used unless it is placed in a loadcase. You will now create one loadcase that uses the previously created load set.

17. Select the **Analysis** tab

18. From the category chooser, select **Loadcases**
19. Select the default loadcase
20. Push the **Modify Loadcase** button in the Edit menu toolbar
21. Enter Name `Pressure_Load_Loadcase`
22. Push **Next>**
23. For **SPC**, select the existing boundary conditions `SPC Set 1`
24. Push **Next>**
25. For Load Set: Select `Pressure_Load_Set`
26. Push **Next>**
27. For Displacement: Select `Post`
28. For Element Stress: Select `Post`
29. Push the **Finish** button

    Verify that the loadcase you just modified is in the list of Loadcases.
    Note that there is a **STAT** label on the left part of the line item. STAT refers to static loadcase.

Defining Sizing Optimization

30. Select the **Design** tab
31. From the category chooser, select **Quick Setup Trails**

    A summary of the design data is shown in the panel. At this point, there is no designed data.

32. Push the **Quick Sizing Setup** button
33. Select PSHELL 3, 4, 5, 6, 23, 28, 33 and 36 from the property list (make sure that PSHELL 37 and 43 are not selected as these components will not be designed). To select one PSHELL at a time hold the **Ctrl** key while selecting the PSHELLs.

    All of the elements picked out in the group are highlighted.

34. Push **Next>**

    Now you will be asked to enter parameters to define the initial value, the lower and the upper bounds of the design variables. All three values are determined by the thickness value multiplied by Mx (multiplier) plus Cx (constant). For example, if you are designing shell thickness which is defined as 1.0 in the shell group property, the initial thickness T0, the lower and the upper bounds TL and TU are calculated by T0=1.0*M1+C1, TL=1.0*M2+C2, and TU=1.0*M3+C3 respectively. MinLB and MaxUB are absolute minimum and maximum regardless of the current group property values.

35. Accept defaults of `0.0` in C1 and `1.0` in M1 fields

    This will make the initial value to be the same, as specified by the analysis data.

36. Enter `0.01` in C2 field and `0.0` in M2 field

    This will make the lower bound value to be 0.01 for all designable thicknesses.

37. Enter `2.0` in C3 field and `0.0` in M3 field

    This will make the upper bound value to be 2.0 for all designable thicknesses.

38. Push the **Finish** button

    Summaries are shown in the panel. Now there are 8 design variables and 8 sizing regions.

Verify the Recently Created Design Variables

39. From the **Design** category chooser, select **Design Variables**

    You should now see 8 design variables:

DESIGN VARIABLE	Initial Value	Lower Bound	Upper Bound
1	0.5	0.01	2.0
2	0.375	0.01	2.0
3	0.1875	0.01	2.0
4	0.1875	0.01	2.0
5	0.375	0.01	2.0
6	0.6875	0.01	2.0
7	0.5625	0.01	2.0
8	0.1875	0.01	2.0

Note:

The format for the design variables in Genesis is:

DVAR	ID	LABEL	INIT	LB	UB				

You do not need to memorize or study this format in detail, as Design Studio takes care of it. Here it is written for your information only.

The following information will be written in the Genesis input data:

DVAR	1	T3	0.5	0.01	2.0				
DVAR	2	T4	0.375	0.01	2.0				
DVAR	3	T5	0.1875	0.01	2.0				
DVAR	4	T6	0.1875	0.01	2.0				
DVAR	5	T23	0.375	0.01	2.0				

DVAR	6	T28	0.6875	0.01	2.0				
DVAR	7	T33	0.5625	0.01	2.0				
DVAR	8	T36	0.1875	0.01	2.0				

Verify the Recently Created Sizing Data

40. From the **Design** category chooser, select **Sizing**

    Eight designable regions should have a hammer indicating that they are designable:

41. Select PSHELL 3

    Read the bottom of the main menu. There you should be able to find information on the design variable associated with PSHELL 3.

42. Push the **Modify Sizing Design** button

    Verify that Design variable 1 (T3) is being used to design the thickness.

43. Push the **Cancel** button

    If you verify all designed PSHELLs, you will find the following:

PSHELL ID (Provided)	**DESIGN VARIABLE (Automatically Generated)**
3	1
4	2
5	3
6	4
23	5
28	6
33	7
36	8

Note:

The following information will be written in the Genesis input data:

DVPROP3	1	3	SOLID	1					
+	1								
DVPROP3	2	4	SOLID	1					
+	2								
DVPROP3	3	5	SOLID	1					

+	3								
DVPROP3	4	6	SOLID	1					
+	4								
DVPROP3	5	23	SOLID	1					
+	5								
DVPROP3	6	28	SOLID	1					
+	6								
DVPROP3	7	33	SOLID	1					
+	7								
DVPROP3	8	36	SOLID	1					
+	8								

Defining the Design Objective

44. From the **Design** category chooser, select **Objectives**
45. Push the **New Objective** button from the Edit menu toolbar
46. Accept the default, **Mass** response type
47. Accept the **Min** Objective Definition switch
48. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Note:

The following information will be written in the Genesis input data:

DOBJ	1	Obj1							
DRESP1	1	Obj1	MASS						

Defining the Design Stress Constraints

49. From the **Design** category chooser, select **Constraints**
50. Push the **New Constraint** button from the Edit menu toolbar
51. For **Name,** enter `Stresses`
52. Select the **Stress** response
53. Select **Selected Groups** option of **Stress**
54. Enter `20000.0` for the **Upper Bound**
55. Push **Next>**

56. Select PSHELL 3, 4, 5, 6, 23, 28, 33 & 36
57. Push **Next>**
58. For **PSHELL STRESS** accept the default, **von Mises Top & Bottom**
59. Push **Next>**
60. Select the static loadcase
61. Push the **Finish** button

    Verify that now there are two responses in the constraint list. One corresponds to the von Mises stress at the top of the shell elements. The other corresponds to von Mises stress at the bottom of the shell elements.

Note:

The following information will be written in the Genesis input data:

DCONS	2			20000.0					
DRESP1	2	Cons1	STRESS	PSHELL		2	3	4	
+	5	6	23	28	33	36			
DCONS	3			20000.0					
DRESP1	3	Cons1	STRESS	PSHELL		9	3	4	
+	5	6	23	28	33	36			

Defining the Displacement Constraints

62. Push the **New Constraint** button from the Edit menu toolbar
63. For **Name** enter `Displacements`
64. Select the **Displacement** response
65. Enter `0.1` for the **Upper Bound**
66. Push **Next>**
67. Select the following grids: `13579`
68. Push the **Add** button
69. Select the following grids: `13765`
70. Push the **Add** button
71. Select the following grids: `14055`
72. Push the **Add** button
73. Select the following grids: `14235`

74. Push the **Add** button

    Verify that there is a label at the bottom of the page that says: **4 grids selected.**

75. Select Component, **Translational Magnitude**

    This corresponds to the magnitude of the displacements.

76. Push **Next>**

77. Select the static loadcase

78. Push the **Finish** button

    Verify that now there are three responses in the constraint list.

Note:

The following information will be written in the Genesis input data:

DCONS	4			0.1					
DRESP1	4	Cons3	DISP			7	13579	13765	
+	14055	14235							

Clearing the Selection

79. Right-click the Viewport, select **Clear** → **All**

Request the SSOL File

The SSOL file is a Genesis input file where the shell elements are converted into solid elements for the purpose of better visualizing the thicknesses distribution. This file is not intended to be used for analysis or optimization.

80. From the main menu bar, select **Genesis** → **Options...**

81. Select the **File Control** tab

82. For **Shell-to-Solid File**, select **Create(Fixed Norms)**

    The name of the SSOL file will be:
    `SZDSG008_dsgSSOLxx.dat` (where xx corresponds to the last design cycle number).

83. Push the **Apply** button

Note:

The following information will be written in the Solution Control of the Genesis input data:

SSOL=YES,FIXNORM

Request the UPDATE File

The UPDATE file is a Genesis input file that contains the updated data entries. In this case the key updated entries are the PSHELLs.

84. From the main menu bar, select **Genesis** → **Options**
85. Select the **File Control** tab
86. For **Updated Input File**, select **First & Last**

    First & Last option will create two updated files, one corresponding to the first design cycle the other corresponding to the last. The name of the file will be: `SZDSG008_dsgUPDATE00.dat` and `SZDSG008_dsgUPDATExx.dat` (where xx is the last design cycle number).
87. Push the **Apply** button

Note:

The following information will be written in the Solution Control of the Genesis input data:

UPRINT=FLAST

Request the OPOST (Sizing data) Post-Processing File

The OPOST file is a Genesis post-processing file that contains the optimization results, in this case the thicknesses of the shell elements.

88. From the main menu bar, select **Genesis** → **Options**
89. Select the **File Control** tab
90. For **Element Sizing File**, select **Create**
91. Select the **Output Control** tab
92. For **Design Output**, select **All Cycles**

    The thicknesses will be printed for all design cycles.
    The name of the post-processing file will be:
    `SZDSG008_dsgOPOSTxy.pch` (where xy is the design cycle number).

93. Push the **Apply** button

Note:

The following information will be written in the Solution Control of the Genesis input data:

DPRINT=FLAST
SIZING=POST

Optimize the Structure Using Genesis

94. From the main menu bar, select **Genesis** → **Optimize**
95. Study the **Design History** charts; when done, push the **Close** button
96. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Analysis Post-Processing Files

97. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
98. Select the `SZDSG008_dsg00.pch` file and put a checkmark in the **Import Similar Results for All Design Cycles** checkbox

    Putting a checkmark in the checkbox will cause Design Studio to load many result files (one for each design cycle) in one step.

99. Push the **Open** button

Post-Processing some Analysis Results (Stresses)

100. Select the **Post** tab
101. Push the **Deform Mesh/Color Mesh** button
102. Select a Shell Stress Result for any design cycle
103. Select a Shell Stress Result for the last design cycle
104. Push the **Options...** button. Slide the bar to hide elements with low values
105. Push the **Close** button
106. Push the **Smooth Shaded** icon in the panel below
107. Push the **Flat Shaded** icon in the panel below
108. Push the **Shaded Feature** icon in the panel below

109. Right-click in the viewport, select **List Top Ten**

The list is printed in the Messages window.

110. Right-click in the viewport, select **List Bottom Ten**

The list is printed in the Messages window.

111. Change the stress displayed from **max von Mises** to **von Mises at z1**
112. Change the stress displayed from **von Mises at z1** to **von Mises at z2**
113. Fill in the following Table:

Location	Max von Mises Stress Design Cycle 0 Reference Solution (1)	Max von Mises Stress Last Design Cycle Reference Solution (1)	Max von Mises Stress Design Cycle 0	Max von Mises Stress Last Design Cycle
Z1 (bottom)	5026.9	5030.0		
Z2(top)	5043.0	5047.0		

(1) Result from running `SZDSG008_ref.dat`

The values in this table and the ones you obtain should be very similar.

114. Are the stresses feasible in design cycle 0?
115. Are the stresses feasible in the last design cycle?

Hint: the upper bound on the stress constraints is 20,000.00 psi.

Post-Processing some Analysis Results (Displacements)

116. Push the **Clear** button in the Color Mesh
117. Select a Displacement Result for any design cycle
118. Select the Displacement Result for the last design cycle
119. Push the **Oscillate** button
120. Push the **Filled Contour** button in the **Color Mesh**
121. In the Color Mesh list, select a Displacement Result for any design cycle
122. In the Color Mesh list, select the Displacement Result for the last design cycle
123. Right-click in the viewport, select **List Top Ten**
124. Right-click in the viewport, select **List Bottom Ten**

125. Fill in the following Table:

Type	XYZ Magnitude Design Cycle 0 Reference Solution (1)	XYZ Magnitude Last Design Cycle Reference Solution (1)	XYZ Magnitude Design Cycle 0	XYZ Magnitude Last Design Cycle R
Max	0.066	0.1		

(1) Result from running `SZDSG008_ref.dat`

The values in this table and the ones you obtain should be very similar.

126. Are the displacements OK in design cycle 0?
127. Are the displacements OK in the last design cycle?

Hint: the upper bound on the displacement constraints is 0.1.

128. Which constraints are controlling the design, stress or displacements?
129. Push the **Up** button

Import the Design Post-Processing Files

130. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
131. Select the `SZDSG008_dsgOPOSTS00.pch` file and put a checkmark in the **Import Similar Results for All Design Cycles** checkbox

Putting a checkmark in the checkbox will cause Design Studio to load many result files (one for each design cycle) in one step.

132. Push the **Open** button

Post-Processing the Results (Thicknesses on Shells)

133. Select the **Post** tab
134. Push the **Deform/Mesh Color Mesh** button
135. Select a Thickness Result for any design cycle
136. Select a Thickness Result for the last design cycle
137. Push the **Smooth Shaded** icon in the panel below
138. Push the **Shaded Feature** icon in the panel below
139. Right-click in the viewport, select **List Top Ten**

The list is printed in the Messages window.

140. Right-click in the viewport, select **List Bottom Ten**

    The list is printed in the Messages window.

141. Right-click in the viewport, select **List By Group...**
142. For **Group Result Method**, select **Max Element Result in Group** option
143. For **Result to List**, select **Top 10 Displayed Groups** option

    The list is printed in the Messages window. Doing this would print the top ten groups with the maximum thickness value for the group.

144. Fill in the following Table:

Type	Thickness Design Cycle 0 Reference Solution (1)	Thickness Last Design Cycle Reference Solution (1)	Thickness Design Cycle 0	Thickness Last Design Cycle R
Maximum	0.67	0.515		
Minimum	0.1675	0.096		

(1) Result from running `SZDSG008_ref.dat`

The values in this table and the ones you obtain should be very similar.

145. Push the **Up** button

Import the History File

146. From the main menu bar, select **File** → **Import** → **Design History Results...**
147. Select the `SZDSG008_dsg.HIS` file
148. Push the **Open** button

Post-Processing the Histories

149. Select the **Post** tab
150. Push the **Design History Plots** button
151. Select the Design History `SZDSG008_dsg.HIS`
152. Push the **New Plot** button
153. Select all design variables
154. Push the **Finish** button
155. Study the **Design History Plot;** when done, push the **Close** button

156. Push the **Up** button

Import the Solid File

157. From the main menu bar, select **File** → **New**
158. Import the Genesis data file: `SZDSG008_dsgSSOLxx.dat` (xx corresponds to the last design cycle)

Re-Import the Post-Processing Files

159. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
160. Select the `SZDSG008_dsgOPOSTSxx.pch` file (xx corresponds to the last design cycle)

    If you can not see this file and/or to have all file listed: Select **All Files** in the drop-down menu **Files of Type.**

161. Push the **Open** button

Post-Processing the Design Results (Thicknesses on Solids)

162. Select the **Post** tab
163. Push the **Deform/Mesh Color Mesh** button
164. Select the Thickness Results for the last design cycle

Creating a Picture File

Select a good view of the results.

165. From the main menu bar, select **File** → **Print to Image File**
166. Push the **Save** button

    Check your working directory. A picture named `SZDSG008_dsgSSOLxx.png` should be there.

Use One Color

167. Push the **Up** button
168. From the main menu bar, select **Color** → **Group Color Style** → **All One Color**
169. Select the **Display** tab
170. From the **Group Display Style**, select **Flat Shaded**
171. Rotate the model and study the results

Quit Design Studio

172. From the main menu bar, select **File** → **Quit**
173. Push the **Don’t Save** button

Study the Output File

174. In a text editor load the Genesis data file: `SZDSG008_dsg.out`
175. Study briefly the file
176. Using the output file, complete the following table:

Design Variable ID	Original Design Variable Value Reference Solution (1)	Final Design Variable Value Reference Solution (1)	Final Design Variable Value (2)
1	0.5	0.442	
2	0.375	0.268	
3	0.1875	0.096	
4	0.1875	0.133	
5	0.375	0.199	
6	0.6875	0.509	
7	0.5625	0.250	
8	0.1875	0.123	

(1) Result from `SZDSG008_ref.dat`
(2) Result from your run, `SZDSG008_dsg.out`

177. Complete the following table:

	Sizing Reference (1)	Sizing (2)
Number of Design Variables	8	
Initial Mass	2.307E-2	
Optimal Mass	1.768E-2	

(1) Result from running `SZDSG008_ref.dat`
(2) Result from your run, `SZDSG008_dsg.out`

178. Compare the optimal mass with the original structure

Reference answer: (1.768E-2-2.307E-2)/2.307E-2*100= -23% (reduction)

Study the Initial Updated file

179. In a text editor load the Genesis data file: `SZDSG008_dsgUPDATE00.dat`

180. Study briefly the file

181. Using the PSHELL data entries, complete the following table:

PSHELL ID	Thickness Reference Solution (1)	Thickness (2)
3	0.5	
4	0.375	
5	0.1875	
6	0.1875	
23	0.375	
28	0.6875	
33	0.5625	
36	0.1875	

(1) Result from `SZDSG008_ref.dat`
(2) Result from your run, `SZDSG008_dsgUPDATE00.out`

Study the Final (Optimized) Updated file

182. In a text editor load the Genesis data file: `SZDSG008_dsgUPDATExx.dat` (xx corresponds to the last design cycle)

183. Study briefly the file

184. Using the PSHELL data entries, complete the following table:

PSHELL ID	Optimal Thickness Reference Solution (1)	Optimal Thickness (2)
3	0.442	
4	0.268	
5	0.096	
6	0.133	
23	0.199	
28	0.509	
33	0.250	
36	0.123	

(1) Result from `SZDSG008_ref.dat`
(2) Result from your run, `SZDSG008_dsgUPDATExx.out`

4.9 Discrete Optimization - Three Rod Truss

Introduction

The purpose of this exercise is to create and solve a sizing optimization problem using discrete design variables. A three rod truss is used for the analysis. The cross-sectional areas of the rods are designed using discrete design variables.

The following optimization problem will be created and solved:

Minimize Mass

Subject to:

-15000.0 ≤ Stress in the ROD elements ≤ 20000.0

Translation Displacement ≤ 0.20

Designable region:

Cross-sectional areas of the ROD elements

Example ID

`SZDSG009`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SZDSG009.dat`	`Input data`	Provided: Contains the finite element mesh of a portal frame with applied load and boundary conditions.
`SZDSG009_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `SZDSG009.dat`.
`SZDSG009_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SZDSG009_ref.dat`	`Input data`	Provided. Reference result input file. Should be nearly identical to `SZDSG009_dsg.dat`. This file is provided to check your example.

Start Design Studio

1. Start Design Studio

2. Load the Genesis data file: `SZDSG009.dat`
3. Select the XY icon in the Viewport to change to Top view

Review the Existing Loadcase

4. From the **Analysis** category chooser, select **Loadcases**
5. Select the first existing loadcase and study its loading in the viewport
6. Select the second existing loadcase and study its loading in the viewport

Create Three Discrete Design Variables

7. From the **Design** category chooser, select **Design Variables**
8. Push the **New Design Variable** button from the Edit menu toolbar
9. Enter `A1` in the name
10. Select **Discrete Design Variable** radio button
11. Push the **Next>** button.
12. Enter `1.0` for **Initial Value**, `0.1` for **Lower Bound**, and `100.0` for **Upper Bound**.
13. Under **Generate Discrete Values**, enter `0.1` for **First Discrete Value**, `0.1` for **Increment**, and `999` for **Num. of additional values**
14. Push the **Generate & Add Values** button
15. Push **Finish**
16. Push the **New Design Variable** button from the Edit menu toolbar
17. Enter `A2` in the name
18. Select **Discrete Design Variable** radio button
19. Push the **Next>** button.
20. Enter `2.0` for **Initial Value**, `0.1` for **Lower Bound**, and `100.0` for **Upper Bound**.
21. Under **Generate Discrete Values**, enter `0.1` for **First Discrete Value**, `0.1` for **Increment**, and `999` for **Num. of additional values**
22. Push the **Generate & Add Values** button
23. Push **Finish**
24. Select the design variable `A1` created previously
25. From the Edit Menu toolbar, select **Copy Design Variable** button
26. From the Edit Menu toolbar, select **Paste Design Variable** button

27. Select the `Copy of A1` design variable and push the **Modify Design Variable** button in the Edit menu toolbar
28. For the name, enter `A3`
29. Push **Finish**

    Notice that there is an asterisk in front of all the design variable, which indicates that this design variable is not being used.
    The **"D"** after the asterisk indicates that it is an discrete design variable.

Defining Sizing Optimization

30. From the **Design** category chooser, select **Sizing**
31. Select the PROD 11 property in the list
32. Push the **Modify Sizing Design** button to add design attributes.
33. For the **Area**, select the design variable `A1`
34. Push **Finish**
35. Select the PROD 12 property in the list
36. Push the **Modify Sizing Design** button to add design attributes.
37. For the **Area**, select the design variable `A2`
38. Push **Finish**
39. Select the PROD 13 property in the list
40. Push the **Modify Sizing Design** button to add design attributes.
41. For the **Area**, select the design variable `A3`
42. Push **Finish**

    You will see the hammer icon next to the group in the list. This indicates that the group is being size-designed.

Review the Design Objective

43. From the **Design** category chooser, select **Objectives**
44. Select the existing objective
45. Push **Modify Objective** button in the Edit menu toolbar and study the objective to minimize the mass.
46. When done, push the **Cancel** button

Review the Design Constraints

47. From the **Design** category chooser, select **Constraints**
48. Study the constraints defined in the list

Change Maximum Design Cycles

49. From the main menu bar, select **Genesis** → **Options...**
50. Select the **Design Control** tab
51. Select the checkbox for **Maximum Design Cycles** and enter `15`
52. Push the **Apply** button

Optimize the Structure Using Genesis

53. From the main menu bar, select **Genesis** → **Optimize**

    Genesis will start optimizing the model. The Genesis Console Output dialog and Design History dialog will be shown and you can monitor the optimization progress. Once the optimization is finished, you can view the output file by pushing the **View Output File** button or you can close the dialogs by pushing the **Close** button.

Quit Design Studio

54. From the main menu bar, select **File** → **Quit**
55. Push the **Don't Save** button

Study the Output File

56. Start any text editor
57. In the text editor load the Genesis data file: `SZDSG009_dsg.out`
58. Navigate to the end of the file for a listing of the objective values in each design cycle
59. Study the design variables values changes in each design cycle
60. Study the discrete variables violation history table
61. Close the file

4.10 Design Based on Random Response

Introduction

The purpose of this exercise is to create and solve a sizing optimization problem using response based on random analysis.

The following optimization problem will be created and solved:

Minimize Mass

Subject to:

Root Mean Square Velocities at corner grids $\leq$ 100.0

Designable region:

Thickness of the Structure

Example ID

`SZDSG010`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SZDSG010.dat`	`Input data`	Provided: Contains the finite element mesh of a portal frame with applied load and boundary conditions.
`SZDSG010_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `SZDSG010.dat.`
`SZDSG010_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SZDSG010_ref.dat`	`Input data`	Provided. Reference result input file. Should be nearly identical to `SZDSG010_dsg.dat`. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Load the Genesis data file: `SZDSG010.dat`

3. Select the XY icon in the Viewport to change to Top view

Review the Sizing Data

4. From the **Design** category chooser, select **Quick Setup Trails**
5. From the summary, notice that 8 design variables are already defined and associated with 8 Sizing regions.
6. Study the design variables by selecting **Design Variables** from the **Design** category chooser
7. Study the sizing regions by selecting **Sizing** from the **Design** category chooser

Define the Design Objective

8. From the **Design** category chooser, select **Objectives**
9. Push **New Objective** button from the Edit menu toolbar
10. Enter `MASS` in the name field
11. Select the **Mass** radio button as Response Type
12. Check the **Min** radio button as Objective Definition Switch
13. Push the **Finish** button

Define the Random Response (RMSVELO) Constraints

14. From the **Design** category chooser, select **Constraints**
15. Push the **New Constraint** button from the Edit menu toolbar
16. Enter `RMSVELO` in the name field
17. Select the **More Response Types** radio button as the Response Type
18. Enter `100.0` as **Upper Bound**
19. Push **Next>**
20. Select the **Dynamic Velocity** radio button and select **Random RMS (RMSVELO)** option from the listbox
21. Push **Next>**

    Now the 10 grid location for the velocity calculations should be chosen. These grids are already defined in a grid set. This grid set (`Grid Set 4`) is used to make the selection.

22. Select the **Analysis** tab and select **Grid Sets** from the **Analysis** category chooser

23. Select `Grid Set 4` from the list

    Notice that 10 grids are selected in the Viewport

24. Select the **Design** tab

    Design Studio will return to the same location in the Constraints trail.
    Notice that the 10 grids remain selected making it easier for defining the constraint.

25. Select the **Translation 3** radio button

26. Push **Next>**

27. Select the existing loadcase

28. Push the **Finish** button

Request the Element Sizing Data File to be Output

29. From the main menu bar, select **Genesis** → **Options...**

30. Select the **File Control** tab

31. For **Element Sizing File**, make sure that the **Create** option is selected

32. Push the **Apply** button

Optimize the Structure Using Genesis

33. From the main menu bar, select **Genesis** → **Optimize**

    Genesis will start optimizing the model. The Genesis Console Output dialog and Design History dialog will be shown and you can monitor the optimization progress. Once the optimization is finished, you can view the output file by pushing the **View Output File** button or you can close the dialogs by pushing the **Close** button.

Importing the Sizing Results

34. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**

35. Select `SZDSG010_dsgOPOST00.pch` from the file browser.

36. Check the **Import Similar Results for All Design Cycles** check box, then push the **Open** button.

    This will allow all similar files to import in one shot.

Viewing the Sizing Results

37. Select the **Post** tab.

38. Push **Deform Mesh/Color Mesh...** button

39. For the **Color Mesh** list, select a PSHELL Thickness result to plot.

    Compare the thickness values from the first to the last design cycle. Initially, all the shells have the same thickness of 1.0 while in the final design the areas are different.

40. Click on an element in the Viewport to see the value of the thickness in the Design Studio messages window.

Quit Design Studio

41. From the main menu bar, select **File** → **Quit**
42. Push the **Don't Save** button

Study the Output File

43. Start any text editor
44. In the text editor load the Genesis data file: `SZDSG010_dsg.out`
45. Navigate to the end of the file for a listing of the objective values in each design cycle
46. Study the design variables values changes in each design cycle
47. Close the file

CHAPTER 5

Topometry Optimization Examples

- Topometry Optimization of a Flat Plate
- Topometry Optimization of a Hat Structure
- Topometry Optimization of a Pallet Lifter Using Symmetries
- Coarse Topometry Optimization of a Pallet Lifter
- Creating a CAD Mesh that Represents Results of a Topometry Optimized Shell Structure
- Creating a Finite Element Mesh that Represents Results of a Topometry Optimized Structure using Re-grouping
- Reinforcing a Solid Part on a Design Studio Generated Composite Skin Layer
- Topometry Optimization of Welds
- Mode Shape Optimization
- Optimization of Launch Vehicle to Maximize Frequency
- Re-grouping based on Topometry Optimization Results

5.1 Topometry Optimization of a Flat Plate

Introduction

The purpose of this example is to learn how to create a basic topometry optimization problem. After you finish this example, you will know how to plot thickness distributions and how to use the SSOL option to convert a shell mesh into solid mesh to visualize the thickness distribution physically.

The following optimization problem will be created, solved and post-processed:

Minimize Strain Energy

Subject to:

Volume $\leq$ 120.0

Designable region:

The thickness of each element

Example ID

TMDSG001

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
TMDSG001.dat	Input data	Provided: Contains the finite element mesh of a flat plate with applied load and boundary conditions.
TMDSG001_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in TMDSG001.dat.
TMDSG001_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
TMDSG001_dsg.HIS	History file	Generated by a Genesis run within Design Studio. This file contains the objective and design variable histories. Design Studio uses this file for plots.
TMDSG001_dsgOPOSTxy.pch	punch file	Multiple files generated using Genesis within Design Studio. These files contain the thicknesses for all shell elements for post-processing all design cycles.

`TMDSG001_dsgSSOLxx.dat`	`Input data`	Generated using Genesis within Design Studio. This file contains solid elements that represent the shell elements. This file is for visualization purpose and not intended for analysis. This file is for the last design cycle xx.
`TMDSG001_ref.dat`	`Input data`	Provided. Reference result input file. Should be nearly identical to `TMDSG001_dsg.dat`. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG001.dat`

Set up Sizing Optimization Data

3. Select the **Design** tab
4. From the **Design** category chooser, select **Quick Setup Trails**
5. Push the **Quick Sizing Setup** button
6. Select PSHELL 4
7. Push **Next>**
8. For **C1** and **M1**: accept defaults
9. For **C2** and **M2**: Enter `0.1` and `0.0` respectively
10. For **C3** and **M3**: Enter `2.0` and `0.0` respectively

    C1, **M1**, **C2**, **M2**, **C3** and **M3** are the parameters to define the initial value, the lower and the upper bounds of the design variables. All three values are determined by the value on the property card multiplied by Mx (multiplier) plus Cx (constant). MinLB and MaxUB are absolute minimum and maximum regardless of the current group property values.

11. Push the **Finish** button

    Verify that 1 design variable and 1 sizing region are defined

Set up Topometry Optimization Data

12. From the **Design** category chooser, select **Topometry**
13. Select PSHELL 4
14. Push the **Modify Topometry Design** button from the Edit menu toolbar

    “Coarsening” is used in topometry so that elements are linked with the same design variable. One can view this as a group by group sizing optimization as opposed to element by element sizing optimization when no coarse option is used. The groups are automatically created based on the user specified number that indicates the maximum number of elements permitted in each group. The groups are formed so that elements that are adjacent to each other are grouped together. Coarse topometry, is a compromise between sizing and standard topometry

optimization.
The main purpose of using the coarse option is to reduce the number of design variables thereby reducing the computational time.

15. For **Coarse Method:**, choose the **Max elements per design variable** option
16. For **Coarse Parameter:** enter `4`
17. Push the **Finish** button

    Verify that the hammer icon is now next to the PSHELL label.
    Here, the hammer means that the group is being designed via topometry.

Define the Design Objective

18. From the **Design** category chooser, select **Objectives**
19. Push **New Objective** button from the Edit menu toolbar
20. Select the **Strain Energy** response
21. Push **Next>**
22. Select the existing load case
23. Push **Finish**

    Verify that now there is 1 response in the objectives list

Define the Design Constraint

24. From the **Design** category chooser, select **Constraints**
25. Push **New Constraint** button from the Edit menu toolbar
26. Select the **Volume** response and accept the default **Entire Model**
27. Enter `120.0` as **Upper Bound**
28. Push **Finish**

    Verify that now there is 1 response in the constraints list

Request the Element Sizing File to be Output

29. From the main menu bar, select **Genesis** → **Options...**
30. Select the **File Control** tab
31. For **Element Sizing File**, choose the **Create** option
32. Push the **Apply** button

Request Only the First and Last Design Cycle Data to be Output

33. From the main menu bar, select **Genesis** → **Options...**
34. Select the **Output Control** tab
35. For **Analysis Output**, choose the **First & Last** option
36. For **Design Output**, choose the **First & Last** option
37. Push the **Apply** button

Request the SSOL File

The SSOL file is a Genesis input file where the shell elements are converted into solid elements for the purpose of better visualizing the thicknesses distribution. This file is not intended to be used for analysis or optimization.

38. From the main menu bar, select **Genesis** → **Options...**
39. Select the **File Control** tab
40. For **Shell-to-Solid File**, select **Create(Fixed Norms)**

    The name of the SSOL file will be:
    `TMDSG001_dsgSSOLxx.dat` (where xx corresponds to the last design cycle number).
41. Push the **Apply** button

Increase the Maximum Number of Design Cycles

42. From the main menu bar, select **Genesis** → **Options...**
43. Select the **Design Control** tab
44. For **Maximum Design Cycles**, enter `50`
45. Push the **Apply** button

Optimize the Structure Using Genesis

46. From the main menu bar, select **Genesis** → **Optimize**
47. Study the **Design History** charts; when done, push the **Close** button
48. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Design Post-Processing Files

49. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**

50. Select the `TMDSG001_dsgOPOST00.pch` file and put a checkmark in the **Import Similar Results for All Design Cycles** checkbox

    Putting a checkmark in the checkbox will cause Design Studio to load many result files (one for each design cycle) in one step.

51. Push the **Open** button

Post-Processing the Results (Thickness Distribution)

52. Select the **Post** tab
53. Push the **Deform/Mesh Color Mesh** button
54. Select a Thickness Result for any design cycle
55. Select a Thickness Result for the last design cycle
56. Push the **Smooth Shaded** icon in the panel below
57. Push the **Shaded Feature** icon in the panel below
58. Right-click in the viewport, select **List Top Ten**

    The list is printed in the Messages window.

59. Right-click in the viewport, select **List Bottom Ten**

    The list is printed in the Messages window.

60. Push the **Up** button

Import the History File

61. From the main menu bar, select **File** → **Import** → **Design History Results...**
62. Select the `TMDSG001_dsg.HIS` file
63. Push the **Open** button

Post-Processing the Histories

64. Select the **Post** tab
65. Push the **Design History Plots** button
66. Select the Design History `TMDSG001_dsg.HIS`
67. Push the **New Plot** button
68. Select all design variables
69. Push the **Finish** button
70. Study the **Design History Plot;** when done, push the **Close** button

71. Push the **Up** button

Import the Solid File

72. From the main menu bar, select **File** → **New**

    You can save or not save the Design Studio database.

73. Import the Genesis data file: `TMDSG001_dsgSSOLxx.dat` (xx corresponds to the last design cycle)

74. Push the **Open** button

Re-Import the Post-Processing Files

75. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**

76. Select the `TMDSG001_dsgOPOSTxx.pch` file (xx corresponds to the last design cycle)

    If you can not see this file and/or to have all file listed: Select **All Files** in the drop-down menu **Files of Type.**

77. Push the **Open** button

Post-Processing the Design Results (Thicknesses on Solids)

78. Select the **Post** tab

79. Push the **Deform/Mesh Color Mesh** button

80. Select the Thickness Results for the last design cycle

Creating a Picture File

Select a good view of the results.

81. From the main menu bar, select **File** → **Print to Image File**

82. Push the **Save** button

    Check your working directory. A picture named `TMDSG001_dsgSSOLxx.png` should be there.

Quit Design Studio

83. From the main menu bar, select **File** → **Quit**

84. Push the **Don't Save** button

5.2 Topometry Optimization of a Hat Structure

Introduction

The purpose of this example is to introduce the basic steps to create and solve a simple topometry optimization problem. This example will first show how to create static load sets, boundary conditions and static loadcases. Second, this example will show how to make a topometry region designable and how to create an objective function and a constraint. And finally, this example will show how to display topometry results using element thicknesses and animations.

Objective function of the problem:

Minimize the strain energy of the structure

Subject to:

Volume $\leq$ 1600.00

Designable region:

Every element in the structure

Analysis problem:

Point Load
Simply supported in 4-corner grids

Example ID

`TMDSG002`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`TMDSG002.dat`	`Input data`	Provided: Contains the finite element mesh of a hat structure.
`TMDSG002_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `TMDSG002.dat`.
`TMDSG002_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.

TMDSG002_dsg.HIS	History file	Generated by a Genesis run within Design Studio. This file contains the objective and design variable histories. Design Studio uses this file for plots.
TMDSG002_dsgOPOSTxy.pch	punch file	Multiple files generated using Genesis within Design Studio. These files contain the thicknesses for all shell elements for post-processing all design cycles.
TMDSG002_dsgSSOLxx.dat	Input data	Generated using Genesis within Design Studio. This file contains solid elements that represent the shell elements. This file is for visualization purpose and not intended for analysis. This file is for the last design cycle xx.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG002.dat`

Create the Static Loads

You will create a point load located at the center of the front side of the structure.

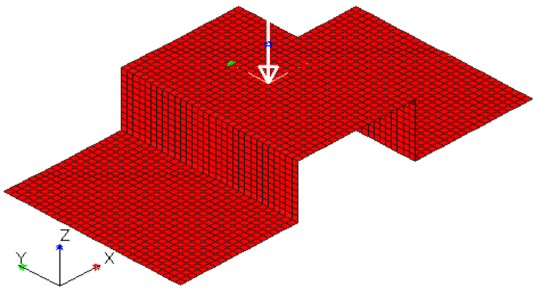
3. Select the **Analysis** tab
4. From the category chooser, select **Static Loads**
5. Push **New Load Set** button from the Edit menu toolbar
6. Enter Name `Point_Load_Set`
7. Push **Next>** (accepting the Force default)

Pay attention to the label that indicates the number of grids selected. Now it should be 0; if not, push the **Select None** button.

Note:

As a general rule, Design Studio keeps all your selections. The reason for keeping all selections is that, in general, it is easier to clear selections than to reproduce them.

8. From the Viewport, select the grid in the middle of the top part, as shown in the following figure:



After you select a grid: Pay attention to the label that indicates the number of grids selected. Now it should be 1; if not, push the **Select None** button and repeat the grid selection until only the desired grid is selected.

9. Enter `-1.0` in the **Z** field
10. Enter `1000.0` in the **Magnitude** field
11. Push the **Add Force** button

    Verify that now there is one force in one grid.

12. Push the **Finish** button

    Verify that now there is one static load set in the **Static Loads** list.

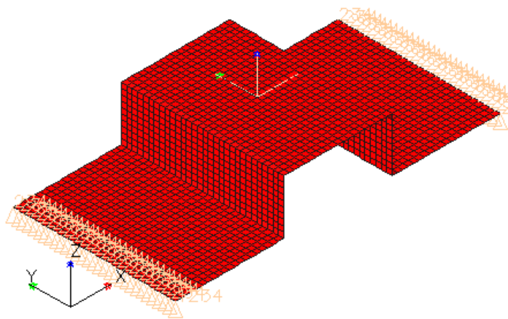
Create the Boundary Conditions

Here, you will create the boundary conditions of the problem. The front and back edges of the structure will be constrained using 234. In addition, one corner of the structure will be constrained in the 1 direction.

13. Select the **Analysis** tab
14. From the category chooser, select **Grid-Component Sets**
15. Push **New Grid-Component Set** button from the Edit menu toolbar
16. Enter Name `Edges_Constraints`
17. Push **Next>** (accepting the SPC default)

    Pay attention to the label that indicates the number of grids selected. Before starting to select nodes it should be 0; if not, push the **Select None** button.
    If you followed the instructions in sequence, you should have had 1 selected grid. That grid corresponds to the grid that you selected for the load.

18. From the Viewport, select all grids that are in the edges, as shown in the following figure:



To select the grids easily you may change view.
Verify that there 62 grids selected.

19. For **Components** enter: `234`

The degrees of freedom of the selected grids that are to be constrained are entered here. The component number for the degrees of freedom are as follows:
1,2,3 are translational degrees of freedom in the x, y, and z-directions respectively
4,5,6 are rotational degrees of freedom about the x, y, and z-axes respectively

20. Push the **Set Components** button

Pay attention to the label that indicates the total number of dof. It should now say **186 total dof on 62 grids**. If it does not say that, push the **Select All**, then push the **Clear Components** button and then push the **Select None** button, after that repeat the process above.

21. Push the **Select None** button

22. From the Viewport, select one of the four corner grids (for example pick the grid with ID 1)

23. For **Components** enter: `1`

24. Push the **Set Components** button

Pay attention to the label that indicates the total number of dof. It should now say **187 total dof on 62 grids**. If it does not say that, push the **Select All,** then push the **Clear Components** button and then push the **Select None** button, after that repeat the process above.

25. Push the **Finish** button

Verify that now there is one Group-Component Set in the list.

Create the Loadcase

The load set and the boundary conditions you just created are not used unless they are in a loadcase. You will now create one loadcase that uses the previous created load set and boundary condition.

26. Select the **Analysis** tab
27. From the category chooser, select **Loadcases**
28. Select the default loadcase
29. Push the **Modify Loadcase** button from the Edit menu toolbar
30. Enter Name `Point_Load_Loadcase`
31. Push **Next>**
32. For SPC: Select `Edges_Constraints`
33. Push **Next>**
34. For Load Set: Select `Point_Load_set`
35. Push **Next>**
36. For Displacement: Select `Post`
37. Push the **Finish** button

    Verify that the loadcase you just created is in the list of Loadcases.
    Note that there is a **STAT** label on the left part of the line item. STAT refers to static loadcase.

Create the Designable Region

38. Select the **Design** tab
39. Select **Quick Setup Trails** from the category chooser**.** Design data summaries are shown in the panel
40. Push **Quick Sizing Setup** button
41. Select the structure from the graphics or select PSHELL 3 from the property list

    All of the elements in the group of the selected property are highlighted.

42. Push **Next>**

    Now you will be asked to enter the initial value, lower and upper bounds of design variables. All three values are determined by the group property value multiplied by Mx (multiplier) plus Cx (constant). For example, if you are designing shell thickness which is defined as 1.0 in the shell group property, initial thickness T0, lower and upper bounds TL and TU are calculated by T0=1.0*M1+C1, TL=1.0*M2+C2, and TU=1.0*M3+C3 respectively. MinLB and MaxUB are absolute minimum and maximum regardless of the current group property values.

43. Enter `0.0` in C1 field and `1.0` in M1 field

    This will make the initial value to be the same, as specified by the analysis data (1.0 in this case).

44. Enter `0.001` in C2 field and `0.0` in M2 field

    This will make the lower bound value to be 10% of the analysis initial thickness.

45. Enter `2.0` in C3 field and `0.0` in M3 field

    This will make the upper bound value to be 2 times the initial thickness.
    Make sure that MinLB is set equal to or smaller than the lowest lower bound value and MaxUB is set equal to or bigger than the biggest upper bound value.

46. Push the **Finish** button

Select the Topometry Optimization Region

47. From the Design category chooser, select **Topometry**
48. Select PSHELL 3
49. Push the **Modify Topometry Design** button from the Edit menu toolbar
50. Push the **Finish** button

Defining the Design Objective

51. From the **Design** category chooser, select **Objectives**
52. Push **New Objective** button from the Edit menu toolbar
53. Accept the default, **Strain Energy** response type. Accept the **Min** Objective Definition switch
54. Push **Next>**
55. Select the existing loadcase
56. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Defining the Design Constraints

57. From the category chooser, select **Constraints**
58. Push **New Constraint** button from the Edit menu toolbar
59. Enter Name `Whole_Structure`
60. Select **Volume** response

61. Enter `1600.0` for the **Upper Bound**

    This constraint will cause Genesis to try to use (66% of the initial material) 1600 out of 2400 of the material.

62. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Optimize the Structure Using Genesis

63. From the main menu bar, select **Genesis** → **Optimize**

    As with the Single Analysis run, a Genesis console window will appear.
    Additionally, two design history charts will display the objective and the maximum constraint violation versus each design cycle. These charts will update, as the Genesis run progresses.

64. Study the **Design History** charts; when done, push the **Close** button

65. Study the **Genesis Console Output** window

Import the Post-Processing Files (Thickness Results)

66. From the **Genesis Console Output** window, select the **Import Post..** button

67. Select all the files using the **Shift** key

    `TMDSG002_dsgOPOST00.pch` file is the file containing the thickness results
    `TMDSG002_dsg00.pch` file is the file containing the finite element analysis results

68. Push the **Import** button

69. From the **Genesis Console Output** window, select the **Close** button to close the window

Post-Processing the Results (Thickness Results)

70. Select the **Post** tab

71. Push the **Deform Mesh/Color Mesh** button

72. Select a Thickness Result for any design cycle

    Study the results.

73. Select a Thickness Result for the last design cycle

    Study the results.

74. Push the **Options...** button. Slide the **Lower Cutoff** slidebar to mask out element with low density values

75. Push the **Close** button

Post-Processing the Results (Thickness & Displacement Results)

76. Push the **Oscillate** button
77. Select a Displacement Result from the **Deform Mesh** list for the last design cycle

    Study the results.
78. Select **Thickness** Result for a different design cycle

    Note that as you change design cycles for thickness, the Displacement results change, as well.
79. Push the **Filled Contours** radio button
80. Select a Displacement Result from the **Color Mesh** list for any design cycle
81. Push the **Options...** button. Slide the **Lower Cutoff** slidebar to mask out element with low displacement values
82. Push the **Close** button

Finding the Top 10 Displacements:

83. Compare the maximum displacement values of the first and the last design cycle

    Hint: Right-click the viewport and select Top 10 to see the 10 highest displacements.
84. Push the **Up** button

Post-Processing the Results (Animate results)

85. Select the **Post** tab
86. Push the **Animation** button
87. Change the **Color Result Type** chooser to **Element Sizing** result
88. Push **Next>**
89. Select all the results you want to animate
90. Push **Next>**
91. Push the **Options...** button; slide the **Lower Cutoff** slide bar to mask out element with low density values
92. Push the **Close** button
93. Select the **Shaded Feature** icon in the panel below
94. Select the **Flat Shaded** icon in the panel below
95. Push the **Up** button

Quit Design Studio

96. From the main menu bar, select **File** → **Quit**
97. Push the **Don’t Save** button

5.3 Topometry Optimization of a Pallet Lifter Using Symmetries

Introduction

The purpose of this example is to become familiar with the creation of topometry optimization data using multiple designable regions and symmetries. You will use a file that already has sizing optimization data.

The following optimization problem will be completed, solved and post-processed:

Minimize Mass

Subject to:

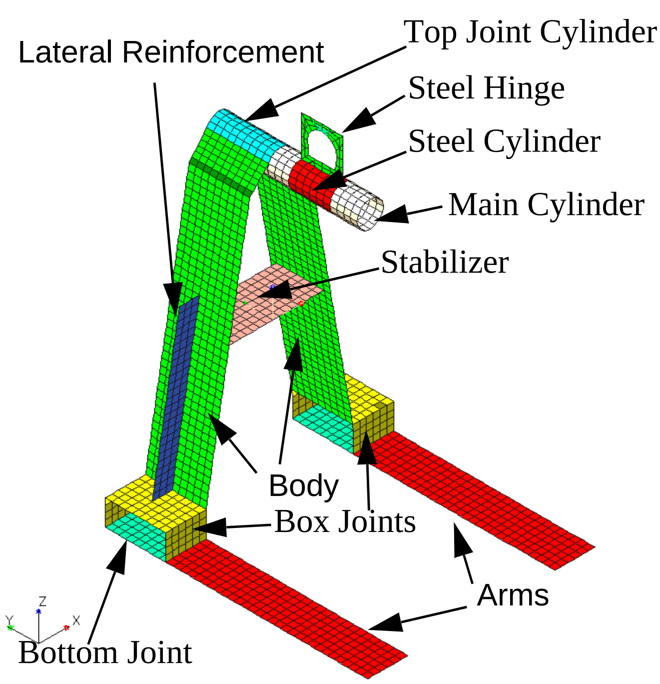
Stress $\leq$ 20,000 psi (all elements)

Magnitude of tip displacements $\leq$ 0.1 (4 tip grids: 13579, 13765, 14055 & 14235)

Designable region:

All aluminum elements independently

The following picture shows the pallet lifter structure and its components.



Example ID

TMDSG003

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
TMDSG003.dat	Input data	Provided: Contains the finite element mesh of a pallet lifter with loading and sizing optimization data.
TMDSG003_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in TMDSG003.dat.
TMDSG003_dsgOPOSTxy.pch	punch file	Multiple files generated using Genesis within Design Studio. These files contain the thicknesses for all shell elements for post-processing all design cycles.
TMDSG003_dsgSSOLxx.dat	Input data	Generated using Genesis within Design Studio. This file contains solid elements that represent the shell elements. This file is for visualization purpose and not intended for analysis. This file is for the last design cycle xx.
TMDSG003_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to TMDSG003_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: TMDSG003.dat

Check the Design Variables

3. Select the **Design** tab
4. From the **Design** category chooser, select **Design Variables**

Verify that there are 8 design variables which use the following values:

Initial Value	Lower Bound	Upper Bound
0.5	0.01	2.0
0.375	0.01	2.0
0.1875	0.01	2.0
0.1875	0.01	2.0

0.375	0.01	2.0
0.6875	0.01	2.0
0.5625	0.01	2.0
0.1875	0.01	2.0

Change the Lower and Upper Bound of all Design Variables

5. Select all the design variables
6. Push the **Modify Design Variable** button from the Edit menu toolbar
7. Push **Next>**
8. Put a checkmark in the **Lower bound** checkbox
9. For **Lower Bound** enter `0.1875`
10. Put a checkmark in the **Upper bound** checkbox
11. For **Upper Bound** enter `0.75`
12. Push the **Finish>** button
13. Verify that the 8 design variables now use the following values:

Initial Value	Lower Bound	Upper Bound
0.5	0.1875	0.75
0.375	0.1875	0.75
0.1875	0.1875	0.75
0.1875	0.1875	0.75
0.375	0.1875	0.75
0.6875	0.1875	0.75
0.5625	0.1875	0.75
0.1875	0.1875	0.75

Verify the Sizing Regions

14. From the **Design** category chooser, select **Sizing**
15. Verify that the PSHELLs with ids are 3, 4, 5, 6, 23, 28, 33, and 36 are being designed

Select the Topometry Optimization Regions and their Symmetry Conditions

16. From the **Design** category chooser, select **Topometry**
17. Select PSHELL 3, 4, 5, 6, 23, 28, 33 and 36

    Notice that this list contains only the groups (PSHELLs) that are being size designed. For example, PSHELLs 37 and 43 are not in this list.

18. Push the **Modify Topometry Design** button from the Edit menu toolbar
19. Set up Symmetry Conditions:

    You will now set up symmetry conditions in all topometry regions. The condition is mirror symmetry with respect to the YZ plane of the local coordinate system 1:

20. Push the **Change** button
21. Select the **Rect ID=1** coordinate system
22. Push **Next>**
23. For **Symmetry 1:**, Select **MYZ: Mirror about YZ plane**
24. Push the **Advanced...** button
25. For **Symmetry Tolerance:** Type `2.0`

    Normally tolerances do not need to be changed. This is not a typical case. In practice, when you use symmetries, it is recommended to compare the number of design variables versus the number of designable elements to see if the default tolerance works; if not, the default value like in this problem can be changed. 2.0 represents 2.0% of the symmetric distance.

26. Push **Next>**
27. Push the **Finish** button

Note:

The following information will be written in the Genesis input data

DSPLIT	3								
+	SYM	1	MYZ				2.0		
DSPLIT	4								
+	SYM	1	MYZ				2.0		
DSPLIT	5								
+	SYM	1	MYZ				2.0		
DSPLIT	6								
+	SYM	1	MYZ				2.0		
DSPLIT	23								
+	SYM	1	MYZ				2.0		
DSPLIT	26								
+	SYM	1	MYZ				2.0		

DSPLIT	33								
+	SYM	1	MYZ				2.0		
DSPLIT	36								
+	SYM	1	MYZ				2.0		

Request the Solid File

28. From the main menu bar, select **Genesis** → **Options**
29. Select the **File Control** tab
30. Make sure that the **Shell-to-Solid File** is selected and the **Create (Fixed Norms)** is selected

Increase the Maximum Number of Design Cycles

31. Select the **Design Control** tab
32. For **Maximum Number of Design Cycles**, enter `20`

    Here you increase the default maximum number of design cycles from 10 to 20.

33. Push the **Apply** button

Optimize the Structure Using Genesis

34. From the main menu bar, select **Genesis** → **Optimize**
35. Study the **Design History** charts; when done, push the **Close** button
36. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Design Post-Processing Files

37. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
38. Select the `TMDSG003_dsgOPOSTS00.pch` file and put a checkmark in the **Import Similar Results for All Design Cycles** checkbox

    Putting a checkmark in the checkbox will cause Design Studio to load many result files (one for each design cycle) in one step.

39. Push the **Open** button

Post-Processing the Design Results (Thicknesses on Shells)

40. Select the **Post** tab
41. Push the **Deform/Mesh Color Mesh** button

42. Select a Thickness Result for any design cycle
43. Select the Thickness Result for the last design cycle
44. Push the **Smooth Shaded** icon from the panel below
45. Push the **Shaded Feature** icon from the panel below
46. Push the **Flat Shaded** icon from the panel below
47. Push the **Close** button
48. Push the **Up** button

Import the Solid File

49. From the main menu bar, select **File** → **New**
50. Using **File** → **Import** → **Input Data..,** import the Genesis data file: `TMDSG003_dsgSSOLxx.dat`

Re-Import the Post-Processing Files

51. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
52. Select the `TMDSG003_dsgOPOSTSxx.pch` file (where xx is the last design cycle number)

    If you can not see this file and/or to have all file listed: Select **All Files** in the drop-down menu **Files of Type.**
53. Push the **Open** button

Post-Processing the Results (Thicknesses on Solids)

54. Select the **Post** tab
55. Push the **Deform/Mesh Color Mesh** button
56. Select the Thickness Result for the last design cycle

Creating a Picture File

Select a good view of the results.

57. From the main menu bar, select **File** → **Print to Image File**

    Check your working directory. A picture named `TMDSG003.png` should be there.
58. Push the **Save** button

Study the Design Space

59. From the main menu bar, select **File** → **New**
60. Import the Genesis data file: `TMDSG003_dsgUPDATE00.dat`
61. Select the **Display** tab
62. Change the **Group Display Style** from **Wire Frame** to **Flat Shaded**

    If you put the cursor, without pressing it, over the Group Display Style buttons a tool tip will reveal the names/function of the buttons.

63. Push the **Show/Hide Groups** button
64. Hide some groups from one side, confirm that the symmetric groups are also hidden, if some elements are not hidden in symmetric pairs, you need to verify our tolerance for symmetries or verify the model itself (it might not be symmetric).

Quit Design Studio

65. From the main menu bar, select **File** → **Quit**
66. Push the **Don’t Save** button

Study the Output File

67. In a text editor load the Genesis data file: `TMDSG003_dsg.out`
68. Study briefly the file
69. Using the output file, complete the following table:

	Sizing Reference Answer (1)	**Topometry Reference Answer (2)**	**Topometry (3)**
Number of Design Variables	8	1018	
Initial Mass	2.307E-2	2.307E-2	
Optimal Mass	1.768E-2	1.497E-2	

(1) Result from Sizing Optimization
(2) Result from your run, `TMDSG003_ref.out`
(3) Result from your run, `TMDSG003_dsg.out`

70. Compare the mass with the original structure

    For reference answer: Change is (1.497E-2-2.307E-2)/2.307E-2*100=-35
    Topometry reduced the mass by 35%.

5.4 Coarse Topometry Optimization of a Pallet Lifter

Introduction

The purpose of this example is to become familiar with using the coarse option available for topometry optimization. You will use a file that already has topometry optimization data.

The following optimization problem will be solved and post-processed:

Minimize Mass

Subject to:

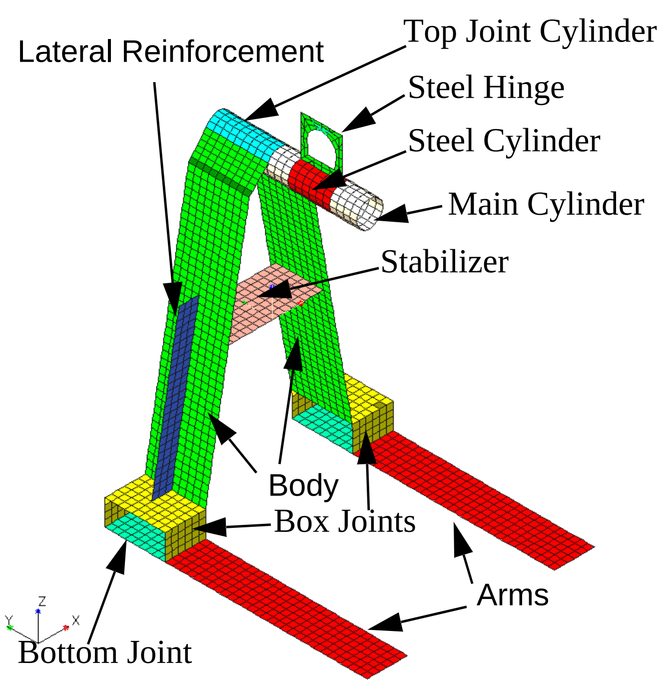
Stress $\leq$ 20,000 psi (all elements)

Magnitude of tip displacements $\leq$ 0.1 (4 tip grids: 13579, 13765, 14055 & 14235)

Designable region:

All aluminum elements in a group by group basis. The groups are created automatically in Genesis controlled by the coarse parameters.

The following picture shows the pallet lifter structure and its components.



Benefits of Coarse Topometry

The usage of coarse parameters allows to:

- Reduce the number of design variables. This allows to reduce computational time
- Obtain structures that are easier to manufacture, as less variability is allowed
- Reduce checkerboard problems
- Deal with minimum member size requirements

Example ID

TMDSG004

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
TMDSG004.dat	Input data	Provided: Contains the finite element mesh of a pallet lifter with loading and sizing optimization data.
TMDSG004_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in TMDSG004.dat.
TMDSG004_dsgOPOSTxy.pch	punch file	Multiple files generated using Genesis within Design Studio. These files contain the thicknesses for all shell elements for post-processing all design cycles.
TMDSG004_dsgSSOLxx.dat	Input data	Generated using Genesis within Design Studio. This file contains solid elements that represent the shell elements. This file is for visualization purpose and not intended for analysis. This file is for the last design cycle xx.
TMDSG004_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to TMDSG004_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: TMDSG004.dat

Verify the Design Variables

3. Select the **Design** tab

4. From the category chooser, select **Design Variables**
5. Verify that there are 8 design variables and that they use the following values:

Initial Value	Lower Bound	Upper Bound
0.5	0.1875	0.75
0.375	0.1875	0.75
0.1875	0.1875	0.75
0.1875	0.1875	0.75
0.375	0.1875	0.75
0.6875	0.1875	0.75
0.5625	0.1875	0.75
0.1875	0.1875	0.75

Verify the Sizing Regions

6. From the category chooser, select **Sizing**
7. Verify that there are 8 designable PSHELLs and that their corresponding ids are 3, 4, 5, 6, 23, 28, 33 and 36.

Verify The Topometry Optimization Data

8. From the **Design** category chooser, select **Topometry**
9. Verify that there are 8 designable PSHELLs and that their corresponding ids are 3, 4, 5, 6, 23, 28, 33, and 36.

Set the Coarse Conditions

The following coarse conditions will be set:

PSHELL ID	Max elements per design variable
3	36
4	81
5	9
23	9

10. Select PSHELL 3, `Arms`

11. Push the **Modify Topometry Design** button from the Edit menu toolbar
12. For **Coarse Method:** select **Max elements per design variable**
13. For **Coarse Parameter**, enter `36`
14. Push the **Finish** button
15. Select PSHELL 4, `Body`
16. Push the **Modify Topometry Design** button from the Edit menu toolbar
17. For **Coarse Method:** select **Max elements per design variable**
18. For **Coarse Parameter**, enter `81`
19. Push the **Finish** button
20. Select PSHELL 5 and 23, `Lateral Reinforcements & Stabilizer`
21. Push the **Modify Topometry Design** button from the Edit menu toolbar
22. Put a checkmark in the **Coarse Method** checkbox
23. For **Coarse Method:** select **Max elements per design variable**
24. For **Coarse Parameter**, enter `9`
25. Push the **Finish** button

Note:

The following information will be written in the Genesis input data

DSPLIT	3								
+	**COARSE**	**MAXELEM**	**36**						
+	SYM	1	MYZ				2.0		
DSPLIT	4								
+	**COARSE**	**MAXELEM**	**81**						
+	SYM	1	MYZ				2.0		
DSPLIT	5								
+	**COARSE**	**MAXELEM**	**9**						
+	SYM	1	MYZ				2.0		
DSPLIT	23								
+	**COARSE**	**MAXELEM**	**9**						
+	SYM	1	MYZ				2.0		

Optimize the Structure Using Genesis

26. From the main menu bar, select **Genesis** → **Optimize**

27. Study the **Design History** charts; when done, push the **Close** button
28. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

29. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
30. Select the `TMDSG004_dsgOPOST00.pch` file and put a checkmark in the **Import Similar Results for All Design Cycles** checkbox

    Putting a checkmark in the checkbox will cause Design Studio to load many result files (one for each design cycle) in one step.

31. Push the **Open** button

Post-Processing the Results (Thicknesses on Shells)

32. Select the **Post** tab
33. Push the **Deform/Mesh Color Mesh** button
34. Select a Thickness Result for any design cycle
35. Select a Thickness Result for the last design cycle
36. Change **Value** to **Value Change**
37. Change **Value Change** to **Value/Original**
38. Push the **Smooth Shaded** icon from the panel below
39. Push the **Shaded Feature** icon from the panel below

Import the Solid File

40. From the main menu bar, select **File** → **New**
41. Import the Genesis data file: `TMDSG004_dsgSSOLxx.dat`

Re-Import the Post-Processing Files

42. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
43. Select the `TMDSG004_dsgOPOSTSxx.pch` file (`xx` corresponds to the last design cycle)

    If you can not see this file and/or to have all file listed: Select **All Files** in the drop-down menu **Files of Type.**

44. Push the **Open** button

Post-Processing the Results (Thicknesses on Solids)

45. Select the **Post** tab
46. Push the **Deform/Mesh Color Mesh** button
47. Select the Thickness Result for the last design cycle

Creating a Picture File

Select a good view of the results.

48. From the main menu bar, select **File** → **Print to Image File**
49. Enter name `TMDSG004.png`
50. Push the **Save** button

Verify your working directory. A picture named `TMDSG004_dsgSSOLxx.png` should be there.

Study the Design Space

51. From the main menu bar, select **File** → **New**
52. Import the Genesis data file: `TMDSG004_dsgUPDATE00.dat`
53. Select the **Display** tab
54. Change the **Group Display Style** from **Wire Frame** to **Flat Shaded**

If you put the cursor, without pressing it, over the Group Display Style buttons a tool tip will reveal the names/function of the buttons.

55. Push the **Show/Hide Groups** button
56. Hide some groups from one side and confirm that the symmetric elements are also hidden. If some elements are not hidden in symmetric pairs, you need to check the tolerance for symmetries or check the model itself (it might not be symmetric).

Quit Design Studio

57. From the main menu bar, select **File** → **Quit**
58. Push the **Don't Save** button

Study the Output File

59. In the text editor load the Genesis data file: `TMDSG004_dsg.out`
60. Study briefly the file

61. Using the output file, complete the following table:

	Sizing Reference Answer (1)	Topometry Reference Answer (2)	Coarse Topometry Reference Answer (3)	Coarse Topometry (4)
Number of Design Variables	8	1018	321	
Initial Mass	2.307E-2	2.307E-2	2.307E-2	
Optimal Mass	1.768E-2	1.497E-2	1.611E-2	

(1) Result from Sizing Optimization
(2) Result from your run without coarsening
(3) Result from your run, `TMDSG004_ref.out`
(4) Result from your run, `TMDSG004_dsg.out`

62. Compare the optimal mass with the initial mass

For reference answer: Change is (1.611E-2-2.307E-2)/2.307E-2*100=-30%
Coarse topometry reduced the mass by 30%.

5.5 Creating a CAD Mesh that Represents Results of a Topometry Optimized Shell Structure

Introduction

The purpose of this example is to learn how to create and to export a file with a CAD geometry that represents the results of a topometry optimized shell structure. The optimization data will be provided. Here you will focus mostly on how to create CAD (IGES) files. You will create two IGES files, one will be coarse and another will be refined.

Objective function of the problem:

Minimize the strain energy of the structure

Subject to:

Volume $\leq$ 700.0

Designable region:

Every element in the structure

Analysis problem:

Static Analysis
Torsion Load produced by two Forces of opposite direction
The four corners of the structure are fixed

Example ID

TMDSG005

Files Used in This Problem

A list of the files, that are either provided to you or the ones you are expected to create during this example, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the example. Later, this list can be used for verifying your results.

File Name	Description
TMDSG005.dat	Provided: Contains a finite element mesh with optimization data.

TMDSG005TSURF1.igs	Generated with Design Studio. This is an IGES file which contains a coarse surface mesh that represents topometry results. This file is created to be used with a third party CAD program.
TMDSG005TSURF2.igs	Generated with Design Studio. This is an IGES file which contains a refined surface mesh that represents topometry results. This file is created to be used with a third party CAD program.
TMDSG005TSURF3.dat	Generated with Design Studio. This is a Genesis file which contains a coarse surface mesh that represents topometry results. This file is not intended for analysis, it is only for showing topometry results in Design Studio.
TMDSG005TSURF4.dat	Generated with Design Studio. This is a Genesis file which contains a refined surface mesh that represents topometry results.This file is not intended for analysis, it is only for showing topometry results in Design Studio.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG005.dat`

Optimize the Structure Using Genesis

3. From the main menu bar, select **Genesis** → **Optimize**

   Wait until Genesis finishes the run.

Import the Post-Processing Files (Thickness Results)

4. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
5. Select the `TMDSG005_dsgOPOSTxx.pch` (where xx represents the last design cycle) file
6. Push the **Open** button

Post-Processing the Results (Thickness Results)

7. Select the **Post** tab
8. Push the **Deform Mesh/Color Mesh** button
9. Select the Thickness Result (for the last design cycle)
10. Push the **Options...** button. Slide the **Lower Cutoff** slidebar to mask out element with low thickness values

    Note: the results obtained can be different depending of which option is selected: **Filled Elements** or **Filled Contour**. For this example use the default option.
11. Push the **Close** button

Export a Coarse Surface Representation of the Topometry Result, Use IGES Data Format

12. From the main menu bar, select **File** → **Export** → **Coarsened Surface...**
13. For **Surface File Format**, select **IGES**
14. Enter `TMDSG005TSURF1.igs`
15. Push the **Save** button

Export a Refined Surface Representation of the Topometry Result, Use IGES Format

16. From the main menu bar, select **File** → **Export** → **Coarsened Surface...**
17. For **Surface File Format**, select **IGES**
18. Move the **Surface Mesh** slider towards the **Fine** setting
19. Enter `TMDSG005TSURF2.igs`
20. Push the **Save** button

Export a Coarse Surface Representation of the Topometry Result, Use Input Data Format

Because Design Studio can not read IGES files, you will create the same mesh as above but using the Genesis input data format.

21. From the main menu bar, select **File** → **Export** → **Coarsened Surface...**
22. Enter `TMDSG005TSURF3.dat`
23. Push the **Save** button

Export a Refined Surface Representation of the Topometry Result, Use Input Data Format

24. From the main menu bar, select **File** → **Export** → **Coarsened Surface...**
25. Move the **Surface Mesh** slider towards the **Fine** setting
26. Enter `TMDSG005TSURF4.dat`
27. Push the **Save** button
28. Push the **Up** button

Import the Coarse Surface Representation of the Topometry Results

Here you will study the coarse file which you have generated above.

29. Import the Genesis data file: `TMDSG005TSURF3.dat`
30. Select the **Display** tab
31. Push the **Show/Hide Groups** button
32. Hide PSHELL 4
33. Study the Viewport
34. Using the **Group Display Style** chooser, select **Flat Shaded**

Import the Refined Surface Representation of the Topometry Results

Here you will study the refined file which you have generated above.

35. Import the Genesis data file: `TMDSG005TSURF4.dat`

Compare the Coarse and the Refined Surface Representation of the Topometry Results

36. Hide PSHELL 8

    Compare the refined with the coarse mesh by displaying and hiding PSHELL 8 and PSHELL 12.

37. Which surface is better?

    From the quality of the mesh point of view, the refined is clearly better. However, many CAD programs are slow to read refined meshes and therefore; sometimes, the coarse representation is adequate. In this case you studied two levels, Design Studio allows to select higher or lower coarse levels. You might select other coarse levels to find the one that is accurate enough and simultaneously small enough.

Counting Number of Elements in the Coarse Mesh

38. Push the **Hide All** button
39. Select PSHELL 8
40. Select the **Analysis** tab
41. From the category chooser, select **Elements**
42. Push the **Select All** button

43. Read the number of elements printed near the bottom of the Form

    Refined answer: 330. This number can vary depending on how many elements you masked out earlier.

44. Push the **Select None** button

Counting Number of Elements in the Fine Mesh

45. Select the **Display** tab
46. Push the **Hide All** button
47. Select the PSHELL 12
48. Select the **Analysis** tab
49. From the category chooser, select **Elements**
50. Push the **Select All** button
51. Read the number of elements printed near the bottom of the Form

    Refined answer: 1008. Again, this number can vary depending on how many elements you masked out earlier.

52. Push the **Select None** button

Quit Design Studio

53. From the main menu bar, select **File** → **Quit**
54. Push the **Don’t Save** button

5.6 Creating a Finite Element Mesh that Represents Results of a Topometry Optimized Structure using Re-grouping

Introduction

Topometry optimization results can give good ideas on where to reinforce structures or where to take material out of them. Sometimes the topometry results require additional steps to minimize the number of components which have different dimensions, in order to manufacture them easier or just to make them simpler. In this example, you will learn how to “re-group” elements to achieve that.

The purpose of this example is two-fold: one is to learn how to re-group elements and the other is how to create a finite element mesh that represents the simplified topometry results. The optimization data will be provided. You will first study the model and the provided optimization data. Then you will focus on how to create the finite element file that represents the topometry optimization results. You will also learn how to re-group elements so the final structure has components with uniform thickness. Finally, you will size optimize the structure to get the final answer.

Objective function of the problem:

Minimize the strain energy of the structure

Subject to:

Volume $\leq$ 600.0

Designable region:

Every element in the structure

Analysis problem:

Static Analysis

Two Point Loads

Two corners of the structure are fixed

Example ID

`TMDSG006`

Files Used in This Problem

A list, of the files that are either provided to you or the ones you are expected to create during this example, is presented next. It is not necessary to study the list in detail at this point. The files listed will be introduced during the example. Later, this list can be used for verifying your results.

Part	File Name	Type	Description
1	TMDSG006_1.dat	Input data	Provided: Contains analysis and topometry optimization data.
1	TMDSG006_1_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file should be identical to TMDSG006_1.dat.
1	TMDSG006_1_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
1	TMDSG006_1_dsgOPOSTxy.pch	punch file	Multiple files generated using Genesis within Design Studio. These files contain the thicknesses for all shell elements for post-processing all design cycles.
1	TMDSG006_2.dat	Input data	Created in this example. This file is created by exporting a modified version of the TMDSG007_1.dat file.The modification consists of deleting the less important elements and moving the remaining elements to three groups.
1	TMDSG006_2_ref.dat	Input data	Provided. Result input file. Should be similar to the file TMDSG006_2.dat. This file is provided in order to optionally verify your example.
2	TMDSG006_3.dat	Input data	Created in this example. This file is created by adding sizing data to the TMDSG006_2.dat file.
2	TMDSG006_3_dsgUPDATExx.dat	Input data	Generated by Genesis. This file is created by running the TMDSG006_3.dat file.
2	TMDSG006_4.dat	Input data	This file is created by combining the bulk data of the TMDSG006_3_dsgUPDATExx.dat file and the executive control and solution control of the TMDSG006_2.dat file. This file represent the optimal answer.
2	TMDSG006_4_ref.dat	Input data	Provided. Result input file. Should be similar to the file TMDSG006_4.dat. This file is provided to optionally check your example.

5.6.1 Part 1

The purpose of this part of the example is to create a Genesis input file that contains a finite element mesh with three main components. When you finish this example, you should have created a file named: `TMDSG006_2.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG006_1.dat`

Study the Analysis Problem

3. From the main menu bar, select **Genesis** → **Model Summary**

   Verify that the model has the following characteristics.
   The mesh is 18x80.
   Number of grids: 1539
   Number of CQUAD4 elements: 1440

4. Push the **Close** button
5. Select the **Analysis** tab
6. From the category chooser, select **Loadcases**
7. Select the **existing loadcase**

   Verify that there are two point loads, and that the structure is fixed at two corners.

Clear the Selection

8. Right-click the Viewport, select **Clear**→ **All**

Study the Design Problem

9. Select the **Design** tab
10. From the category chooser, select **Objectives**

    Verify that the objective is to Minimize the Strain Energy (SENERGY).

11. From the category chooser, select **Constraints**

    Verify that there is a volume (VOLUME) constraint and its upper bound is 600.00.

12. From the category chooser, select **Design Variables**

    Verify that there is one design variable with the following characteristics:

DESIGN VARIABLE Type	LABEL	Initial Value	Lower Bound	Upper Bound
Independent	T4	0.6	0.1	1.2

13. From the category chooser, select **Sizing**

    Verify that PSHELL 4 is designed. A hammer icon should be next to PSHELL.

14. Select PSHELL 4 from the property list

    All the elements in the group are highlighted.

15. Push the **Modify Sizing Design** button from the Edit menu toolbar

    Verify that thickness of PSHELL 4 is being designed by design variable 1, T4.

16. Push the **Cancel** button

    You can also verify that PSHELL 4 is being designed by design variable 1, T4, by reading the bottom of the Design Studio main window.

17. From the category chooser, select **Topometry**

    Verify that PSHELL 4 is designed. A hammer icon should be next to PSHELL.

18. From the category chooser, select **Quick Setup Trails**

    Review the list in the panel.

Clear the Selection

19. Right-click the Viewport, select **Clear**→ **All**

Optimize the Structure Using Genesis

20. From the main menu bar, select **Genesis** → **Optimize**

    Wait until Genesis finishes the run.

Briefly Study the Results

21. In a text editor, load the file: `TMDSG006_1_dsg.out`

22. Complete the following table:

Result	Topometry Reference Solution (1)	Topometry Solution (2)
Strain Energy (Initial Design)	265.3	
Strain Energy (Final Design)	223.8	
Volume (Initial Design)	864.0	
Volume (Final Design)	598.2	

(1) Result from `TMDSG006_1.out`
(2) Result from your run, `TMDSG006_1_dsg.out`

Verify that the objective function has been reduced by about 15%.
Verify that the volume that was originally 45% higher than the upper constraint bound, now is satisfied (volume is less than 600.0).

Import the Post-Processing Files (Thickness Results)

23. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
24. Select the `TMDSG006_1_dsgOPOSTxx.pch` (where xx represents the last design cycle) file
25. Push the **Open** button

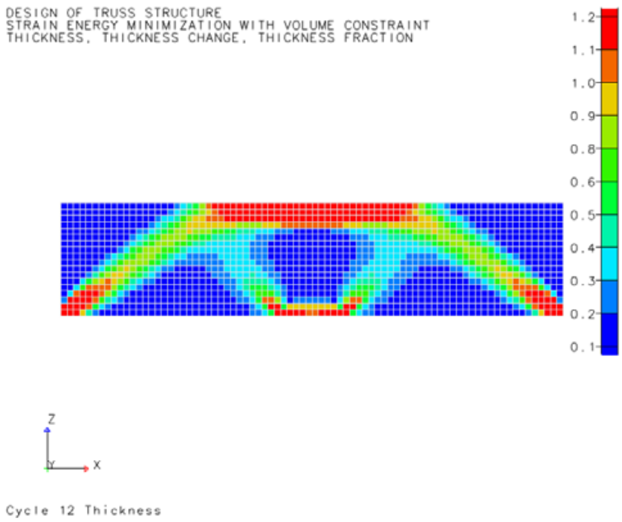
Change View

26. Push the **Left view** button to change view to the X-Z Plane

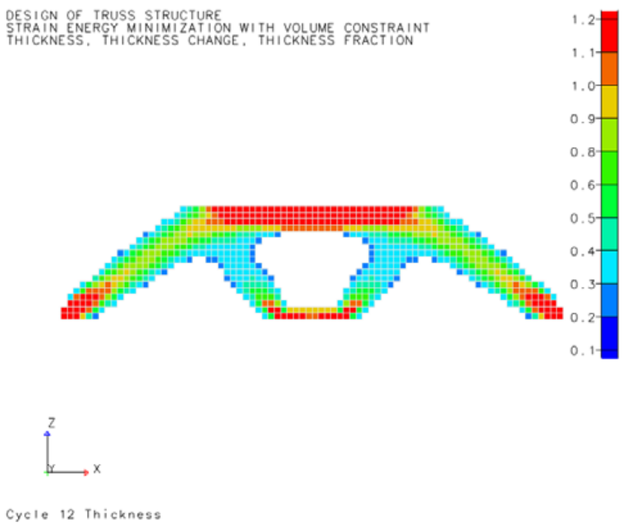
Post-Processing the Results (Thickness Results)

27. Select the **Post** tab
28. Push the **Deform Mesh/Color Mesh** button

29. Select the **Thickness** Result (for the last design cycle)



30. Push the **Options...** button. Slide the **Lower Cutoff** slidebar to mask out elements with low thickness values (0.3 or less)
31. Push the **Close** button



Create Three New Groups

32. Select the **Display** tab
33. Push the **Manage Groups** button
34. Push **New Group** button from the Edit menu toolbar
35. Enter Name: `upper_member`
36. For Type, select **PSHELL**

37. Select Blue Color
38. Push the **Finish** button
39. Push **New Group** button from the Edit menu toolbar
40. Enter Name: `lower_member`
41. For Type, select **PSHELL**
42. Select Yellow Color
43. Push the **Finish** button
44. Push **New Group** button from the Edit menu toolbar
45. Enter Name: `diagonals`
46. For Type, select **PSHELL**
47. Select Green Color
48. Push the **Finish** button

Finding Appropriate Values for Thickness of Groups

Here you will select some thickness values to represent the horizontal and diagonal members. The thickness values are only estimates.

49. Select the **Post** tab
50. Select one element from the top horizontal part of the structure

    In the **Design Studio Messages** windows you will be able to see the value selected. Reference pick: 1.2 (rounded)

51. Select one element from the bottom horizontal part of the structure

    Reference pick: 1.1 (rounded)

52. Select one element to represent the diagonal parts of the structure

    Reference pick: 0.8 (rounded)

Assigning Thickness Values to the Groups

53. Select the **Analysis** tab
54. From the category chooser, select **Group Properties**

    You should now see 4 PSHELL's listed.

55. Select **PSHELL 5,** `upper_member`
56. Push the **Modify Group Property** button from the Edit menu toolbar
57. Enter `1.2` for thickness

58. Push the **Finish** button
59. Select the **PSHELL 6,** `lower_member`
60. Push the **Modify Group Property** button from the Edit menu toolbar
61. Enter `1.1` for thickness
62. Push the **Finish** button
63. Select the **PSHELL 7,** `diagonals`
64. Push the **Modify Group Property** button from the Edit menu toolbar
65. Accept `0.8` for thickness
66. Push the **Finish** button

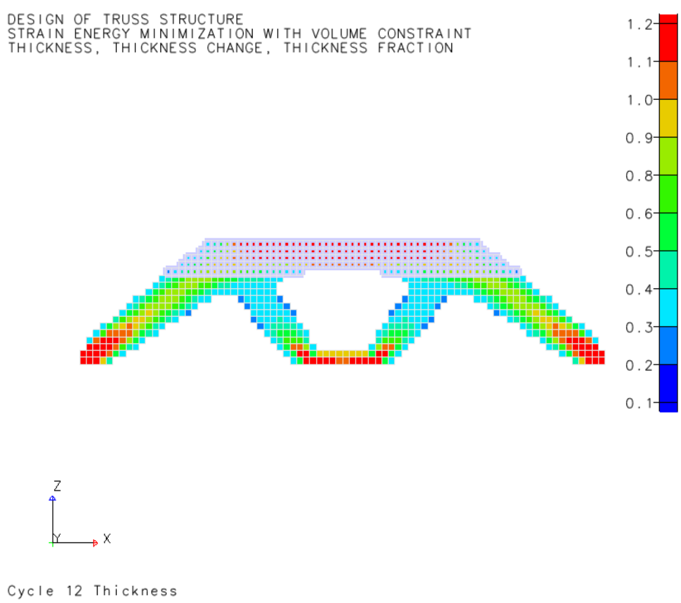
Moving the Elements to the First Group

67. From the **Analysis** category chooser, select **Elements**
68. Make sure that there are no selected elements. For that, check the bottom of the Design Studio main window. If there are elements selected; push the **Select None** button

    Most likely there was one element selected. As earlier you selected elements to find out the value of the thickness for the group.

69. Select the elements on the top part of the structure

    As shown in the next figure, select the elements on the first 5 rows of the structure. After you select them, the **Elements** Panel should indicate that about 226 elements have been selected. The 226 number might vary in your case, as this number depends on how many elements you previously masked out.

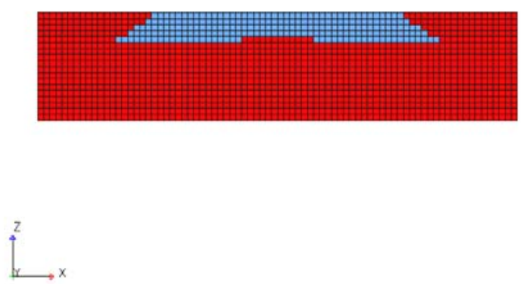


70. Push the **Modify Elements** button from the Edit menu toolbar
71. Accept the default option, **Change Element's Group**
72. Push **Next>**
73. Select **PSHELL 5,** `upper_member`
74. Push the **Finish** button

Verify the Moving of the Elements to the First Group and Hide them

75. Select the **Post** tab
76. Push the **Up** button

    The elements in the fist group should be blue.

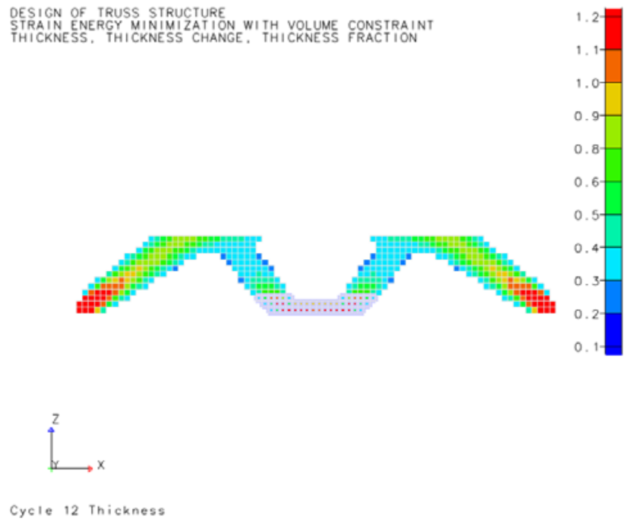


77. Select the **Display** tab
78. Push the **Up** button
79. Push the **Show/Hide Groups** button
80. Hide the `upper_member` group

Moving the Elements to the Second Group

81. Select the **Post** tab
82. Push the **Deform Mesh/Color Mesh** button
83. Select the Thickness Result (for the last design cycle)
84. Select the **Analysis** tab
85. From the category chooser, select **Elements**

86. Select the elements on the middle bottom part of the structure



Select the elements on the last 3 rows of the structure. After you select them, the Elements Panel should indicate that about 46 elements have been selected.

87. Push the **Modify Elements** button from the Edit menu toolbar
88. Accept the defaults option, **Change Element's Group**
89. Push **Next>**
90. Select **PSHELL 6,** `lower_member`
91. Push the **Finish** button

Verify the Moving of the Elements to the Second Group and Hide them

92. Select the **Post** tab
93. Push the **Up** button

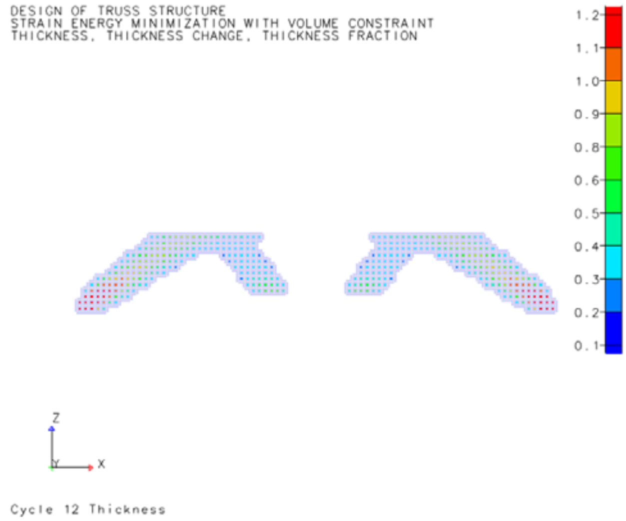
The elements in the second group should be yellow.

94. Select the **Display** tab
95. Hide the `lower_member` group

Moving the Elements to the Third Group

96. Select the **Post** tab
97. Push the **Deform Mesh/Color Mesh** button
98. Select the Thickness Result (for the last design cycle)

99. Select the **Analysis** tab
100. From the category chooser, select **Elements**
101. Push the **Select All** button to select the remaining elements of the structure



After you select them, the Elements Panel should indicate that about 414 elements have been selected.

102. Push the **Modify Elements** button from the Edit menu toolbar
103. Accept the defaults option, **Change Element's Group**
104. Push **Next>**
105. Select **PSHELL 7,** `diagonals`
106. Push the **Finish** button

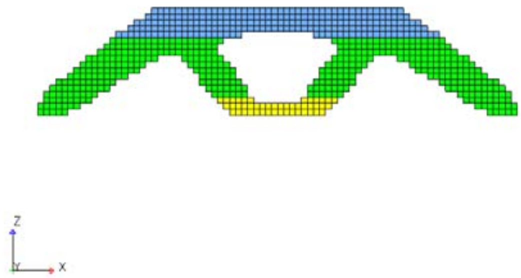
Verify the Moving of the Elements to the Third Group and Hide them

107. Select the **Post** tab
108. Push the **Up** button

The elements in the third group should be in green.

109. Select the **Display** tab
110. Push the **Show All** button

111. Select the **PSHELL 4** to hide it



Now you should see the final structure.

The next task will be to delete the thin elements. But before doing that, you will delete the existing topometry, sizing and design variables.

Delete the Existing Topometry Data

112. Select the **Design** tab
113. From the category chooser, select **Topometry**
114. Select **PSHELL 4**
115. Push the **Delete Topometry Design**

Delete the Existing Sizing Data

116. From the category chooser, select **Sizing**
117. Select **PSHELL 4**
118. Push the **Delete Sizing Design**

Delete the Existing Design Variable

119. From the category chooser, select **Design Variables**
120. From the main menu bar, select **Edit** → **Select All**
121. From the Edit menu toolbar, select **Delete Design Variable** button

Delete the Thin Elements

122. Select the **Display** tab
123. Select the **Show/Hide Groups** button
124. Push the **Invert All** button

   Now you should see the thin elements in red; these elements are not very important, so you will delete them.

125. Select the **Analysis** tab
126. From the category chooser, select **Elements**
127. From the main menu bar, select **Edit** → **Select All**
128. From the Edit menu toolbar, select **Delete Elements** button
129. From the category chooser, select **Grids**
130. From the main menu bar, select **Edit** → **Select All**
131. From the Edit menu toolbar, select **Delete Grids** button

   Now the Design Studio Viewport should be with no elements or grids.

132. Select the **Display** tab
133. Push the **Show All** button

   Now you should see the structure assembled with 3 groups.

Delete the Constraints

134. Select the **Design** tab
135. From the category chooser, select **Constraints**
136. Select the existing constraint
137. From the Edit menu toolbar, select **Delete Constraint** button

Delete the Objective

138. From the category chooser, select **Objectives**
139. Select the existing objective
140. From the Edit menu toolbar, select **Delete Objective** button

Save the Design Studio Database File

141. From the main menu bar, select **File** → **Save As...**

142. Enter `TMDSG006_2` as the Filename

143. Push the **Save** button (as a Design Studio File)

Export the Input File

144. From the main menu bar, select **File** → **Export** → **Input Data...**

145. Enter `TMDSG006_2.dat`

146. Do not check the **Only Export Visible Groups/Elements** checkbox

147. Push the **Save** button

At this point, you have an answer that satisfies one of the original requirements: Design a structure with three different parts. This structure however, might or might not satisfy the volume constraint. To see if that is the case, you will analyze it.

Analyze the Structure

148. From the main menu bar, select **Genesis** → **Single Analysis**

After Genesis finishes the run, check the volume of the structure.

Quit Design Studio

149. From the main menu bar, select **File** → **Quit**

150. Push the **Don't Save** button

Study the Output File

151. In a text editor, load the file: `TMDSG006_2_dsg.out`

152. Complete the following table:

Result	Analysis Reference Solution (1)	Analysis Solution (2)
Strain Energy	237.8	
VOLUME	655.4	

(1) Result from `TMDSG006_2_ref.out`
(2) Result from your run, `TMDSG006_2_dsg.out`

153. Is the volume obtained acceptable?

If the volume constraint is satisfied, you could stop here and accept this as the final answer. Otherwise, you could reduce the thickness manually or better yet, re-optimize the structure by performing sizing optimization to keep using only three different thicknesses.
In the next part, you will re-optimize the structure using sizing optimization.

5.6.2 Part 2

The purpose of this part of the example is to size optimize the finite element mesh created in Part 1 of this example.

When you finish this example, you should have created a file named: `TMDSG006_4.dat`.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG006_2.dat`

   If you do not have the `TMDSG006_2.dat` file generated in part 1, use the provided `TMDSG006_2_ref.dat` file.

Defining Sizing Optimization

3. Select the **Design** tab
4. From the category chooser, select **Quick Setup Trails**

   A summary of the design data is shown in the panel, it should now indicate that at this point there is no design data.
5. Push the **Quick Sizing Setup** button
6. Select **PSHELL** 5, 6 and 7 from the property list

   All elements should now be highlighted.
7. Push **Next>**
8. Accept defaults of `0.0` in **C1** and `1.0` in **M1** fields

   This will make the initial value to be the same, as specified by the analysis data.
9. Enter `0.1` in **C2** field and `0.0` in **M2** field

   This will make the lower bound value to be 0.1 for all designable thicknesses.
10. Enter `1.2` in **C3** field and `0.0` in **M3** field

    This will make the upper bound value to be 1.2 for all designable thicknesses.
11. Push the **Finish** button.

    The summary now shows that there are 3 design variables and 3 sizing regions defined.

Defining the Design Objective

12. From the category chooser, select **Objectives**
13. Push **New Objective** button from the Edit menu toolbar

14. For **Name,** enter `Strain_Energy`
15. Select the **Strain Energy**, as the response type
16. Accept the **Min** Objective Definition switch
17. Push **Next>**
18. Select the existing loadcase
19. Push the **Finish** button

Defining the Design Constraints

20. From the category chooser, select **Constraints**
21. Push **New Constraint** button from the Edit menu toolbar
22. For **Name,** enter `Volume`
23. Select the **VOLUME** response
24. Enter `600.0` for the **Upper Bound**
25. Push the **Finish** button

Request the UPDATE File

The UPDATE file is a Genesis input file that contains the updated data entries. In this case the key updated entries are the PSHELLs.

26. From the main menu bar, select **Genesis** → **Options**
27. Select the **File Control** tab
28. For **Updated Input File**, pick **Last Cycle**

    The Last Cycle option will create one updated file corresponding to the last design cycle. The name of the file will be: `TMDSG006_3_dsgUPDATExx.dat` (where `xx` is the last design cycle number).

29. Push the **Apply** button

Save the Design Studio Database File

30. From the main menu bar, select **File** → **Save As...**

    Use `TMDSG006_3` name

Optimize the Structure Using Genesis

31. From the main menu bar, select **Genesis** → **Optimize**

32. Study the **Genesis Console Output**; when done, push the **Close** button

    After Genesis finishes the run, check the volume of the structure.

Briefly Study the Objective and Constraints Values

33. In a text editor, load the file: `TMDSG006_3_dsg.out`
34. Complete the following table:

Result	Topometry Reference Solution (1)	Sizing Solution (2)
Strain Energy (Initial Design)	242.8	
Volume (Initial Design)	655.4	
Strain Energy (Final Design)	257.9	
Volume (Final Design)	598.2	

(1) Result from `TMDSG006_3_ref.out`
(2) Result from your run, `TMDSG006_3_dsg.out`

Briefly Study the Thickness and the UPDATE File

35. In a text editor, load the file: `TMDSG006_3_dsgUPDATExx.dat`

36. Complete the following table:

	Initial Solution (1)	Sizing Reference Solution (2)	Sizing Solution (3)
PSHELL 5 thickness	1.2	1.02	
PSHELL 6 thickness	1.1	1.00	
PSHELL 7 thickness	0.8	0.77	

(1) Result from `TMDSG006_2_ref.out`
(2) Result from `TMDSG006_3_refUPDATExx.dat`
(3) Result from your run, `TMDSG006_3_dsgUPDATExx.dat`

Quit Design Studio

37. From the main menu bar, select **File** → **Quit**
38. Push the **Don’t Save** button

Create a Genesis Input file

39. Copy the file `TMDSG006_3_dsgUPDATExx.dat to TMDSG006_4.dat`

    The UPDATE file contains valid bulk data, but non valid executive and solution control data. To create a working file, you need to add valid Executive Control and Solution control entries. You will do this in the next step.

40. Replace the Executive Commands and the Solution Control commands in `TMDSG006_4.dat` for the ones in `TMDSG006_2.dat`

    The Executive Control Commands and the Solution Control Commands correspond to all data entries above the Begin Bulk entry.

    The `TMDSG006_4.dat` should now be a working file that can be used to run Genesis.

5.7 Reinforcing a Solid Part on a Design Studio Generated Composite Skin Layer

Introduction

The main purpose of this example is for you to learn how to reinforce solid parts using composite skins. You will first learn how to generate the composite skin using Design Studio. Later you will perform topometry optimization to optimize the skin. You will solve the problem using different mass constraints to perform a trade-off study between mass and frequency gains.

Objective function of the problem:

Maximize the first elastic natural frequency (mode 7)

Subject to:

Case 1: Mass ≤ 0.384 Mg (384Kg =374 Kg + 10Kg, this will allow to add up to 2.5%)
Case 2: Mass ≤ 0.394 Mg (394Kg =374 Kg + 20Kg, this will allow to add up to 5.0%)
Case 3: Mass ≤ 0.404 Mg (404Kg =374 Kg + 30Kg, this will allow to add up to 7.5%)
Case 4: Unconstrained Maximization problem
Case 5: Unconstrained Minimization problem

Designable region:

Every element of an added skin

Analysis problem:

Natural Vibrations. The structure is free to move and it has 6 rigid body modes

The problem is divided into two parts that will help you learn/review the following tasks:

1) How to generate a surface mesh of composite elements (skin)

2) How to create a topometry optimization problem with natural frequency responses

The two parts of this example should be performed sequentially. However, if you want to skip the first part you can, as the needed results are provided.

Example ID

TMDSG007

Files Used in This Problem

A list, of the files that are either provided to you or the ones you are expected to create during this example, is presented next. It is not necessary to study the list in detail at this point. The files listed will be introduced during the example. Later, this list can be used for verifying your results.

Part	File Name	Description
1	TMDSG007_1.dat	Provided: Contains a finite element mesh with an eigenvalue loadcase.
1	TMDSG007_1_dsg.out	Generated using Genesis within Design Studio. This is a Genesis output file. It contains the values of all the frequencies below 2000Hz.
1	TMDSG007_1_dsg00.pch	Generated using Genesis within Design Studio. This file contains FE post-processing results for analysis. In this case, the FE results are eigenvectors (mode shapes).
1	TMDSG007_2.dat	Generated with Design Studio. This file contains the information in `TMDSG007_1.dat` plus an added composite skin.
1-2	TMDSG007_2_ref.dat	Provided. Result of Part1. Should be nearly identical to `TMDSG007_2.dat` and can be used to perform part 2 without performing part 1.
2	TMDSG007_3.dat	Generated using Design Studio. This file is obtained by adding optimization data to the `TMDSG007_2.dat` file.
2	TMDSG007_3_dsgxy.pch	Generated using Genesis within Design Studio. This file contains the mode shape for post-processing results for design cycle xy.
2	TMDSG007_3_dsgOPOSTxy.pch	Generated using Genesis within Design Studio. This file contains the density of each designable element, for design cycle xy.
2	TMDSG007_3_ref.dat	Provided. Result of Part2. Should be nearly identical to `TMDSG007_3.dat` and can be used to check part 2.
2	TMDSG007_4_ref.dat	Provided.This file is identical to `TMDSG007_3_ref.dat` but the mass constraint has been changed to 0.394
2	TMDSG007_5_ref.dat	Provided.This file is identical to `TMDSG007_3_ref.dat` but the mass constraint has been changed to 0.404
2	TMDSG007_6_ref.dat	Provided.This file is identical to `TMDSG007_3_ref.dat` but the mass constraint has been deleted.
2	TMDSG007_7_ref.dat	Provided. This file is identical to `TMDSG007_6_ref.dat` but the objective function has been changed from Max to Min,.

Benefits of Trade-off Studies

Trade-off studies are useful because they allow you to learn the impact of selected quantities in the objective or in the constraints. In this case, you will learn the impact of using different mass constraints on the objective of this problem.

5.7.1 Part 1

The purpose of this part of the example is to learn how to create a composite skin around solid parts. You will also study the natural frequency and the mode shapes of the structure. If you are familiar with these tasks, you can skip this part and go to Part 2.

When you finish this example, you should have created a file named: `TMDSG007_2.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG007_1.dat`

Analyze the Structure Using Genesis

3. From the main menu bar, select **Genesis** → **Single Analysis**

   Study the **Genesis Console Output**; when done, push the **Close** button.

Import the Post-Processing File

4. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
5. Select the `TMDSG007_1_dsg00.pch` file
6. Push the **Open** button

Post-Processing the Results (Mode Shape)

7. Select the **Post** tab
8. Push the **Deform Mesh/Color Mesh** button
9. Push the **Filled Contours** radio button, this button is in the Color Mesh Window
10. From the **Color Mesh** list, select Mode 1 result for design cycle 0
11. To animate, push the **Oscillate** radio button
12. Verify that the first 6 modes are rigid body modes
13. Inspect mode 7 and some others
14. Verify that mode 7 is the first elastic mode
15. What is the frequency of mode 7?

    Hint: Read the value in the upper left corner of the Viewport. Reference answer is approximately: 332.43 Hz.

16. Push the **Up** button

Study the Output File

17. Start a text editor
18. In the text editor load the Genesis data file: `TMDSG007_1_dsg.out`
19. Study briefly the file
20. Using the output file complete the following table:

Quantity	Solid Parts Only Reference Solution (1)	Solid Parts Only (2)
Frequency 7 [Hz]	332.43	
Mass [Kg]	374.34	

(1) Result from `TMDSG007_1.out`
(2) Result from your run, `TMDSG007_1_dsg.out`
Your values should be very close to the reference values.
Note: The units in Genesis for this problem are in Megagrams. 1Kg=0.001*Megagram

21. Close the output file

Create a New Material for Bonding

22. Select the **Analysis** tab
23. From the category chooser, select **Materials**
24. Push **New Material** button from the Edit menu toolbar
25. Enter **Name**: `Bonding_Material`
26. Select **Isotropic(MAT1)**
27. Push **Next>**
28. For the **E** enter `20000.0` (twenty thousand)
29. For **Nu** enter `0.28`
30. Push the **Finish** button

Verify that there are now two materials in the Materials list.

Create a New Group

31. Select the **Display** tab

32. Push the **Manage Groups** button
33. Push **New Group** button from the Edit menu toolbar
34. Enter Name: `Composite_Reinforcement_Skin`
35. For Type, select **PCOMP**
36. Select Yellow Color
37. Push the **Finish** button
38. Push the **Up** button

Assign Property Values to the New Group

39. Select the **Analysis** tab
40. From the category chooser, select **Group Properties**
41. Select the **PCOMP** you just created, `Composite_Reinforcement_Skin`
42. Push the **Modify Group Property** button from the Edit menu toolbar
43. For **Z0**, Enter `0.0`

    The composite layer will grow from the grids out.

44. For layer 1, change the material to: `Bonding_Material`
45. For **Thickness** of **Layer** 1: Accept `1.0`
46. Push the **+** button to add a new layer
47. For layer 2, select MAT 1 Steel
48. For **Thickness** of **Layer** 2: Enter `12.0`
49. Push the **Finish** button

Create the Skin

50. From the **Analysis** category chooser, select **Elements**
51. Push **New Elements** button from the Edit menu toolbar
52. Select the **Surface elements from Selected Solid Elements** option
53. Push **Next>**
54. Push the **Select All** button

    Verify that there are 5442 elements selected.

55. Push **Next>**
56. Select the PCOMP group

57. Push the **Finish** button, to finish the creation of the skin

    The skin is hidden, as it uses same grids of the solid parts.

58. Push the **Select None** button

Visualize the Skin

59. Select the **Display** Tab
60. Push the **Show/Hide Groups** button
61. Hide the PSOLID groups

    Notice that the shell elements forming the skin are shown in yellow

62. Push the **Up** button
63. For the **Model Cutaway**, select the **Cut V.C.S, X axis, hide - side** option
64. Use the slider below to change the cutting plane

    Notice that the only a layer of shell is created for the surface solid elements.

65. For the **Model Cutaway**, select the **None** option to view the entire model
66. Push the **Show/Hide Groups** button and select the Show All button to display all groups

Orientation of the Skin

67. Select the **Analysis** Tab
68. From the category chooser, select **Elements**
69. Push the **Select All** button

    Now all elements of the skin should be selected.
    Verify that there are 2872 elements selected.

70. Push the **Generate Orientations Vectors** button

    Verify that all vectors are pointing out of the structure. If there are orientation vectors that do not point outward, you can fix them by: 1) selecting the elements to be fixed, 2) pushing the **Modify Elements** button from the Edit Menu toolbar, 3) selecting the **Flip 2-D Elements' Orientation** option, and finally 4) pushing the **Finish** button.
    Once the orientation vectors are pointing outward of the structure and because Z0=0.0, then the composite will "grow" out of the existing solid part of the structure.

Clear the Orientation Vectors

71. Right-click the Viewport, select **Clear**→ **All**

Save the Design Studio Database File

72. From the main menu bar, select **File** → **Save As...**
73. Enter `TMDSG007_2` as the Filename and push **Save (**as a Design Studio File**)**

Export the Input File

74. From the main menu bar, select **File** → **Export** → **Input Data...**
75. Enter `TMDSG007_2.dat`
76. Push the **Save** button

Analyze the Structure

77. From the main menu bar, select **Genesis** → **Single Analysis**

Quit Design Studio

78. From the main menu bar, select **File** → **Quit**

Study the Output File

79. In a text editor, load the file: `TMDSG007_2_dsg.out`
80. Complete the following table:

Quantity	Solid Parts + Skin Reference Solution (1)	Solid Parts + Skin (2)
Frequency 7 [Hz]	336.54	
Mass [Kg]	636.82	

(1) Result from `TMDSG007_2_ref.out`
(2) Result from `TMDSG007_2_dsg.out`

81. Has the first elastic frequency (frequency 7) increased?

    Reference answer: Yes. The frequency was increased by 4.3 Hz (1%).

82. What does this result mean?

    Reference answer: Adding the composite skin has not improved significantly the first elastic mode. This is despite that a significant mass was added. Added Mass=636.82-374.34=262.48Kg.

5.7.2 Part 2

The main purpose of this part of the example is to learn how to create the necessary design data in order to perform topometry optimization.

If you do not have the `TMDSG007_2.dat` file generated in part 2, copy the file `TMDSG007_2_ref.dat` to `TMDSG007_2.dat`.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG007_2.dat`

Review the Skin

3. Select the **Display** Tab
4. Push the **Show/Hide Groups** button
5. Hide the PSOLID groups

   The skin is in yellow.
6. Push the **Up** button

Create a Discrete Design Variable

7. Select the **Design** tab
8. From the **Design** category chooser, select **Design Variables**
9. Push **New Design Variable** button from the Edit menu toolbar
10. Leave `X1` as the Name and select **Discrete Design Variables**
11. Push **Next>**
12. Enter `1.0` for **Initial Value**
13. Enter `0.01` for **Lower Bound**, and `1.0` for **Upper Bound**
14. Enter `0.01` for **Discrete Value**
15. Push the **+** button to add an additional discrete value
16. Accept the value `1.0` for second discrete value
17. Push the **Finish** button

    Notice that an asterisk (*****) is in front of the discrete design variable, which indicates that this design variable is not being used yet. The **"D"** next to the asterisk indicates this is a **D**iscrete design variable.

Create an Equation Design Variable

18. Push **New Design Variable** button from the Edit menu toolbar to create a new design variable
19. Enter `12*X1` as the Name and select **Equation Design Variable**
20. Push **Next>**
21. Replace "F =" by: `F=12*Arg1`
22. Select (highlight) the Design Variable `1 x1` from the **Master Variable** list
23. In the **Equation Parameter** list, replace `Arg1` by `X`, then push the keyboard key Return or Enter

    This step is optional.
    Notice that if you replace Arg1 by X, Design Studio will automatically update the equation from `F=12*Arg1` to `F=12*X`. The name of the Design Variable x1 itself does not matter to show that you used x (or Arg1) as the argument instead of x1.

24. Push the **Finish** button

    Notice that there is an asterisk in front of the equation design variable, which indicates that this design variable is not being used yet. The "**E"** indicates this is an **E**quation design variable.

Create the Sizing Region

25. Stay with the **Design** tab
26. From the **Design** category chooser, select **Sizing**
27. Select the PCOMP `Composite_Reinforcement_Skin`
28. Push the **Modify Sizing Design** button from the Edit menu toolbar
29. For the **Thickness** of **Layer 1,** select design variable `1 X1`
30. For the **Thickness** of **Layer 2,** select design variable `12*X1`
31. Push the **Finish** button

    Verify that the hammer icon is next to PCOMP 5.
    The hammer icon indicates that the PCOMP group is being size-designed.

Create the Topometry Region

32. From the **Design** category chooser, select **Topometry**

    Only sizing regions are listed here. That is why only PCOMP 5 is now listed.

33. Select **PCOMP** 5

    Verify that there is no hammer icon next to the PCOMP label.
    Here, the absence of a hammer icon means that the group has not yet been topometry designed.

34. Push the **Modify Topometry Design** button from the Edit menu toolbar

35. Push the **Finish** button

    Verify that the hammer icon is now next to the PCOMP 5 label.
    Here, the hammer icon means that the group is now being topometry designed.

Defining the Design Objective

Next, you will create one objective function which will be: to Maximize the first elastic mode (mode 7).

36. From the category chooser, select **Objectives**

37. Push **New Objective** button from the Edit menu toolbar

38. For Name enter: `Freq_7`

39. Select **Frequency Mode Number** response type

40. For the **Frequency Mode Number,** enter `7`

41. Select the **Max** for the **Objective Definition Switch**

42. Push **Next>**

43. Select the existing loadcase

44. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Defining the Design Constraints

Next, you will create a mass constraint.

45. From the category chooser, select **Constraints**

46. Push **New Constraint** button from the Edit menu toolbar

47. Enter Name `Total_Mass`

48. Select the **Mass** response

49. Enter `0.384` for the **Upper Bound**

50. Push the **Finish** button

    Verify that now there is one response in the constraints list.

Increase the Maximum Number of Design Cycles

51. From the main menu bar, select **Genesis** → **Options...**
52. Select the **Design Control** tab
53. For **Maximum Number of Design Cycles**, enter `25`

    The default for the maximum number of design cycles has been changed to 25.
54. Push the **Apply** button

Save the Design Studio Database File

55. From the main menu bar, select **File** → **Save As...**
56. Enter `TMDSG007_3` as the Filename and push **Save (**as a Design Studio File**)**

Export a Genesis Input Data File

57. From the main menu bar, select **File** →**Export** → **Input Data...**
58. Enter `TMDSG007_3` as the Filename and push **Save (**as a Genesis File**)**

Study the Genesis Input File

59. In a text editor, load the file: `TMDSG007_3.dat`

The following design data should be written in the Genesis input data

DOPT	25								
DVAR	1	X1	1.0	0.01	1.0				
+	1								
DVSET	1	0.01	1.0						
DVPROP4	1	5							
+	Layer1		1						
DVPROP2	2	5	102	2					
+	DVAR	1							
DEQATN	2	F(x)=	12*x						
DSPLIT	5								
DOBJ	1	Freq_7	14	MAX					
DRESP1	1	Freq_7	FREQ	7					
DCONS	2			0.384					
DRESP1	2	tal_Mass	MASS						

Briefly verify this information. If something does not match, fix it using Design Studio. Note: In Genesis the labels use only 8 characters, that is why the “Total_Mass” Design Studio label was written as “tal_Mass”.

Using Design Studio, you do not need to know in great detail the Genesis format. However, if there are error messages reported by Genesis, you might need to study the input file in order to fix the problem. Most of the time Genesis error messages point to the appropriate data entries, that usually help to determine which data needs fixing.

60. What would happen if you delete the DSPLIT entry and run Genesis with the modified file?

    Answer: You will run a sizing optimization problem, not a topometry one. In this case, only one independent discrete design variable would be used.

61. What would happen if you delete the DRESP1 2 and DCONS 2 entries and you run Genesis with the modified file?

    Answer: You would run an unconstrained optimization problem.

62. What would happen if you delete the DVPROP2 entry and you run Genesis with the modified file?

    Answer: The thickness associated with layer 2 of PCOMP5 would not be designable and it would not change its original value of 12 mm.

Optimize the Structure

63. From the main menu bar, select **Genesis** → **Optimize**

Import the Design Post-Processing Files (OPOST Files)

64. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
65. Select the `TMDSG007_3_dsgOPOST00.pch` file and check off the **Import Similar Results for All Design Cycles** checkbox
66. Push the **Open** button

Post-Processing the Design (Thicknesses) Results

67. Select the **Post** tab
68. Push the **Deform Mesh/Color Mesh** button
69. Select the **Thickness** Result for the last cycle
70. Push the **Options...** button
71. Check off the **Hide Elements With no Value...** checkbox

    This step is necessary only if the solid elements are not hidden.
72. Push the **Close** button
73. What is the largest thickness?

    Hint: Right-click the viewport, select **List Top 10**, then read the numbers in the Design Studio Message Window.

Import the Analysis Post-Processing (Modes) Files

74. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
75. Select the `TMDSG007_3_dsg00.pch` file and check off the **Import Similar Results for All Design Cycles** checkbox
76. Push the **Open** button

Post-Processing the Results (Mode Shapes)

77. Animate modes 7, 8 and 9 for design cycle 0 and for the last design cycle
78. Push the **Up** button

Clean the Post-Processing Results

79. Push the **Manage Result Dataset** button
80. From the main menu bar, select **Edit** → **Select All**

81. From the Edit menu toolbar, select **Delete Result Set** button
82. Push the **Up** button

Study the Output File

83. In a text editor load the file: `TMDSG007_3_dsg.out`
84. Complete the following table:

Result	Topometry Reference Solution (1)	Topometry Solution (2)
Frequency 7 [Hz]	343.04	
MASS [Kg]	382.72	

(1) Result from reference `TMDSG007_3_ref.dat`
(2) Result from your run, `TMDSG007_3_dsg.out`

85. Study the design cycle history printed at the end of the `TMDSG007_3_dsg.out` file

Trade-Off Study

You will now perform additional optimization runs.

Case 2: Change the Design Constraints

86. Select the **Design** tab
87. From the category chooser, select **Constraints**
88. Select the existing constraint: `Total_Mass`
89. Push the **Modify Constraint** button from the Edit menu toolbar
90. Enter `0.394` for the **Upper Bound**
91. Push the **Finish** button

Case 2: Save the Design Studio Database File

92. From the main menu bar, select **File** → **Save As...**
93. Enter `TMDSG007_4` as the Filename and push **Save (**as a Design Studio File**)**

Case 2: Re-Optimize the Structure

94. From the main menu bar, select **Genesis** → **Optimize**

95. Complete the following table:

Result	Topometry Reference Solution (1)	Topometry Solution (2)
Frequency 7 [Hz]	354.67	
MASS [Kg]	392.70	

(1) Result from `TMDSG007_4_ref.out`
(2) Result from your run, `TMDSG007_4_dsg.out`

Case 3: Change the Design Constraints

96. Change the upper bound of the mass constraint to `0.404`

97. Save the data base using the following name: `TMDSG007_5`

98. Re-optimize the problem

99. Complete the following table

Result	Topometry Reference Solution (1)	Topometry Solution (2)
Frequency 7 [Hz]	361.29	
MASS [Kg]	402.31	

(1) Result from `TMDSG007_5_ref.out`
(2) Result from your run, `TMDSG007_5_dsg.out`

Case 4: Unconstrained Maximization

Delete the Mass constraint.

100. Select the **Design** tab

101. From the category chooser, select **Constraints**

102. Select the constraint

103. From the Edit menu toolbar, select **Delete Constraint** button

104. Save the database using the following name: `TMDSG007_6`

105. Re-optimize the problem

106. Complete the following table:

Result	Topometry Reference Solution (1)	Topometry Solution (2)
Frequency 7 [Hz]	391.93	
MASS [Kg]	553.01	

(1) Result from `TMDSG007_6_ref.out`
(2) Result from your run, `TMDSG007_6_dsg.out`

Case 5: Unconstrained Minimization

107. From the category chooser, select **Objective**
108. Select the objective function
109. Push the **Modify Objective** button from the Edit menu toolbar
110. Change the objective type to MIN
111. Push the **Finish** button
112. Save the database using the following name: `TMDSG007_7`
113. Re-optimize the problem
114. Complete the following table:

Result	Topometry Reference Solution (1)	Topometry Solution (2)
Frequency 7 [Hz]	277.61	
MASS [Kg]	485.85	

(1) Result from `TMDSG007_7_ref.out`
(2) Result from your run, `TMDSG007_7_dsg.out`

Trade-Off Study Table

115. Using the previous tables, complete the first two empty columns of the following table, then optionally calculate the last four columns

Case	Note	Mass	Objective Function (Freq 7)	Mass Added	Objective Function Gain (Freq 7)	Mass Added	Objective Function Gain (Freq 7)
				Mass-374.34	Freq-332.43	(Mass-374.34)/3.7434	(Freq-332.24)/3.3243
		[Kg]	[Hz]	[Kg]	[Hz]	%	%
No Skin	(1)			-	-	-	-
Case 1	(2)						
Case 2	(3)						
Case 3	(4)						
Case 5 Unconstraint Minimization	(5)						
Case 4 Unconstraint Maximization	(6)						
Full Skin	(7)						

(1) Result from your run: `TMDSG007_1_dsg.out`
(2) Result from your run: `TMDSG007_3_dsg.out`
(3) Result from your run: `TMDSG007_4_dsg.out`
(4) Result from your run: `TMDSG007_5_dsg.out`
(5) Result from your run: `TMDSG007_7_dsg.out`
(6) Result from your run: `TMDSG007_6_dsg.out`
(7) Result from your run: `TMDSG007_2.out`

Reference Trade-Off Study Table

116. Reference table

Case	Note	Mass	Objective Function (Freq 7)	Mass Added	Objective Function Gain (Freq 7)	Mass Added	Objective Function Gain (Freq 7)
				Mass-374.34	Freq-332.43	(Mass-374.34)/3.6434	(Freq-332.24)/3.3243
		[Kg]	[Hz]	[Kg]	[Hz]	%	%
No Skin	(1)	374.34	332.43	-	-	-	-
Case 1	(2)	382.72	343.04	8.38	10.61	2.24	3.19
Case 2	(3)	392.70	354.67	18.36	22.24	4.90	6.69
Case 3	(4)	402.31	361.28	27.97	28.85	7.47	8.68
Case 5 Unconstraint Minimization	(5)	485.85	277.61	111.51	-54.82	29.79	-16.49
Case 4 Unconstraint Maximization	(6)	543.01	391.93	168.67	59.50	45.06	17.90
Full Skin	(7)	636.82	336.54	262.48	4.11	70.12	1.24

(1) Result from reference run using: `TMDSG007_1_dsg.out`
(2) Result from reference run: `TMDSG007_3_ref.out`
(3) Result from reference run: `TMDSG007_4_ref.out`
(4) Result from reference run: `TMDSG007_5_ref.out`
(5) Result from reference run: `TMDSG007_7_ref.out`
(6) Result from reference run: `TMDSG007_6_ref.out`
(7) Result from reference run: `TMDSG007_2_ref.out`

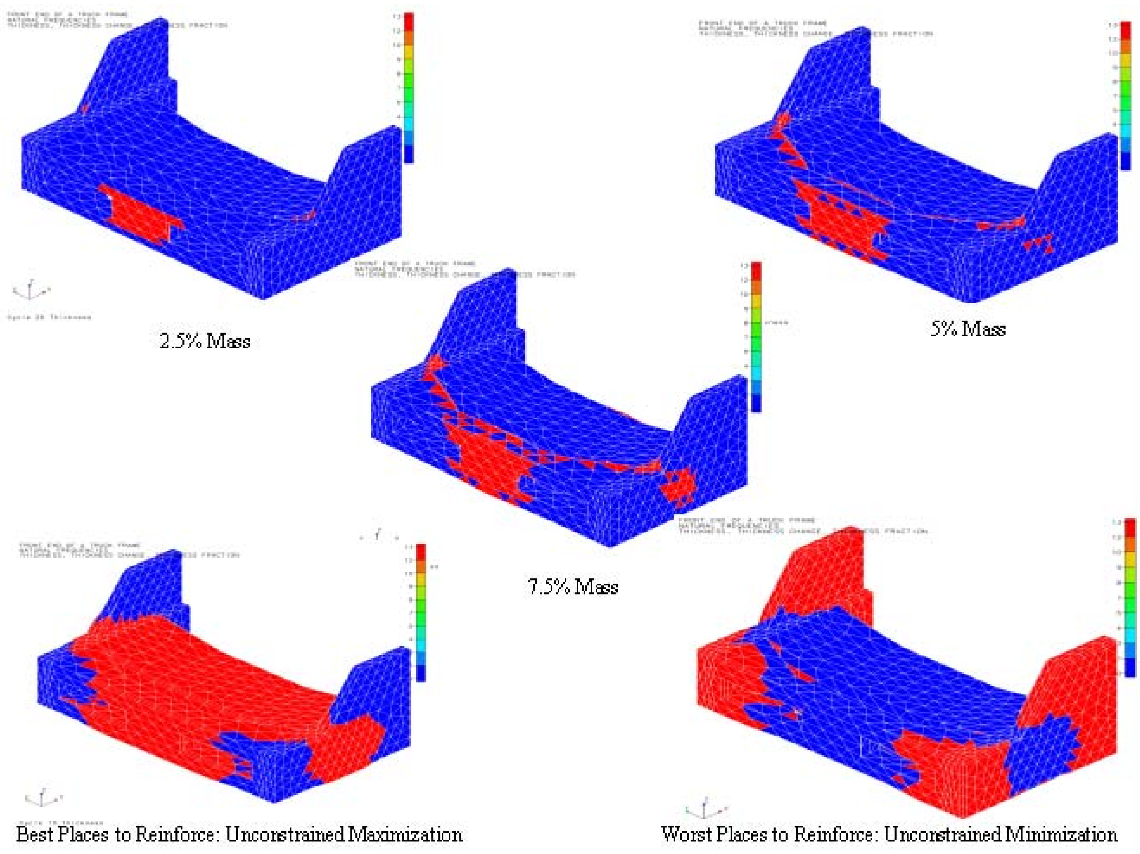
Case 1, 2 and 3 gave us the trade-off information for adding about 2.5%, 5% and 7.5% of mass.
Case 4 represents the best gains you could have expected using the best parts of the skin. This case can be used to understand the upper bound on the objective.
Case 5 shows that by reinforcing the wrong places, the vibration modes can be worse than not reinforcing at all. This case gives a lower bound on the vibration for the given skin.

Note:

In this example you plotted the total thickness of the composite. If individual thicknesses are needed, you will need to create a dummy static loadcase, request stresses for that loadcase, re-run Genesis and post-process the stresses. The thicknesses of individual layers are stored with the stresses.

Trade-Off Study (Alternatives) Solutions

The following picture shows the three constrained optimization problems and the two unconstrained ones. In the figure red represents a reinforcement of 13 mm (12+1) while blue represents the areas with no reinforcement.



5.8 Topometry Optimization of Welds

Introduction

The main purpose of this example is to learn how to set up a topometry optimization problem to select the best welds to keep from a pool of candidate welds. This example will teach you how to create synthetic responses, equation design variables, and how to create and use smart assemblies. The welds in this example are given. The welds are modeled using CBUSH elements.

Objective function of the problem:

Maximize the first natural torsional frequency

Note: Other objectives are also possible. Since a high value for the global torsional frequency is often desired, it is used in this example. Other option could be; Maximize the sum of the first global bending and the first global torsional frequency.

Subject to:

Selection of the 10 best welds out of 22 candidate welds. This is accomplished by defining a DSELECT Synthetic Constraint. The bound of the constraint is obtained by dividing the number of welds to select by the number of candidates (10/22 = 0.45)

DSELECT Constraint=< 0.455

Note: By defining the DSELECT constraint, genesis internally created the two constraints below. These two constraints will force, at the end of the optimization, 10 design variables Xi to be close to 1.0 and 12 design variables Xi to be close to 0.0. Other values would violate one or both constraints.

(X1+X2 +... +X22)/22=< 0.455 (synthetic response 1: AVG1)

(X1**3+X2**3 +... +X22**3)/22>= 0.455 (synthetic response 2: AVG3)

Independent Designable variables:

0.00001<=Xi<=1.0; Xi_initial=0.455 (design variables)

Equation design variables:

Ktrans=100,000*Xi**3

Krot=1,000,000*Xi**3

Designable region:

Every CBUSH element

The translational stiffness (K1,K2,K3) and rotational (K4,K5,K6) stiffness corresponding to the PBUSH properties are linked to the equation design variables using the following equations:

K1i=Ktrans (=100,000*xi**3)

K2i=Ktrans (=100,000*xi**3)

K3i=Ktrans (=100,000*xi**3)

K4i=Krot= (1,000,000*xi**3)

K5i=Krot (=1,000,000*xi**3)

K6i=Krot (=1,000,000*xi**3)

Note: When the design variable xi is 1.0, K1i=K2i=K3i=100,000.0 and K4i=K5i=K6i=1,000,000.0. When the design variable xi is close to the lower bound, all Ki terms are close to 0.0.

Analysis problem:

Natural Vibrations. The structure is free to move and it has 6 rigid body modes

The problem is divided into three parts that will help you learn the following tasks:

1) How to create an eigenvalue loadcase with mode tracking and how to post-process vibration modes.

2) How to create a topometry optimization problem with natural frequency responses, with synthetic responses and with equation design variables.

3) How to create an updated input file that contains the optimal results. With this part you will also learn how to create assemblies.

The 3 parts of this example should be performed sequentially. However, if you want to skip any of the parts you can, as the needed files for that are provided.

Note:

In this problem each CBUSH element is used to connect two grids, one in the top of the truck cabin, the other in the bottom of the truck cabin. The terms K1, K2 and K3 correspond to translational stiffness and the terms K4, K5 and K6 correspond to rotational stiffness.

$$K = \begin{bmatrix} K_1 & & & & & \\ & K_2 & & & & \\ & & K_3 & & & \\ & & & K_4 & & \\ & & & & K_5 & \\ & & & & & K_6 \end{bmatrix}$$

If all Ki terms are 0.0 or close to zero, the connections provided by the CBUSH vanish.

Example ID

TMDSG008

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the example. Later, this list can be used for verification.

Part	File Name	Description
1	TMDSG008_1.dat	Provided: Contains the finite element mesh of a truck cabin but without a loadcase. The top part of the cabin is welded to the bottom part using 22 bush elements.
1	TMDSG008_2.dat	Generated in this example: This file is the same as the TMDSG008_1.dat file but it contains an eigenvalue loadcase.
1	TMDSG008_2_dsg00.pch	Generated using Genesis within Design Studio. This file contains FE post-processing results for analysis. In this case, the FE results are eigenvector (mode shapes).
1	TMDSG008_2_dsg.out	Generated by Genesis run within Design Studio. This is a Genesis output file. It contains the values of all the frequencies below 30Hz.
1-2	TMDSG008_2_ref.dat	Provided. Result of Part 1. Should be nearly identical to TMDSG008_2_dsg.dat (and TMDSG008_2.dat) and can be used as a replacement of it to perform part 2 without performing part 1.
2	TMDSG008_3.dat	Generated using Design Studio. This file is obtained by adding topometry data to the TMDSG008_2.dat file.
2	TMDSG008_3_dsgxy.pch	Generated using Genesis within Design Studio. This file contains the mode shape for post-processing results for design cycle xy. For xy=00, the modes are calculated together with the bush elements with initial value of 0.455. This file allows to identify the first torsional mode.
2	TMDSG008_3_dsgUPDATExy.pch	Generated using Genesis within Design Studio. This file contains the updated PBUSH properties, for the last design cycle xy.

2-3	TMDSG008_3_ref.dat	Provided. Result of Part 2. Should be nearly identical to TMDSG008_3.dat and can be used as a replacement of it to perform part 3 without performing part 2.
3	TMDSG008_4.dat	Generated using Design Studio.This is a Genesis input data with the FE model of the structure connected with the best 10 welds (12 welds deleted).
3	TMDSG008_4_dsg00.pch	Generated using Genesis within Design Studio. This file contains the modes of the structure using the best 10 welds kept.

5.8.1 Part 1

The purpose of this part of the example is to learn how to create an eigenvalue loadcase. You will also learn how to analyze the structure and how to animate mode shapes. If you are familiar with these tasks, you can skip this part and go to Part 2.

When you finish this example, you should have created a file named: `TMDSG008_2.dat`.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG008_1.dat`

   The model has 22 unreferenced (free) grids. Their only purpose is to show where the welds are.

Hide the Free Grids

3. Select the **Display** tab
4. Select the **Hide** radio button, which is to the right of **Free Grids**

Identifying the Welds

5. Select the **Display** tab
6. Push the **Show/Hide Groups** button
7. Hide all PSHELLs

   The welds of the problems are modeled using 22 CBUSH elements. All of the bush elements reference PBUSH 801.

8. Push the **Show All** button

Define the Eigenvalue Method

You will define the method to calculate the eigenvalue.

9. Select the **Analysis** tab
10. From the category chooser, select **Eigenvalue Methods**
11. Push **New Eigenvalue Method** button from the Edit menu toolbar
12. Enter `SMS-30HZ` in the **Name** field

    One can choose any of the eigenvalue methods available in GENESIS.
    Normally. Lanczos and SMS are faster and efficient that Subspace iteration. However for

large problems, SMS tends to be faster than Lanczos

13. Push the **Finish** button (accepting the defaults)

    Verify that there is now one eigenvalue method in the list.

Create the Loadcase

The eigenvalue method just created is not used, unless it is selected in a loadcase. You will now create one loadcase that uses the previous created eigenvalue method.

14. Select the **Analysis** tab
15. From the category chooser, select **Loadcases**
16. Push **New Loadcase** button from the Edit menu toolbar
17. Enter Name `Eigenvalue_Loadcase`
18. Select **Normal Modes** as the loadcase type
19. Push **Next>**

    This problem has no boundary condition.

20. Push **Next>**
21. For **Eigenvalue Method**: select `1 SMS-30HZ`
22. For **Mode Tracking**: select `Yes`

    The option YES, will track only the modes of interest in the optimization problem that will be created in Part 2. The option ALL would track all the modes below 30Hz.

23. Push **Next>**
24. For Displacement: select `Post`

    This allows post-processing of the mode shapes.

25. Push the **Finish** button

    Verify that the loadcase you just created is in the list of Loadcases.

Delete the Default Loadcase

When there are no loadcases in the original input data you read, Design Studio generates a default static loadcase. Since in this case this loadcase will not be used, simply delete it as is shown next.

26. Select the Static Loadcase in the loadcases list
27. From the Edit menu toolbar, select **Delete Loadcase** button

    Verify that the loadcase list has only the loadcase you created.

Change the Subtitle of the Project

This part is optional.

28. From the main menu bar, select **Genesis** → **Options**
29. Select the **Output Control** tab
30. Change the **Subtitle** to: `Vibration Analysis`
31. Push the **Apply** button

Export the Input File

From time to time it is good practice to save intermediate steps. Now, would be a good time since you have a complete executable analysis file.

32. From the main menu bar, select **File** → **Export** → **Input Data...**
33. Enter `TMDSG008_2.dat`
34. Push the **Save** button

Save the Design Studio Database File

35. From the main menu bar, select **File** → **Save As...**
36. Enter `TMDSG008_2` as the Filename and push **Save (**as a Design Studio File**)**

    Note that this will change the database name to `TMDSG008_2` so the post-processing file will use this name instead of `TMDSG008_1.`

Analyze the Structure Using Genesis

37. From the main menu bar, select **Genesis** → **Single Analysis**

    Study the **Genesis Console Output**; when done, push the **Close** button.

Import the Post-Processing Files

38. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
39. Select the `TMDSG008_2_dsg00.pch` file
40. Push the **Open** button

Post-Processing the Results (Mode Shape)

41. Select the **Post** tab
42. Push the **Deform Mesh/Color Mesh** button

43. Push the **Filled Contours** radio button, this button is in the Color Mesh Window
44. From the Color Mesh list, select Mode 1 result for cycle 0 (analysis results)
45. To animate, push the **Oscillate** radio button

    Animate all other modes.

46. Verify that the first 6 modes are rigid body modes

    Frequency values should be near zero.

47. Find out which is the torsional mode
48. Write down the number of the mode and the value of the torsional frequency

    The value is written in the screen (and also in the `TMDSG008_2_dsg00.out file).` Since our goal of the problem is to optimize the torsional frequency, you will need to use that number.

File	Mode Number	Frequency Value
`TMDSG008_2_dsg00.out`		

Reference Answers:

File	Mode Number	Frequency Value
`TMDSG008_2_ref.out`	8	8.309 Hz

Your answer and the reference answer should be very similar (computer round-off error can change it slightly).

49. Push the **Up** button

    Now the animation stopped.

Quit Design Studio

50. From the main menu bar, select **File** → **Quit**

Study the Input data File

51. Start any text editor
52. In the text editor load the Genesis data file: `TMDSG008_2.dat`
53. Study briefly the file
54. Study the loadcase you just created
55. Study the EIGR data statement you just created

56. Study how many CBUSH entries are in the file
57. Study how many PBUSH entries are in the file

    Verify that K1=K2=K3=100,000 and K4=K5=K6=1,000,000.

58. Close the file

Study the Output File

59. Start any text editor
60. In a text editor load the Genesis data file: `TMDSG008_2_dsg.out`
61. Study briefly the file
62. Check the value of the natural frequencies 1 to 6 and 8
63. Close the file

5.8.2 Part 2

The purpose of this part of the example is to create the necessary topometry data and to perform topometry optimization of the welds (the welds are modeled using CBUSH elements).

If you do not have the `TMDSG008_2.dat` file generated in part 1, copy the file `TMDSG008_2_ref.dat` to `TMDSG008_2.dat`.

When you finish this example, you should have created a file named: `TMDSG008_3.dat.`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG008_2.dat`

Create an Independent Design Variable

3. Select the **Design** tab
4. From the **Design** category chooser, select **Design Variables**
5. Push **New Design Variable** button from the Edit menu toolbar to create a new design variable
6. Leave `X1` as the Name and select **Independent Design Variable**
7. Push **Next>**
8. Enter `0.455` for **Initial Value**, `0.00001` for **Lower Bound**, and `1.0` for **Upper Bound**

   Note: The initial value of 0.455 is used because 10/22 is approximately 0.455. The lower bound is simply a small number. You will not use 0.0 because you do not want to create a singularity in the FE mesh.

9. Push the **Finish** button

   Notice that there is an asterisk in front of the independent design variable, it indicates that this design variable is not being used. The "**I**" indicates this is an **I**ndependent design variable.

Create two Equation Design Variables

10. From the **Design** category chooser, select **Design Variables**
11. Push **New Design Variable** button from the Edit menu toolbar to create a new design variable
12. Enter `Ktrans` as the Name and select **Equation Design Variable**

13. Push **Next>**
14. Enter `X1` as the **Equation Parameter** name
15. Enter `F=100000*X1**3`
16. Push the **Finish** button

    Notice that there is an asterisk in front of the equation design variable, it indicates that this design variable is not being used. The **"E"** indicates this is an **E**quation design variable.

17. Select (highlight) the equation design variable you just created
18. From the Edit menu toolbar, select **Copy Design Variable** button
19. From the Edit menu toolbar, select **Paste Design Variable** button
20. Select the new equation design variable (`Copy of Ktrans`)
21. Push the **Modify Design Variable** button from the Edit menu toolbar
22. Enter `Krot` as the Name
23. Push **Next>**
24. Change the equation to: `F=1000000*X1**3`
25. Push the **Finish** button

    Verify that there are three design variables in the list. The two equation design variables are unused at this point.

Create the Sizing Region

26. Select the **Design** tab
27. From the **Design** category chooser, select **Sizing**
28. Select PBUSH 801 from the list
29. Push the **Modify Sizing Design** button from the Edit menu toolbar to add design attributes

    There are 17 menus, these menus have no selections because you have not defined any design variables yet.

30. Select `Ktrans` for the **K1, K2** and **K3** menus
31. Select `Krot` for the **K4, K5** and **K6** menus
32. Push the **Finish** button

    Verify that the hammer icon is next to the PBUSH 801.
    The hammer icon indicates that the group is being size-designed.

Create the Topometry Region

33. Stay with the **Design** tab
34. From the category chooser, select **Topometry**
35. Select PBUSH 801
36. Push the **Modify Topometry Design** button from the Edit menu toolbar
37. Push the **Finish** button

    Verify that the hammer icon is next to the PBUSH 801.
    The hammer icon now indicates that the group is being topometry-designed.

Defining the Design Objective

38. From the category chooser, select **Objectives**
39. Push **New Objective** button from the Edit menu toolbar
40. Select **Frequency Mode Number** response type
41. For the **Frequency Mode Number,** enter 8

    You will need to verify this number. After the design data has been added, the mode number corresponding to torsion could change. For now you use 8. Note, that if the initial value of the design variables would have been 1.0, the number 8 would not need to be verified, as the initial values of all PBUSHs would be the same as in the original input data.

42. Select the **Max** Objective Definition switch
43. Push **Next>**
44. Select the existing loadcase
45. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Set up the DSELECT Synthetic Constraints

46. From the **Design** category chooser, select **Synthetic Responses**
47. Push **New Synthetic Response** button from the Edit menu toolbar to create a new response
48. Select **Design Variable Selection (DSELECT)** as the response type
49. Push **Next>**
50. Push the + button to add the design variables
51. Select the X1 design variable from the dropdown listbox
52. Push **Next>**

53. Enter `0.455` in the **Fraction** field

    Use 0.455 because it corresponds to 10/22. We want to select the 10 best weld locations out of 22 candidates.

54. Enter `0.005` for the **Tolerance**

    This corresponds to a tolerance over the fraction value entered. (1% or half percent could be a good guess, as well).

55. Push **Finish**

Save the Design Studio Database File

56. From the main menu bar, select **File** → **Save As...**
57. Enter `TMDSG008_3` as the Filename and push **Save (**as a Design Studio File**)**

Export the Input File

58. From the main menu bar, select **File** → **Export** → **Input Data...**
59. Enter `TMDSG008_3.dat`
60. Push the **Save** button

Analyze the Structure Using Genesis

You will perform one single analysis to check if mode 8 is still the torsional mode.

61. From the main menu bar, select **Genesis** → **Single Analysis**
62. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

63. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
64. Select the `TMDSG008_3_dsg00.pch` file
65. Push the **Open** button

Post-Processing the Results (Mode Shape of Analysis)

66. Select the **Post** tab
67. Push the **Deform Mesh/Color Mesh** button
68. Push the **Filled Contour** Button, this button is for the Color Mesh Window
69. Animate the first 7 to 10 modes searching for the torsional mode

70. Verify that the first 6 modes are rigid body modes

71. Verify that the 8th mode is still the first torsional mode

    Since the mode has not changed, our objective function is still mode 8. If the torsional mode is not 8, change the number in the Objective Function definition.

72. Push the **Up** button

Request the Update File

The following steps will cause Genesis to create two input files which represent the results of the first and last design cycle. The names of these files would be `TMDSG008_3_dsgUPDATE00.dat` and `TMDSG008_3_dsgUPDATExy.dat`, where xy corresponds to the last design cycle. The second file will be used in Part 3.

73. From the main menu bar, select **Genesis** → **Options**

74. Select the **File Control** tab

75. For **Updated Input File**, pick **First** & **Last**

76. Push the **Apply** button

Change the Maximum Number of Design Cycles

77. From the main menu bar, select **Genesis** → **Options**

78. Select the **Design Control** tab

79. For **Maximum Design Cycles:** enter `30`

80. Push the **Apply** button

Optimize the Structure

81. From the main menu bar, select **Genesis** → **Optimize**

Study the Input data with Topometry Optimization

This is the file that you just ran.

82. In a text editor load the file: `TMDSG008_3_dsg.dat`

83. How many CBUSH entries are in the file?

84. How many PBUSH entries are in the file?

85. How many DVPROP2 entries are in the file?

86. How many DEQATN entries are in the file?

87. How many DSPLIT entries are in the file?

88. How many DVAR entries are in the file?
89. How many DRESP3 entries are in the file?
90. How many DCONS entries are in the file?
91. How many DOBJ entries are in the file?
92. Close the file

Study the Update File Corresponding to Design Cycle 0

This file was created by Genesis based on the topometry data. This file is actually not used by Genesis, but it allows to check what are the values of the properties that Genesis uses in design cycle 0.

93. In a text editor load the file: `TMDSG008_3_dsgUPDATE00.dat`
94. Study how many CBUSH entries are in the file
95. Study how many PBUSH entries are in the file

    The original file welds.dat has one PBUSH element. Topometry data creates the additional 21 so each element can be designed independently.

96. Close the file

Study the Output File

97. In a text editor load the file: `TMDSG008_3_dsg.out`
98. Study briefly the “MODE TRACKING TABLE” in every design cycle. Make sure that the status is 0 at the end of the optimization

    If the status is not 0, mode tracking might have lost the mode. When losing modes, the easier fix is to re-run Genesis using smaller move limits. Sometimes, the number of modes or the upper level of frequencies V2 has to be increased as frequencies move up in value.

    The best check, to know if mode tracking has correctly tracked the mode, is to animate the modes of the last design cycle. If the mode is lost, you might find it and might be able to re-start Genesis with the new number from the last design cycle that ran without problems.

Clean the Post-Processing Results

99. Select the **Post** tab
100. Push the **Manage Result Dataset** button
101. From the main menu bar, select **Edit** → **Select All**
102. From the Edit menu toolbar, select **Delete Result Set** button

Import the Post-Processing Files (Modes of All Design Cycles)

103. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
104. Select the `TMDSG008_3_dsg00.pch` file and put a checkmark in the **Import Similar Results for All Design Cycles** checkbox
105. Push the **Open** button

Post-Processing the Results (Mode Shape of All Design Cycles)

106. Push the **Up** button
107. Push the **Deform Mesh/Color Mesh** button
108. Push the **Filled Contours** Button
109. Animate mode 8 for the last design cycle
110. Verify that the 8th mode is still the first torsional mode

    If mode 8 is still the torsional mode, you do not need to change anything.

111. Animate the mode 8 for different design cycle
112. Push the **Up** button

Import the Post-Processing Files (Design Property data)

113. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
114. Select the `TMDSG008_3_dsgOPOST00.pch` file and put a checkmark in the **Import Similar Results for All Design Cycles** checkbox
115. Push the **Open** button

Post-Processing the Results (Design Property data)

116. Push the **Deform Mesh/Color Mesh** button
117. Push the **Filled Elements** Button
118. Select the PBUSH K1 value for the first design cycle
119. Select the PBUSH K1 value for the last design cycle

    Notice the change in the values and the spread of the values in the color bar. Notice that 10 out of the 22 PBUSHs are colored red indicating that they close to 100000 while the remaining are color blue representing that they are close to 0.

120. Push the **Up** button

Quit Design Studio

121. From the main menu bar, select **File** → **Quit**
122. Push the **Don't Save** button

Study Further the Output File

123. In a text editor, load the file: `TMDSG008_3_dsg.out`
124. Complete the following table

Natural Frequency Number	Natural Frequency Original Reference Solution (1)	Natural Frequency Topometry Reference Solution (2)	Natural Frequency Topometry Results (3)
7	6.49	6.24	
8	8.31	8.17	
9	11.71	9.08	
10	12.93	9.21	

(1) Result from `TMDSG008_2_ref.dat`
(2) Result from the last design cycle in `TMDSG008_3_ref.dat`
(3) Result from the last design cycle of your run in `TMDSG008_3_dsg.out`

Notes:

The natural frequencies of the topometry results are lower because only 10 out of 22 welds are now used. The torsional frequency (mode 8) is reduced as well, but not significantly because it was maximized.

125. Optionally: Using the final values of the design variables in the history files, complete the following table:

Design Variable Number	Topometry Reference Solution(4)	Topometry Results(5)
1	0.998 ~ 1.0	
2	0.999 ~ 1.0	
3	0.999 ~ 1.0	
4	0.999 ~ 1.0	
5	0.998 ~ 1.0	
6	1E-5 ~ 0.0	
7	1E-5 ~ 0.0	
8	1E-5 ~ 0.0	
9	1E-5 ~ 0.0	
10	1E-5 ~ 0.0	
11	0.13 ~ 0.0	
12	1E-5 ~ 0.0	
13	1E-5 ~ 0.0	
14	1E-5 ~ 0.0	
15	1E-5 ~ 0.0	
16	0.987 ~ 1.0	
17	0.998 ~ 1.0	
18	0.999 ~ 1.0	
19	0.999 ~ 1.0	
20	0.998 ~ 1.0	
21	1E-5 ~ 0.0	
22	1E-5 ~ 0.0	

(4) Result from `TMDSG008_3_ref.HIS`
(5) Result from your run in `TMDSG008_3_dsg.HIS`

126. How many design variables are close to 1.0?

Reference answer: 10

127. How many design variables are close to 0.0?

Reference answer: 12

5.8.3 Part 3

The purpose of this part of the example is to learn how to create an input file containing only the most important welds discarding the least important ones. Also, this example will be used to verify the topometry optimization results of the last design cycle obtained in Part 3. In addition, you will learn how to make and use smart assemblies.

If you do not have the `TMDSG008_3.dat` file generated in part 2, copy the file `TMDSG008_3_ref.dat` to `TMDSG008_3.dat`.

When you finish this example, you should have created a file named: `TMDSG008_4.dat.`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG008_3.dat`

Optimize the Structure

3. From the main menu bar, select **Genesis** → **Optimize**

Import the Post-Processing Files (Design Property data)

If you the results from part 2 you can import those post-processing files.

4. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
5. Select the `TMDSG008_3_dsgOPOSTxx.pch` file where `xx` is the last design cycle number
6. Push the **Open** button

Post-Processing the Results (Design Property data)

7. Push the **Deform Mesh/Color Mesh** button
8. Select the PBUSH K1 value for the first design cycle
9. Select the PBUSH K1 value for the last design cycle
10. Push the **Options...** button.
11. Select the **Hide Elements with No Value** checkbox
12. Slide the **Lower Cutoff** slidebar to mask out elements with low values (50000 or less)

13. Push the **Close** button

Create New PBUSH Group

14. Select the **Analysis** tab
15. From the **Analysis** Category chooser, select the **Group Properties**
16. Push **New Group Property** button from the Edit menu toolbar
17. Enter `Important Welds` for the **Name**
18. For Type, select **PBUSH**
19. Select Red Color
20. Push **Next>**
21. Enter `100000.0` for **K1**, **K2** and **K3**
22. Enter `1000000.0` for **K4**, **K5** and **K6**
23. Push the **Finish** button

Moving the Important PBUSH Elements to the Group

24. From the **Analysis** category chooser, select **Elements**
25. Make sure that there are no selected elements. For that, check the bottom of the Design Studio main window. If there are elements selected; push the **Select None** button
26. Push the **Select All** button to select the elements on the Viewport

    After you select them, the **Elements** Panel should indicate that about 10 elements have been selected.
27. Push the **Modify Elements** button from the Edit menu toolbar
28. Accept the default option, **Change Element's Group**
29. Push **Next>**
30. Select the new PBUSH (`Important Welds`) created earlier
31. Push the **Finish** button

Verify the Moving of the Elements

32. Select the **Post** tab
33. Push the **Up** button
34. Select the **Display** tab

35. Push the **Show/Hide Groups** button
36. Hide all the groups except the two PBUSH groups
37. Push the **Up** button
38. For the **Free Grids** chooser, select **Hide** radio button

    Note that 10 of the PBUSH elements are colored in red which is the color of the group with important welds

Delete the Existing Topometry Data

39. Select the **Design** tab
40. From the category chooser, select **Topometry**
41. Select **PBUSH 801**
42. Push the **Delete Topometry Design**

Delete the Existing Sizing Data

43. From the category chooser, select **Sizing**
44. Select **PBUSH 801**
45. Push the **Delete Sizing Design**

Delete the Existing Design Variable

46. From the category chooser, select **Design Variables**
47. From the main menu bar, select **Edit** → **Select All**
48. From the Edit menu toolbar, select **Delete Design Variable** button

Delete the DSELECT Synthetic Constraint

49. From the category chooser, select **Synthetic Response**
50. Select the existing objective
51. From the Edit menu toolbar, select **Delete Synthetic Response** button

Delete the Objective

52. From the category chooser, select **Objectives**
53. Select the existing objective

54. From the Edit menu toolbar, select **Delete Objective** button

Delete the Least Important Welds

Now you will delete the 12 CBUSH elements that are least important.

55. Select the **Display** tab
56. Push the **Show/Hide Groups** button
57. Hide all groups except PBUSH 801 group
58. Select the **Analysis** tab
59. From the category chooser, select **Elements**
60. Push the **Select All** button to select all the visible bush elements
61. From the Edit menu toolbar, select **Delete Elements** button
62. Select the **Display** tab
63. Push the **Show/Hide Groups** button
64. Push the **Show All** button
65. Push the **Up** button
66. Push the **Manage Groups** button
67. Select PBUSH 801
68. From the Edit menu toolbar, select **Delete Group** button
69. Push the **Up** button

Save the Design Studio Database File

70. From the main menu bar, select **File** → **Save As...**
71. Enter `TMDSG008_4` as the Filename and push **Save (**as a Design Studio File**)**

Export the Input File

72. From the main menu bar, select **File** → **Export** → **Input Data...**
73. Enter `TMDSG008_4.dat`
74. Push the **Save** button

Analyze the Structure

75. From the main menu bar, select **Genesis** → **Single Analysis**

Import the Post-Processing Files (Modes of All Design Cycles)

76. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
77. Select the `TMDSG008_4_dsg00.pch` file
78. Push the **Open** button

Post-Processing the Results (Mode Shape of All Design Cycles)

79. Select the **Post** tab
80. Push the **Deform Mesh/Color Mesh** button
81. Push the **Filled Contours** Button
82. Verify that the 10th mode is now the first torsional mode

    Due to the reduced number of welds, 2 additional modes appeared.
83. Animate the different modes
84. Push the **Up** button

Quit Design Studio

85. From the main menu bar, select **File** → **Quit**

5.9 Mode Shape Optimization

The purpose of this example is to learn how to optimize mode shape components and use them as matching objectives. You will learn how to create AVI files that show how the eigenvector changes.

In this problem you want the eigenvector nodes to move toward the ends of the structure. For that purpose, 4 grid points have been selected to be matched to zero.

The following objective and constraint will be created:

Minimize U1**2 + U2**2 + U3**2 + U4**2

where, U1, U2, U3 and U4 are components of mode shape 7 (first elastic mode shape)

Subject to:

$8.6E-5 \leq \text{Mass} \leq 9.0E-5$

Example ID

`TMDSG009`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
TMDSG009.dat	Input data	Provided: Contains the finite element mesh with loading conditions.
TMDSG009_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `TMDSG009.dat`.
TMDSG009_dsgOPOSTxy.pch	punch file	Multiple files generated using Genesis within Design Studio. These files contain the thicknesses for all shell elements for post-processing all design cycles.
TMDSG009_dsgSSOLxx.dat	Input data	Generated using Genesis within Design Studio. This file contains solid elements that represent the shell elements. This file is for visualization purpose and not intended for analysis. This file is for the last design cycle xx.
TMDSG009_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to `TMDSG009_dsg.dat`. This file is provided to check your example.

Start Design Studio

1. Load the Genesis data file: `TMDSG009.dat`

Set up Design Variables

2. Select the **Design** tab
3. From the **Design** category chooser, select **Design Variables**
4. Push **New Design Variable** button from the Edit menu toolbar
5. Enter T1 in the name and accept the **Independent Design Variable** type
6. Push **Next>**
7. Enter `1.0` for **Initial Value**, `0.1` for **Lower Bound**, and `2.0` for **Upper Bound**
8. Push **Finish**
9. Push **New Design Variable** button again, to create a second design variable
10. Enter T2 in the name and accept the **Independent Design Variable** type
11. Push **Next>**
12. Enter `1.0` for **Initial Value**, `0.1` for **Lower Bound**, and `2.0` for **Upper Bound**
13. Push **Finish**
14. Push **New Design Variable** button for the third time, to create a third design variable
15. Enter T3 in the name and accept the **Independent Design Variable** type
16. Push **Next>**
17. Enter `1.0` for **Initial Value**, `0.1` for **Lower Bound**, and `2.0` for **Upper Bound**
18. Push **Finish**

Set up Sizing Optimization Data

19. From the **Design** category chooser, select **Sizing**
20. Select PSHELL 27
21. Push the **Modify Sizing Design** button from the Edit menu toolbar to add sizing design attributes
22. Select `T1` in the **Thickness** menu
23. Push **Finish**
24. Select PSHELL 28
25. Push the **Modify Sizing Design** button from the Edit menu toolbar to add sizing design attributes
26. Select `T2` in the **Thickness** menu

27. Push **Finish**
28. Select PSHELL 30
29. Push the **Modify Sizing Design** button from the Edit menu toolbar to add sizing design attributes
30. Select `T3` in the **Thickness** menu
31. Push **Finish**

    Verify that the hammer icon is now next to each of the PSHELL labels. Here, the hammer means that the group is being size designed.

Set up Topometry Optimization Data

32. From the **Design** category chooser, select **Topometry**
33. Select PSHELL 27, 28 and 30
34. Push the **Modify Topometry Design** button from the Edit menu toolbar
35. Push the **Finish** button

    Verify that the hammer icon is now next to each of the PSHELL labels. Here, the hammer means that the group is being topometry designed.

Defining a Matching Objective

36. From the **Design** category chooser, select **Objectives**
37. Push **New Objective** button from the Edit menu toolbar
38. Select the **More Response Types...**
39. Select the **Target** button
40. Enter `0.0` for Target and `1.0` for Weight
41. Push **Next>**
42. Select the **Mode Shape Number** response
43. Enter `7` in the box for **Mode Shape Number**
44. Push **Next>**

    Verify that there are no grids selected. If there are grids selected, push the **Select None** button.

45. Enter `13229` on the **Select by Grid ID** box and accept **Translation 1**
46. Push **Add**

    Verify that there is 1 grid selected.

47. Push **Next>**

48. Select the existing loadcase
49. Push **Finish**
50. Define 3 more objectives using same information as above but for grids use: `13247`, `14201` and `14219`.

    In each case, do not forget to verify that one and only one grid is selected.
51. Verify that now there are 4 responses in the objectives list

Defining the Constraints

52. From the **Design** category chooser, select **Constraints**
53. Push **New Constraint** button from the Edit menu toolbar
54. Select the **Mass** response and accept the default **Entire Model**
55. Enter `8.6E-5` as **Lower Bound**
56. Enter `9.0E-5` as **Upper Bound**
57. Push the **Finish** button

    Verify that there is one response in the constraints list.

Request the Element Sizing Data File to be Output

58. From the main menu bar, select **Genesis** → **Options...**
59. Select the **File Control** tab
60. For **Element Sizing File**, make sure that the **Create** option is selected
61. Push the **Apply** button

Request the Mode Shape to be Output

62. From the **Analysis** category chooser, select **Loadcases**
63. Select Loadcase 6003
64. Push the **Modify Loadcase** button from the Edit menu toolbar
65. Push **Next>**
66. Push **Next>**
67. Push **Next>**
68. For displacement: Make sure that the **Post** and **All** options are selected
69. Push the **Finish** button

Optimize the Structure Using Genesis

70. From the main menu bar, select **Genesis** → **Optimize**
71. Study the **Design History**; when done, push the **Close** button.
72. Study the **Genesis Console Output** window.

Import the Post-Processing Files

73. From the **Genesis Console Output** window, select **Import Post..** button
74. Select all the post processing files to import by using the **Shift** key

    `TMDSG009_dsgxx.pch` files are the file with finite element analysis results for each design cycle `xx`
    `TMDSG009_dsgOPOST00.pch` files are the file with finite element analysis results for each design cycle `xx`

75. Push the **Import** button
76. From the **Genesis Console Output** window, select **Close** button

Post-Processing the Results (Thickness Distribution)

77. Select the **Post** tab
78. Push the **Deform Mesh/Color Mesh** button
79. Select a Thickness result for any design cycle
80. Study the results
81. Push the **Up** button

Post-Processing the Results (Mode Shape)

82. Push the **Deform Mesh/Color Mesh** button
83. Push the **Filled Contours** radio button
84. Select Mode 7 result for design cycle 0
85. To animate, push the **Oscillate** radio button
86. Select Mode 7 result for the last design cycle

    Compare the two modes. Verify that the mode shape nodes (points that do not move) have moved outwards.

87. Push the **Up** button

Post-Processing the Results (Create an gif File That Shows Mode Shape Improvements)

88. Push the **Animation** button
89. For the **Deform Result Type,** select **Displacement/Eigenvector**
90. For the **Color Result Type,** select **Displacement/Eigenvector**
91. Push **Next>**
92. For the **Choose Deform Mesh Frames,** select the Mode 7 result for each design

    Press the control key while selecting list items to select multiple discontinuous items.
93. Push **Next>**
94. For the **Choose Color Mesh Frames**, also select the Mode 7 result for each design

    Study the animations. Study how the nodes move outward.
95. Push **Next>**
96. Push the **Save Animation** button

    The **Save Animation Frames...** button can be used to save each frame of the animation into separate image files.
97. In the file selection window, enter a file name and push the **Save** button.

    An avi file is created. Avi files can be used to show the animation outside of Design Studio. E.g., you can put the avi file inside a PowerPoint presentation.
98. Push the **Finish** button

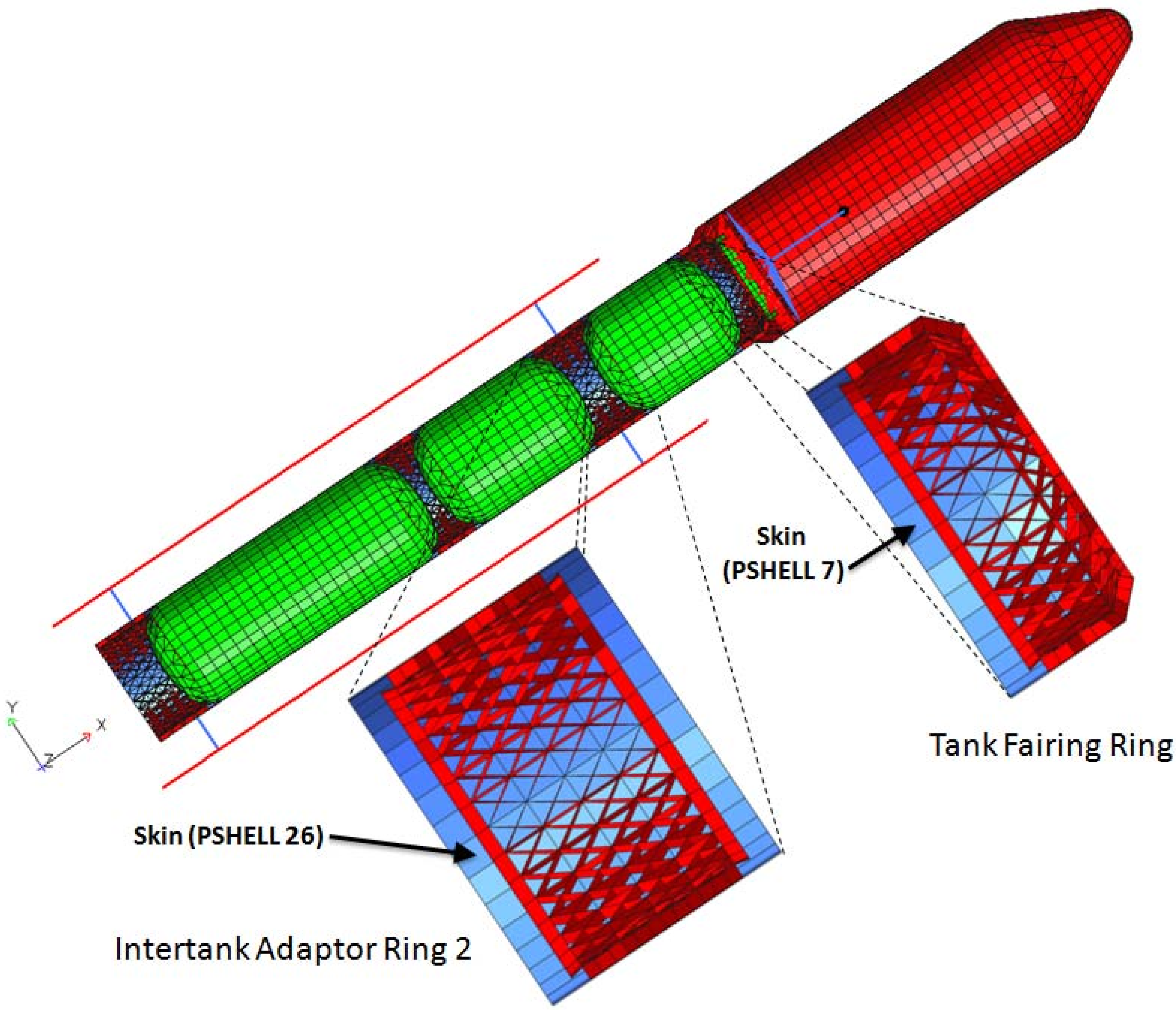
Quit Design Studio

99. From the main menu bar, select **File** → **Quit**

5.10 Optimization of Launch Vehicle to Maximize Frequency

Introduction

The purpose of this example is to learn how to maximize a specific modal frequency without increasing the structural weight. In this example we will use a finite element model of an aerospace launch vehicle (LV) with it's payload (PL) simulated as a CG mass connected at the LV/PL interface via rigid elements. This example also goes through the process of how to set up and execute a normal modes analysis.



Only the skins of the Tank Fairing Ring, and the Intertank Adaptor Ring 2 will be modified. Shell element groups (consisting of 4 elements per group) will be allowed to decrease or increase to one half or twice their initial thickness. The rib supports on the rings will not be modified.

In this example, you learn how to perform topometry sizing.The following optimization problem will be created, solved and post-processed:

Maximize the frequency of mode 2

Subject to:

A design constraint that does not allow an increase of the total structural volume (VOL_{tot} = 2.126514E6 as printed in file `TMDSG008_1_dsg.out`)

This design constraint is defined this way so that the skin mass of the two ring structures may be re-distributed, but not increased in order to maximize the frequency of mode 2.

Designable region:

The shell element thickness for PSHELL 7, and PSHELL 26.

Example ID

`TMDSG010`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`TMDSG010.dat`	`Input data`	Provided: Contains the finite element mesh of the launch vehicle and rigid payload
`TMDSG010_ref.dat`	`Input data`	Provided as a back up: This file contains all the data generated in this example plus the data in `TMDSG010.dat`
`TMDSG010.dsg`	`DSG file`	Generated by Design Studio to run Design Studio. This file contains all the data generated in this example plus the data in `TMDSG010.dat`
`TMDSG010_dsg.out`	`output file`	Generated using Genesis within Design Studio. This file contains FE post-processing results for analysis.
`TMDSG010_dsgOPOSTxy.pch`	`punch file`	Generated using Genesis within Design Studio. These files contain the sizing data of the optimization for post-processing all design cycles.

`TMDSG010_dsgxy.pch`	`punch file`	Generated using Genesis within Design Studio. These files contain the analysis results (mode shapes) for post-processing all design cycles.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG010.dat`

Create the Eigenvalue Method

3. From the Analysis category chooser, select **Eigenvalue Methods**
4. Push the **New Eigenvalue Method** button
5. For the **Name**, enter `Lanczos`
6. For **Method**, select the **Lanczos** radio button
7. For **V1**, enter `1.0`

   For this unrestrained model using 1.0 Hz for the lower cutoff frequency eliminates the rigid body modes from the analysis output. Mode 1 will thus be the first elastic mode.
8. For **V2**, enter `32.0`
9. Push the **Finish** button

Create the Normal Loads Loadcase

10. From the Analysis category chooser, select **Loadcases**
11. Select `Loadcase 1`
12. Push the **Delete Loadcase** button
13. Push the **New Loadcase** button
14. For the **Name**, enter `Unrestrained Normal Modes`
15. Push the **Normal Modes** radio button
16. Push the **Next>** button
17. Push the **Next>** button
18. From the **Eigenvalue method** category chooser, select `Lanczos`
19. Push **Next>**
20. From the Displacement: category chooser, select `Post`

    Verify that All appears in the 2nd category chooser.
21. Push the **Finish** button

Analyze the structure using Genesis

22. From the main menu bar, select **Genesis** → **Single Analysis**
23. Study the **Genesis Console Output**
24. From the **Genesis Console Output** window, push the **Import Post...** button
25. Select the punched output file and push the **Import** button
26. When done, push the **Close** button

Post-Processing the Results

27. Select the **Post** tab
28. Push the **Deform Mesh/Color Mesh** button
29. Select **Mode 1** and push the **Oscillate** radio button

    Mode 1 is bending in the X-Y plane. This is not the mode that would excite the payload local mode.

30. Select **Mode 2**

    Mode 2 is bending in the X-Z plane. This is the mode that would excite the payload local mode if an elastic payload model was coupled to the LV.

    The calculated system mode is 15.77 hz. Since the local elastic payload mode has a frequency of 15.75 Hz. in the X-Z plane it is highly likely that two modes would couple together. It is desirable to try to separate the modal frequencies of these two modes.

31. Examine the other modes by selecting them
32. Push the **Up** button

Create the Sizing Design

33. From the **Design** category chooser, select **Quick Setup Trails**
34. Push the **Quick Sizing Setup** button
35. Select **PSHELL 7** and **PSHELL 26**

    Hold the <Ctrl> button on the keyboard to select multiple items

36. Push the **Next>** button
37. For the Lower Bound change **M2** to `0.5`
38. For the Upper Bound change **M3** to `2.0`

39. Push the **Finish** button

    Notice at the bottom of the page that 2 Design Variables and 2 Design Regions have been defined.

Define the Topometry Sizing Regions

40. From the **Design** category chooser, select **Topometry**
41. Select both **PSHELL 7** and **PSHELL 26** groups
42. Push the **Modify Topometry Design** button
43. For Coarse Method, select **Max elements per design variable** from the pull down menu
44. For Coarse Parameter, enter `4`
45. For Symmetry Coord. Sys, push the **Change** button
46. Select the **_LV(Cyl)** coordinate system
47. Push the **Next>** button
48. For Symmetry 1:, select **MYZ: Mirror bout YZ plane** from the pull down menu
49. For Symmetry 2:, select **MZX: Mirror bout XZ plane** from the pull down menu
50. Push the **Finish** button

Defining the Volume constraint

51. From the Design category chooser, select **Constraints**
52. Push the **New Constraint** button
53. Enter `Volume Constraint` for the Name
54. Push the **Volume** radio button
55. For Lower Bound, enter `2.1265E6`
56. For Upper Bound, enter `2.1265E6`
57. Push the **Finish** button

Defining the Design Objective

58. From the Design category chooser, select **Objectives**
59. Push the **New Objective** button
60. Enter `Frequency Mode 2` for the Name

61. For Response Type, push the **Frequency Mode Number** radio button
62. Enter `2` for the mode number
63. For Objective Definition Switch, push the **Max** radio button
64. Push the **Next>** button
65. Select the `Unrestrained Normal Modes` Loadcase
66. Push the **Finish** button

    Verify that the objective you just created is listed

Turn On Mode Tracking

67. From the Analysis category chooser, select **Loadcases**
68. Select the existing loadcase
69. Push the **Modify Loadcase** button
70. Push the **Next>** button
71. Push the **Next>** button
72. For Mode Tracking:, select **Yes** from the pull down menu
73. Push the **Finish** button

Setup the Genesis Options

74. From the main menu bar, select **Genesis** → **Options...**
75. In the Output Control tab, select **Print Module Times: Both**
76. In the Output Control tab, select **Analysis Output: First & Last**
77. In the Output Control tab, select **Design Output: First & Last**
78. Push the **Apply** button

Save the Design Studio database

79. From the main menu bar, select **File** → **Save**

Optimize the structure using Genesis

80. From the main menu bar, select **Genesis** → **Optimize**
81. Study the **Genesis Console Output**
82. From the **Genesis Console Output** window push the **Import Post...** button

83. Select all the punched output files and push the **Import** button
84. When done, push the **Close** button

Post-Processing the Results

85. Select the **Display** tab
86. Push the **Show / Hide Groups** button
87. Only display PSHELL groups 7 and 26

    Zoom in on this part of the structure.

88. Push the **Up** button
89. Select the **Post** tab
90. Push the **Deform Mesh/Color Mesh** button
91. Push the **Oscillate** radio button, and examine the mode shapes

    Notice that mode 2 has increased in frequency from 15.77 Hz. for Cycle 0, to 16.55 Hz. for the last Design Cycle.

92. For the Color Mesh, select the first, then the last design cycle `Thickness`

    The Cycle 0 Thickness shows that the initial thickness for the Tank Fairing Ring skin is 0.75 inches, and the initial thickness for the other ring skin is 1.25 inches. With the final thickness color mesh displayed, cycle through the drop down menu at the bottom center for Thickness, Thickness Change, and Thickness/Original. Notice the re-distributed element thickness profile that increases the natural frequency of mode 2. Check the thickness distribution for symmetry.

The topometry optimization has designed the shell element skin of the two rings to modify their thickness from 1/2 to 2.0 times their original thickness in order to maximize the frequency of mode 2.

A system volume constraint (which acts like a system mass constraint) was also imposed to insure no extra weight would be added to the design. The constraint response can be close to the upper or lower bounds within a small threshold value. The volume constraint for this problem is satisfied, but considered “Active” by it’s Bound Status in the table below that was taken from the Genesis output file.

```
                    *********************************************
                    *  D E S I G N   C Y C L E      5 (DESIGN)  *
                    *********************************************
CONSTRAINT

                       MASS,VOLUME
 ----------------------------------------------------------------------------
            PROP/MAT                          RESPONSE    BOUND    BOUND      BOUND
  TYPE          ID     TYPE                   VALUE       TYPE     VALUE      STATUS
 ----------------------------------------------------------------------------
  VOLUME            0   SYS              2.120122E+06   LB    2.126500E+06    A
  VOLUME            0   SYS              2.120122E+06   UB    2.126500E+06    A
```

It is interesting to note that the system volume constraint was driven to the active lower bound rather than the upper bound. This can happen when maximizing a modal frequency by designing parts of the structure that contributes to the generalized mass of the target mode (i.e. less mass can help to increase the frequency).

Quit Design Studio

93. From the main menu bar, select **File** → **Quit**
94. Push the **Don't Save** button

5.11 Re-grouping based on Topometry Optimization Results

Introduction

Topometry optimization results can give good ideas on where to reinforce structures or where to take material out of them. Sometimes these results require additional steps to minimize the number of components which have different dimensions, in order to manufacture them easier or just to make them simpler. In this example, you will learn how to “re-group” elements to achieve that.

This example demonstrates how to re-group elements and create a finite element mesh that represents the simplified topometry results. The provided data file contains all the optimization data needed.A plate structure is used as the analysis model. Two corners of the plate are constrained and five inplane point loads are applied in the middle of the opposite edge. The following optimization problem is solved.

Objective function of the problem:

Minimize the volume of the structure

Subject to:

Maximum Displacement $\leq$ 0.1

Designable region:

Every element in the structure

Example ID

TMDSG011

Files Used in This Problem

A list, of the files that are either provided to you or the ones you are expected to create during this example, is presented next. It is not necessary to study the list in detail at this point. The files listed will be introduced during the example. Later, this list can be used for verifying your results.

File Name	Type	Description
TMDSG011.dat	Input data	Provided: Contains analysis and topometry optimization data.

TMDSG011_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file should be identical to TMDSG011.dat.
TMDSG011_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
TMDSG011_dsgOPOSTxy.pch	punch file	Multiple files generated using Genesis within Design Studio. These files contain the thicknesses for all shell elements for post-processing all design cycles.
TMDSG011_new.dat	Input data	Created in this example. This file is created by exporting a modified version of the TMDSG011.dat file.The modification consists of deleting the less important elements and moving the remaining elements to three groups.
TMDSG011_ref.dat	Input data	Provided. Result input file. Should be similar to the file TMDSG011_new.dat. This file is provided in order to optionally verify your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TMDSG011.dat`

Study the Analysis Problem

3. From the main menu bar, select **Genesis** → **Model Summary**

   Verify that the model has the following characteristics.
   Number of grids: 2701
   Number of CQUAD4 elements: 2592

4. Push the **Close** button
5. Select the **Analysis** tab
6. From the category chooser, select **Loadcases**
7. Select the existing loadcase and study the viewport for the loading conditions

Clear the Selection

8. From the main-menu, select **Edit**→ **Deselect All**

Study the Design Problem

9. Select the **Design** tab
10. From the category chooser, select **Objectives**

    Verify that the objective is to minimize the volume of the structure.

11. From the category chooser, select **Constraints**

    Verify that there is a displacement constraint and its upper bound is 0.1

12. From the category chooser, select **Design Variables**

    Verify that there is one design variable with the following characteristics:

DESIGN VARIABLE Type	LABEL	Initial Value	Lower Bound	Upper Bound
Independent	THCK4	1.0	0.05	1.0

13. From the category chooser, select **Sizing**

    Verify that PSHELL 4 is designed. A hammer icon should be next to PSHELL.

14. Select PSHELL 4 from the property list

    All the elements in the group are highlighted.
    Notice the variable information that is designing the PSHELL appears in the status bar below the Design Studio main window.

15. From the category chooser, select **Topometry**

    Verify that PSHELL 4 is designed. A hammer icon should be next to PSHELL.

16. From the category chooser, select **Quick Setup Trails**

    Review the list in the panel.

Clear the Selection

17. Right-click the Viewport, select **Clear**→ **All**

Optimize the Structure Using Genesis

18. From the main menu bar, select **Genesis** → **Optimize**

    Wait until Genesis finishes the run.
    Note the initial and final objective value from the Genesis Console Output window.
    Reference Solution: Initial =162.0, Final = 91.35

Import the Post-Processing Files (Thickness Results)

19. From the **Genesis Console Output** window, select the **Import Post...** button

20. Select the `TMDSG011_dsgOPOSTxx.pch` (where `xx` represents the last design cycle) file

21. Push the **Import** button

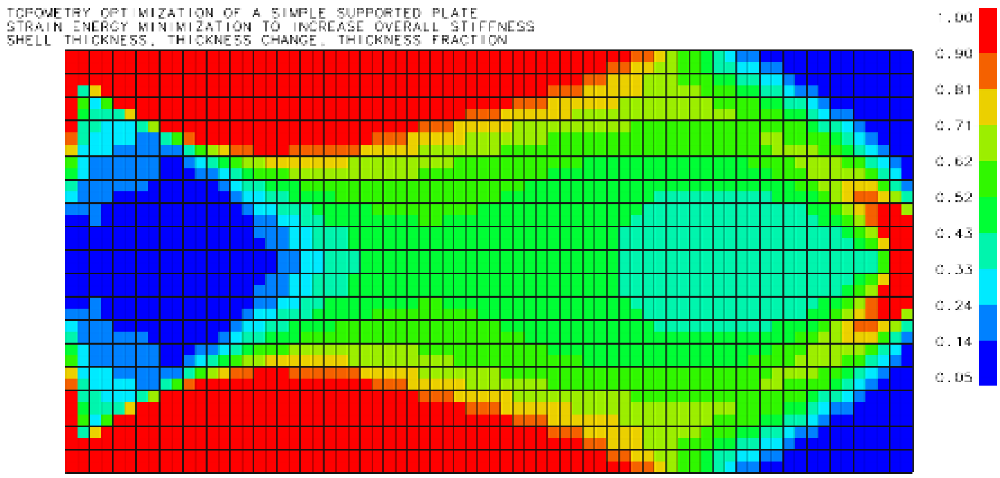
22. When done, push the **Close** button in the **Genesis Console Output** window

Change View

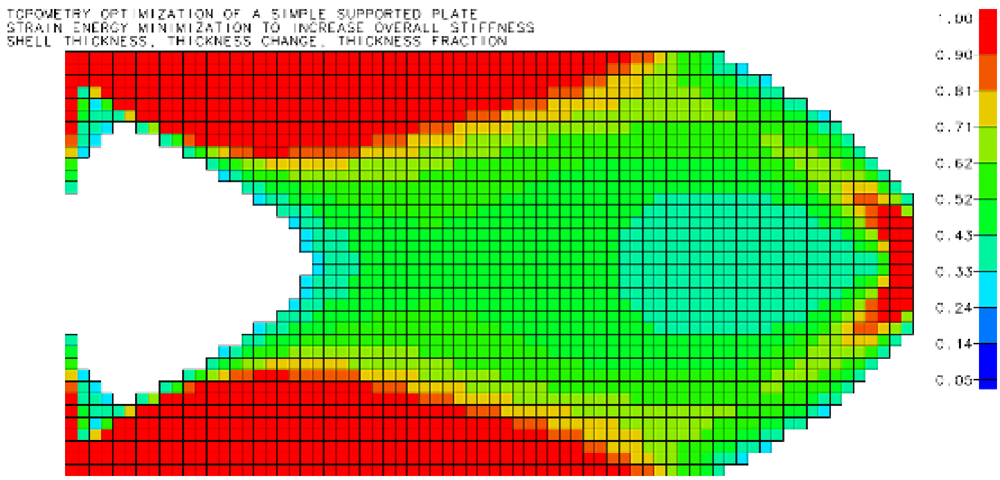
23. Push the **Top view** button to change view to the X-Y Plane

Post-Processing the Results (Thickness Results)

24. Select the **Post** tab
25. Push the **Deform Mesh/Color Mesh** button
26. Select the **Thickness** Result (for the last design cycle)



27. Push the **Options...** button. Slide the **Lower Cutoff** slidebar to mask out elements with low thickness values (0.3 or less)
28. Push the **Close** button



You will regroup this portion of the structure into two groups: one group with the elements on

the top and the bottom boundary and the other group with the elements inside
A thickness of 1.0 is assigned for the group along the top and bottom boundary and a value of 0.6 is assigned for the thickness of the group with the inside elements.
One can check the value of the thickness of the elements by clicking on an element in the Viewport. The thickness value is printed in the Design Studio Messages window

Create Two New Groups

29. Select the **Analysis** tab
30. From the category chooser, select **Group Properties**
31. Push **New Group Property** button from the Edit menu toolbar
32. Enter Name: `Top&Bottom`
33. For Type, select **PSHELL**
34. Select Blue Color
35. Push the **Next>** button
36. Enter `1.0` for thickness
37. Push the **Finish** button
38. Push **New Group Property** button from the Edit menu toolbar
39. Enter Name: `Inside`
40. For Type, select **PSHELL**
41. Select Yellow Color
42. Push the **Next>** button
43. Enter `0.6` for thickness
44. Push the **Finish** button

Moving the Elements to the First Group

45. From the **Analysis** category chooser, select **Elements**

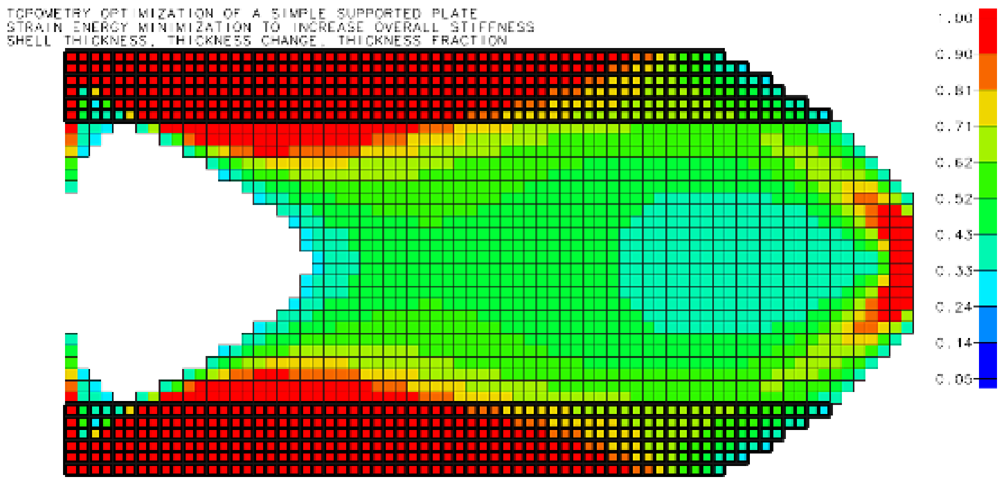
    Make sure that there are no selected elements. For that, check the bottom of the Design Studio main window. If there are elements selected; push the **Select None** button

46. Select the elements to move as shown in the figure below. Select the top and bottom six rows of elements

    The **Elements** Panel should indicate that about 726 elements have been selected. This number might vary in your case, as this number depends on how many elements you previously masked out.

47. Push the **Modify Elements** button from the Edit menu toolbar

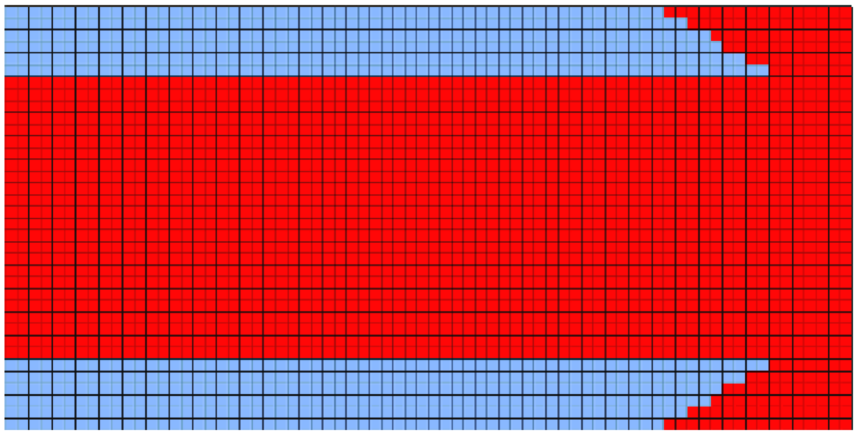
48. Accept the default option, **Change Element's Group**
49. Push **Next>**
50. Select **PSHELL 5,** `Top&Bottom`
51. Push the **Finish** button



Verify the Moving of the Elements to the First Group and Hide them

52. Select the **Post** tab
53. Push the **Up** button

    The elements in the first group should be blue.



54. Select the **Display** tab
55. Push the **Up** button
56. Push the **Show/Hide Groups** button

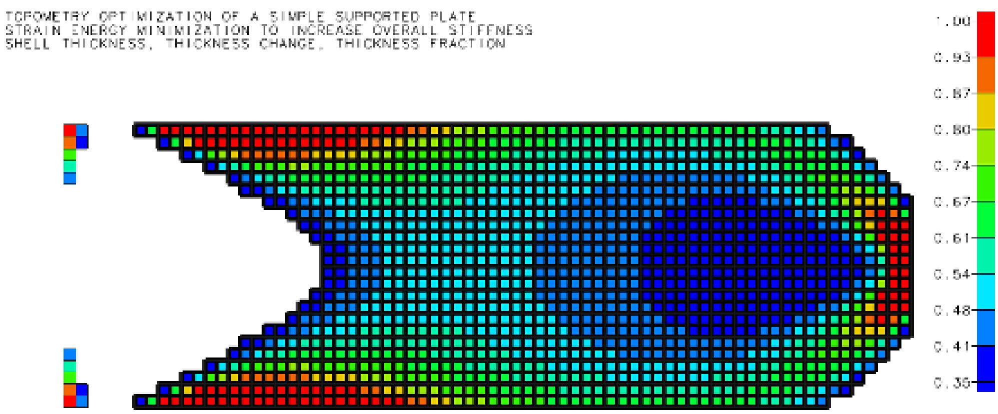
57. Hide the `Boundary` group

Moving the Elements to the Second Group

58. Select the **Post** tab
59. Push the **Deform Mesh/Color Mesh** button
60. Select the Thickness Result (for the last design cycle)
61. Select the **Analysis** tab
62. From the category chooser, select **Elements**
63. Select the elements to move as shown in the figure below

    The **Elements** Panel should indicate that about 1312 elements have been selected. This number might vary in your case, as this number depends on how many elements you previously masked out.

64. Push the **Modify Elements** button from the Edit menu toolbar
65. Accept the default option, **Change Element's Group**
66. Push **Next>**
67. Select **PSHELL 5,** `Inside`
68. Push the **Finish** button



Verify the Moving of the Elements to the Second Group and Hide them

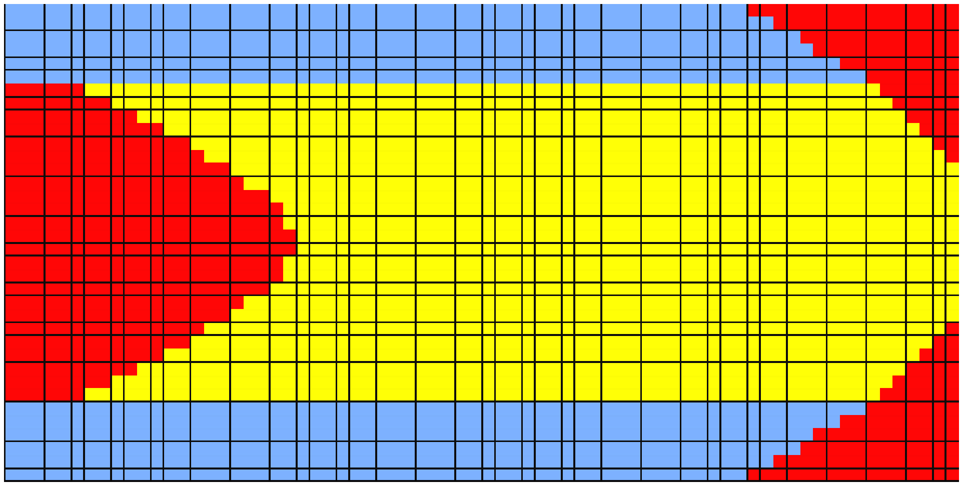
69. Select the **Post** tab
70. Push the **Up** button

    The elements in the second group should be yellow.

71. Select the **Display** tab

72. Push the **Show All** button to show all the groups

The model should look like the figure below.



Delete the Existing Topometry Data

73. Select the **Design** tab
74. From the category chooser, select **Topometry**
75. Select **PSHELL 4**
76. Push the **Delete Topometry Design**

Delete the Existing Sizing Data

77. From the category chooser, select **Sizing**
78. Select **PSHELL 4**
79. Push the **Delete Sizing Design**

Delete the Existing Design Variable

80. From the category chooser, select **Design Variables**
81. From the main menu bar, select **Edit** → **Select All**
82. From the Edit menu toolbar, select **Delete Design Variable** button

Delete the Constraints

83. From the category chooser, select **Constraints**

84. Select the existing constraint
85. From the Edit menu toolbar, select **Delete Constraint** button

Delete the Objective

86. From the category chooser, select **Objectives**
87. Select the existing objective
88. From the Edit menu toolbar, select **Delete Objective** button

Delete the Thin Elements

Now you should see the thin elements in red; these elements are not very important, so you will delete them.

89. Select the **Analysis** tab
90. From the category chooser, select **Elements**
91. From the main menu bar, select **Edit** → **Select All**
92. From the Edit menu toolbar, select **Delete Elements** button
93. From the category chooser, select **Grids**
94. From the main menu bar, select **Edit** → **Select All**
95. From the Edit menu toolbar, select **Delete Grids** button

Now the Design Studio Viewport should be with no elements or grids.

96. Select the **Display** tab
97. Push the **Show/Hide Groups** button
98. Push the **Show All** button

Now you should see the structure assembled with 2 groups.

Save the Design Studio Database File

99. From the main menu bar, select **File** → **Save As...**
100. Enter `TMDSG011_new` as the Filename
101. Push the **Save** button (as a Design Studio File)

Export the Input File

102. From the main menu bar, select **File** → **Export** → **Input Data...**
103. Enter `TMDSG011_new.dat`

104. Push the **Save** button

At this point, you have an answer that satisfies one of the original requirements: Design a structure with three different parts. This structure however, might or might not satisfy the volume constraint. To see if that is the case, you will analyze it.

Analyze the Structure

105. From the main menu bar, select **Genesis** → **Single Analysis**

After Genesis finishes the run, import and check the displacement.

Import the Analysis Results

106. From the **Genesis Console Output** window, select the **Import Post...** button
107. Select the `TMDSG011_new_dsg00.pch`
108. Push the **Import** button
109. When done, push the **Close** button in the **Genesis Console Output** window

Post-Processing the Displacements

110. Select the **Post** tab
111. Push the **Deform Mesh/Color Mesh** button
112. Select the **Filled Contours** radio button for a contour plot of displacement
113. Select the Displacement result of the Color Mesh
114. Right-Click on the viewport and select List Top Ten to view the top 10 displacements in the Design Studio Messages window

Note that the maximum displacements are close to the constraint value of 0.1. If needed, one can use other types of optimization to obtain a better design.

Quit Design Studio

115. From the main menu bar, select **File** → **Quit**
116. Push the **Don’t Save** button

CHAPTER 6

Shape Optimization Examples

- Design using 1-D (LINE) Domains
- Design using 2-D (QUAD) Domain - Uniform Variation
- Design using 2-D (QUAD) Domain - Linear Variation
- Design using 2-D (QUAD) Domain - Quadratic Variation
- Design using Multiple 2-D (QUAD) Domains
- Design using 3-D (HEXA) Domains
- Design using 3-D (HEXA) and 2-D (QUAD) Domains
- Design a Feature - Using 2-D (TRIA & QUAD) Domains
- Design a Feature - Using 2-D (QUAD) Domains
- Design of a Steering Knuckle - Review of Shape Domains
- Using Different Domains to Perform Similar Design
- Making Copies of Existing Domains - Wheel Example
- Design Location of Member - Using Synthetic Responses
- Design of a Solid Cantilever Beam using DOMAINs

- **Design of a Solid Cantilever Beam using Natural Perturbation Vectors**
- **Using Hexa Domains**
- **Using Multiple Quad Domains**
- **Using Axisymmetric Bar Domains**
- **Using Axisymmetric Quad and Axisymmetric Bar Domains**
- **Combining Shape and Sizing Optimization I**
- **Combining Shape and Sizing Optimization II - Use of Domains and Grid Perturbations**

6.1 Design using 1-D (LINE) Domains

Introduction

The purpose of this example is to learn how to create a basic shape optimization problem. A cantilevered I-beam is modeled with shell elements. The width and height of the I-Beam is designed. After you finish this example, you will know how to plot the shape changes.

In this problem, 1-dimensional domains are used to associate the grids for defining the shape changes. Two LINE domains are created - one along the width of the beam and another along the height of the beam. Perturbations are applied on these domains to design the cross-sectional dimensions of the beam.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

VonMises Stress $\leq$ 50.0

Designable region:

Overall Height and Width of the I-Beam

Example ID

`SHDSG001`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG001.dat`	`Input data`	Provided: Contains the finite element mesh of a cantilevered I-Beam modeled plate with applied load and boundary conditions.
`SHDSG001_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `SHDSG001.dat`
`SHDSG001_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SHDSG001_dsg.SHP`	`Shape Change data`	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization

SHDSG001_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SHDSG001_dsg.dat. This file is provided to check your example.

Start Design Studio

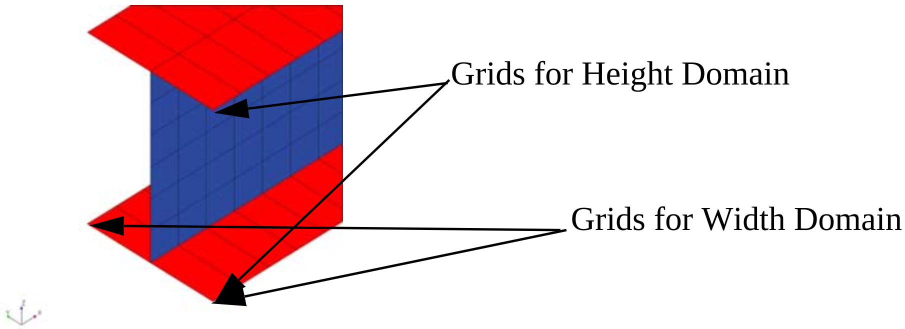
1. Start Design Studio
2. Import the Genesis data file: `SHDSG001.dat`

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase
5. View the loading and boundary conditions on the beam in the Viewport

Create Two 1-D Shape Domains

6. From the **Design** category chooser, select **Shape Domains**
7. Push the **New Domain** button from the Edit menu toolbar
8. Enter `Height` for the Name
9. Check the **New Domains Quick Setup** radio button
10. Push **Next**>
11. Select `Create New Domain Group` item
12. Push **Next>**
13. Select the **Point by Point** radio button
14. Select the **Lines** icon
15. Check the **Pick existing grids** radio button
16. Select two grids one on the top and another on the bottom surface of the I-beam as shown in the figure below
17. Push the **Finish** button
18. Push the **New Domain** button from the Edit menu toolbar
19. Enter `Width` for the Name
20. Check the **New Domains Quick Setup** radio button
21. Push **Next**>
22. Select `Create New Domain Group` item

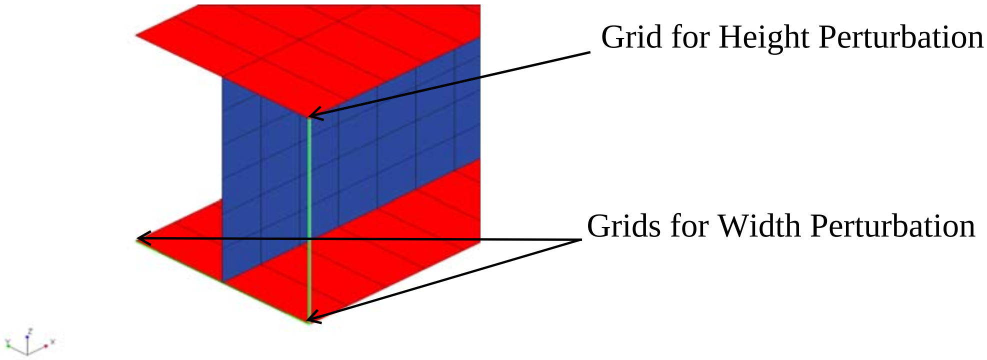


23. Push **Next>**
24. Check the **Point by Point** radio button
25. Select the **Lines** icon
26. Check the **Pick existing grids** radio button
27. Select two grids as shown in the figure
28. Push the **Finish** button

Create Perturbations associated with the Domains

29. From the **Design** category chooser, select **Shape Morphing Sets**
30. Push the **New Shape Set** button from the Edit menu toolbar
31. Enter `Height` in the Name field
32. Select the **Domain Morphing Set** radio button
33. Push **Next>**
34. Select `Height` from the **Select Domains to Act Upon** list
35. Push **Next>**
36. Push the **Select None** button
37. From the Viewport, select the grid (shown in the figure below) to apply the perturbation

Verify that there is 1 grid selected. The perturbation will be applied on the selected grid.



38. Enter `0.0`, `0.0`, `1.0` for the **X, Y, Z** to define the direction of the perturbation
39. Enter `90.0` for the **Magnitude**
40. Push the **Add Perturbation** button

    Verify that there is "**1 perturbations on 1 grids"** at the bottom of the window
41. Push the **Finish** button
42. Push the **New Shape Set** button from the Edit menu toolbar
43. Enter `Width` in the Name field
44. Select the **Domain Morphing Set** radio button
45. Push **Next>**
46. Select `Width` from the **Select Domains to Act Upon** list
47. Push **Next>**
48. Push the **Select None** button to clear any existing grid selections
49. From the Viewport, select one of the width grids (shown in the figure above) to apply the perturbation
50. Enter `0.0`, `1.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation
51. Enter `15.0` for the **Magnitude**
52. Push the **Add Perturbation** button

    Verify that there is "**1 perturbations on 1 grids"** at the bottom of the window
53. Push the **Select None** button to clear any existing grid selections
54. From the Viewport, select the other width grid (shown in the figure above)
55. Change the **Y** value to `-1.0` for the direction of perturbation
56. Enter `15.0` for the **Magnitude**

57. Push the **Add Perturbation** button

    Verify that there is "**2 perturbations on 2 grids"** at the bottom of the window

58. Push the **Finish** button

Preview the shape changes

59. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
60. Under the Deform Mesh, select a Shape Morphing Set Preview
61. Select the **Oscillate** radio button
62. Push the **Up** button

Define the Design Objective

63. From the **Design** category chooser, select **Objectives**
64. Push the **New Objective** button from the Edit menu toolbar
65. Enter `Mass` for the Name
66. Make sure the **Mass** radio button is selected
67. Push **Finish**

    Verify that now there is 1 response in the objectives list

Define the Design Constraint

68. From the **Design** category chooser, select **Constraints**
69. Push the **New Constraint** button from the Edit menu toolbar
70. Enter `Stress` for the Name
71. Make sure the **Stress** response is selected and accept the default **Selected Groups**
72. Enter `50.0` as **Upper Bound**
73. Push **Next>**
74. Push the **Select** button to select both the existing groups
75. Push **Next>**
76. Make sure the **Stress Components** are the **vonMises Top & Bottom**
77. Push **Next>**
78. Select the existing loadcase (`VERTICAL LOAD`)

79. Push **Finish**

    Note that there is 2 response in the constraints list. One constraint for vonMises on the top layer and one for the bottom layer.

Optimize the Structure Using Genesis

80. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button

Import the Shape Changes File

81. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
82. Select the `SHDSG001_dsg.SHP` file
83. Push the **Open** button

Post-Processing the Results (Shape Changes)

84. Select the **Post** tab
85. Push the **Deform/Mesh Color Mesh** button
86. Select a Shape Change for the last design cycle
87. Push the **Filled Contours** radio button
88. Select a Shape Change for the last design cycle in the **Color Mesh** frame
89. Push the **Up** button

Creating a Picture File

Select a good view of the results.

90. From the main menu bar, select **File** → **Print to Image File**
91. Enter `SHDSG001_dsg.png` for the name
92. Push the **Save** button

Quit Design Studio

93. From the main menu bar, select **File** → **Quit**
94. Push the **Don’t Save** button

6.2 Design using 2-D (QUAD) Domain - Uniform Variation

Introduction

The purpose of this example is to learn how to create a basic shape optimization problem. A cantilevered I-beam is modeled with shell elements. The width and height of the I-Beam are designed. After you finish this example, you will know how to animate the shape changes.

In this problem, a 2-dimensional domain is used to associate the grids for defining the shape changes. One QUAD domains is created to enclose all the grids of the beam. Perturbations are applied on this domain to design the cross-sectional dimensions of the beam.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

VonMises Stress $\leq$ 50.0

Designable region:

Overall Height of the I-Beam and Width of the I-Beam

Example ID:

`SHDSG002`

Special Features:

In this example, perturbations are applied such that an entire edge of the QUAD domain is perturbed uniformly. This uniform perturbation is achieved by applying two identical perturbations on both the ends of the edge in the QUAD domain.

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description

`SHDSG002.dat`	`Input data`	Provided: Contains the finite element mesh of a cantilevered I-Beam modeled plate with applied load and boundary conditions.
`SHDSG002_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `SHDSG002.dat`
`SHDSG002_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SHDSG002_dsg.SHP`	`Shape Change data`	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization
`SHDSG002_ref.dat`	`Input data`	Provided. Reference result input file. Should be nearly identical to `SHDSG002_dsg.dat`. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG002.dat`

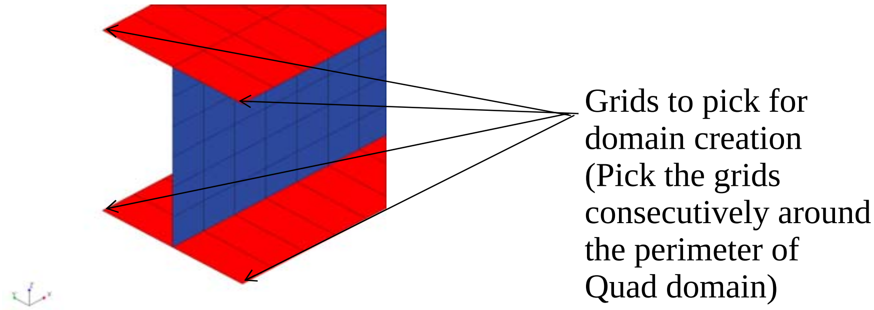
Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase
5. View the loading and boundary conditions on the beam in the Viewport

Create a 2-D (QUAD) Shape Domains

6. From the **Design** category chooser, select **Shape Domains**
7. Push the **New Domain** button from the Edit menu toolbar
8. Enter `Quad` for the Name
9. Check the **New Domains Quick Setup** radio button
10. Push **Next**>
11. Select `Create New Domain Group` item
12. Push **Next>**
13. Select the **Point by Point** radio button
14. Select the **Quads** icon
15. Select the **Pick existing grids** radio button

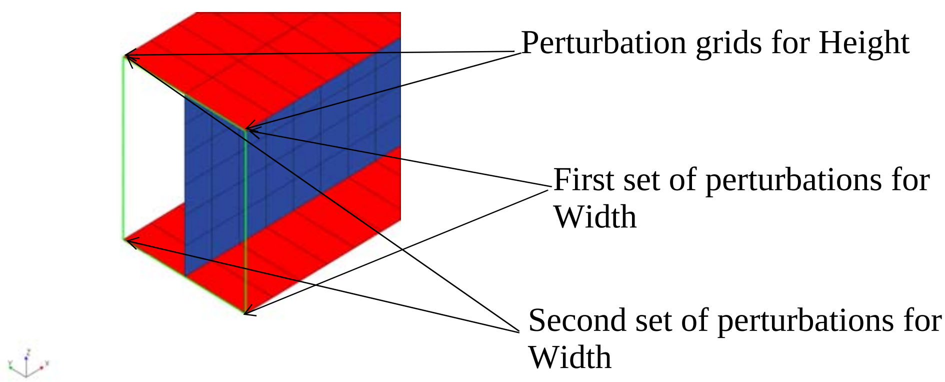
16. Select the four corner grids on the beam cross-section as shown in the figure below.



17. Push the **Finish** button

Create Perturbations associated with the Domains

18. From the **Design** category chooser, select **Shape Morphing Sets**
19. Push the **New Shape Set** button from the Edit menu toolbar
20. Enter `Height` in the Name field
21. Select the **Domain Morphing Set** radio button
22. Push **Next>**
23. Select `Quad` from the **Select Domains to Act Upon** list
24. Push **Next>**
25. Push the **Select None** button
26. From the Viewport, select the two corner grids on the top surface (shown in the figure below) to apply the perturbation



27. Enter `0.0`, `0.0`, `1.0` for the **X, Y, Z** to define the direction of the perturbation
28. Enter `90.0` for the **Magnitude**

29. Push the **Add Perturbation** button

    Verify that there is "**2 perturbations on 2 grids"** at the bottom of the window.

    By applying two identical perturbations on both the ends of the edge in the QUAD domain, the edge is perturbed uniformly.

30. Push the **Finish** button
31. Push the **New Shape Set** button from the Edit menu toolbar
32. Enter `Width` in the Name field
33. Select the **Domain Morphing Set** radio button
34. Push **Next>**
35. Select `Quad` from the **Select Domains to Act Upon** list
36. Push **Next>**
37. Push the **Select None** button
38. From the Viewport, select two grids (shown in the figure above) to apply the first set of perturbations
39. Enter `0.0`, `1.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation
40. Enter `15.0` for the **Magnitude**
41. Push the **Add Perturbation** button

    Verify that there is "**2 perturbations on 2 grids"** at the bottom of the window

42. Push the **Select None** button
43. From the Viewport, select the other set of grids (shown in the figure above)
44. Change the **Y** value to `-1.0` for the direction of perturbation
45. Enter `15.0` for the **Magnitude**
46. Push the **Add Perturbation** button

    Verify that there is "**4 perturbations on 4 grids"** at the bottom of the window

47. Push the **Finish** button

Preview the shape changes

48. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
49. Select the **Oscillate** radio button
50. Push the **Filled Contours** radio button
51. Under the **Color Mesh**, select a **Shape Morphing Set Preview**
52. Push the **Up** button

Define the Design Objective

53. From the **Design** category chooser, select **Objectives**
54. Push the **New Objective** button from the Edit menu toolbar
55. Enter `Mass` for the Name
56. Make sure the **Mass** radio button is selected
57. Push **Finish**

Define the Design Constraint

58. From the **Design** category chooser, select **Constraints**
59. Push the **New Constraint** button from the Edit menu toolbar
60. Enter `Stress` for the Name
61. Make sure the **Stress** response is selected and accept the default **Selected Groups**
62. Enter `50.0` as **Upper Bound**
63. Push **Next>**
64. Push the **Select** button to select both the existing groups
65. Push **Next>**
66. Make sure the **Stress Components** are the **vonMises Top & Bottom**
67. Push **Next>**
68. Select the existing loadcase (`VERTICAL LOAD`)
69. Push **Finish**

    Note that there is 2 response in the constraints list. One constraint for vonMises on the top layer and one for the bottom layer.

Optimize the Structure Using Genesis

70. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**.

Import the Shape Changes File

71. From the **Genesis Console Output** window, select the **Import Post..** button
72. Select the `SHDSG002_dsg.SHP` file

73. Push the **Import** button
74. From the **Genesis Console Output** window, select the **Close** button

Post-Processing the Results (Shape Changes)

75. Select the **Post** tab
76. Push the **Deform/Mesh Color Mesh** button
77. Select a Shape Change for the last design cycle
78. Push the **Filled Contours** radio button
79. Select a Shape Change for the last design cycle in the **Color Mesh** frame
80. Push the **Up** button

Creating an Animation

81. Select the **Post** tab
82. Push the **Animation** button
83. Select **Shape Change** for the **Deform Result Type**
84. Select **Shape Change** for the **Color Result Type**
85. Push **Next>**
86. Use the Shift key and select all the shape changes for the Deform Mesh Frames
87. Push **Next>**
88. Use the Shift key and select all the shape changes for the Color Mesh Frames
89. Push **Next>**
90. Push the **Save Animation...** button
91. Enter `SHDSG002.gif` for the name
92. Push the **Save** button

    By default, the animation will be saved in the working directory unless the user changes it.

Quit Design Studio

93. From the main menu bar, select **File** → **Quit**
94. Push the **Don't Save** button

6.3 Design using 2-D (QUAD) Domain - Linear Variation

Introduction

The purpose of this example is to learn how to create a basic shape optimization problem.This example also goes through the process of enforcing geometric (DRESPG) constraints. The analysis model is a cantilevered I-beam modeled with shell elements.

In this problem, a 2-dimensional domain is used to associate the grids for defining the shape changes. One QUAD domain is created along the length of the beam to enclose all the grids. Perturbations are applied on this domain to design the cross-sectional height and the taper along the length of the beam.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

VonMises Stress $\leq$ 10.0

Distance between the top and bottom surfaces at the end $\geq$ 20.0

Designable region:

Height of the I-Beam

Taper of the beam along its length

Example ID

`SHDSG003`

Special Features:

In this example, perturbations are applied such that some edges of the QUAD domain vary linearly. This linear variation is achieved by applying a perturbation to only one of the ends of the edge in the QUAD domain.

In this example, geometric (DRESPG) constraints are enforced so that the top and bottom surfaces of the I-beam don't intersect during the optimization process. Because the shape of the structure is modified during shape optimization, geometric constraints are handy as they provide the user with the flexibility to constrain the geometry.

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SHDSG003.dat	Input data	Provided: Contains the finite element mesh of a cantilevered I-Beam modeled plate with applied load and boundary conditions.
SHDSG003_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in SHDSG003.dat
SHDSG003_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
SHDSG003_dsg.SHP	Shape Change data	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization
SHDSG003_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SHDSG003_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: SHDSG003.dat

Review the Loadcase

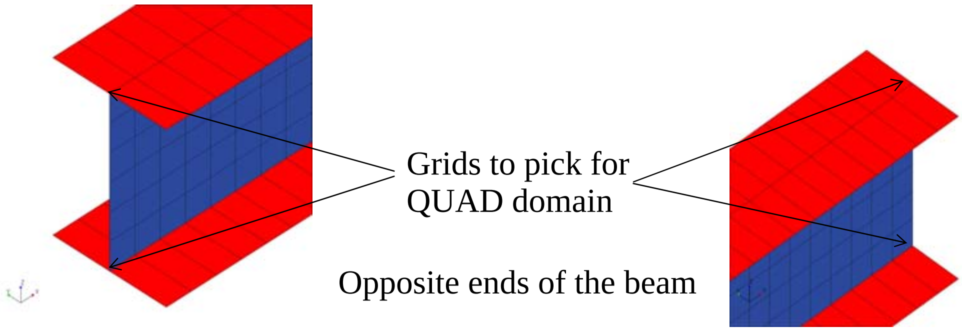
3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase
5. View the loading and boundary conditions on the beam in the Viewport

Create a 2-D (QUAD) Shape Domains

6. From the **Design** category chooser, select **Shape Domains**
7. Push the **New Domain** button from the Edit menu toolbar
8. Enter Quad for the Name
9. Check the **New Domains Quick Setup** radio button
10. Push **Next**>
11. Select Create New Domain Group item
12. Push **Next>**
13. Select the **Point by Point** radio button
14. Select the **Quads** icon

15. Select the **Pick existing grids** radio button
16. Select the four grids as shown in the figure below to create a QUAD domain along the length of the beam

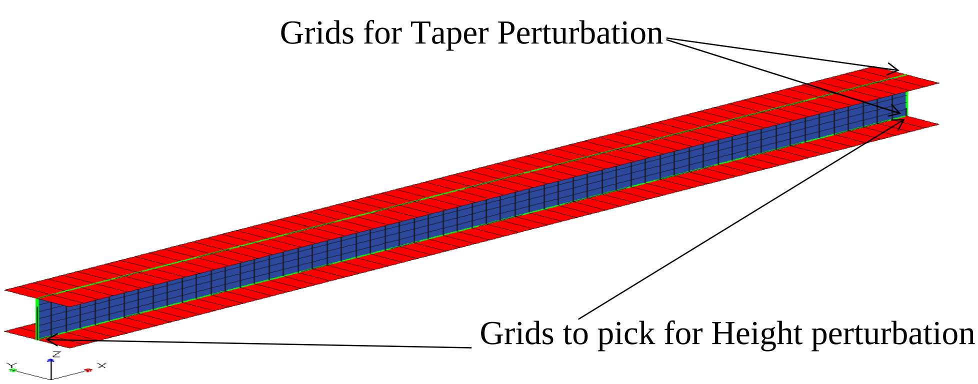
    Make sure to select the four grids in either the clockwise or the counter-clockwise direction.



17. Push the **Finish** button

Create Perturbations associated with the Domains

18. From the **Design** category chooser, select **Shape Morphing Sets**
19. Push the **New Shape Set** button from the Edit menu toolbar
20. Enter `Height` in the Name field
21. Select the **Domain Morphing Set** radio button
22. Push **Next>**
23. Select `Quad` from the **Select Domains to Act Upon** list
24. Push **Next>**
25. Push the **Select None** button
26. From the Viewport, select the two grids on the QUAD domain along the length of the beam as shown in the figure below to apply the perturbation



27. Enter `0.0`, `0.0`, `1.0` for the **X, Y, Z** to define the direction of the perturbation
28. Enter `40.0` for the **Magnitude**
29. Push the **Add Perturbation** button

    Verify that there is "**2 perturbations on 2 grids"** at the bottom of the window
30. Push the **Finish** button
31. Push the **New Shape Set** button from the Edit menu toolbar
32. Enter `Taper` in the Name field
33. Select the **Domain Morphing Set** radio button
34. Push **Next>**
35. Select `Quad` from the **Select Domains to Act Upon** list
36. Push **Next>**
37. Push the **Select None** button
38. From the Viewport, select one of the two grids (shown in the figure above for Taper perturbation) to apply the first perturbation
39. Enter `0.0`, `0.0`, `1.0` for the **X, Y, Z** to define the direction of the perturbation
40. Enter `30.0` for the **Magnitude**
41. Push the **Add Perturbation** button

    Verify that there is "**1 perturbations on 1 grids"** at the bottom of the window
42. Push the **Select None** button
43. From the Viewport, select the other grid for the Taper perturbation (shown in the figure above)
44. Change the **Z** value to `-1.0` for the direction of perturbation
45. Enter `30.0` for the **Magnitude**
46. Push the **Add Perturbation** button

    Verify that there is "**2 perturbations on 2 grids"** at the bottom of the window
47. Push the **Finish** button

Preview the shape changes

48. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
49. Select the **Oscillate** radio button
50. Under the **Deform Mesh**, select a **Shape Morphing Set Preview**

51. Push the **Up** button

Define the Design Objective

52. From the **Design** category chooser, select **Objectives**
53. Push the **New Objective** button from the Edit menu toolbar
54. Enter `Mass` for the Name
55. Make sure the **Mass** radio button is selected
56. Push **Finish**

Define the Stress Design Constraint

57. From the **Design** category chooser, select **Constraints**
58. Push the **New Constraint** button from the Edit menu toolbar
59. Enter `Stress` for the Name
60. Make sure the **Stress** response is selected and accept the default **Selected Groups**
61. Enter `10.0` as **Upper Bound**
62. Push **Next>**
63. Push the **Select** button to select both the existing groups
64. Push **Next>**
65. Make sure the **Stress Components** are the **vonMises Top & Bottom**
66. Push **Next>**
67. Select the existing loadcase (`VERTICAL LOAD`)
68. Push **Finish**

    Note that there is 2 response in the constraints list. One constraint for vonMises on the top layer and one for the bottom layer.

Define the Geometric (DRESPG) Design Constraint

69. From the **Design** category chooser, select **Constraints**
70. Push the **New Constraint** button from the Edit menu toolbar
71. Enter `End_Height` for the Name
72. Select the **More Response Types...** radio button
73. Enter `20.0` as **Lower Bound**

74. Push **Next>**
75. Make sure the **Geometric** radio button is selected
76. Push **Next>**
77. From the Viewport, select the two grids that are used for the `Taper` perturbation
78. Select the **Distz** radio button for the type of geometric response

    Distz is the absolute value of the difference between the z-coordinates of the two selected grids. More details about the different geometric responses can be obtained from the GENESIS Design Reference manual (volume2.pdf on unix based systems).

79. Push **Finish**

    Note that there is 3 response in the constraints list.

Optimize the Structure Using Genesis

80. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button

Import the Shape Changes File

81. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
82. Select the `SHDSG003_dsg.SHP` file
83. Push the **Open** button

Post-Processing the Results (Shape Changes)

84. Select the **Post** tab
85. Push the **Deform/Mesh Color Mesh** button
86. Select a Shape Change for the last design cycle
87. Push the **Filled Contours** radio button
88. Select a Shape Change for the last design cycle in the **Color Mesh** frame
89. Push the **Up** button

Import the Analysis results

90. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
91. Select the `SHDSG003_dsg00.pch` file and put a checkmark in the **Import Similar Results for All Design Cycles** checkbox

92. Push the **Open** button

Post-Processing the Results (Analysis Results)

93. Under Color Mesh, Select a Stress result for any cycle to view them in the Viewport
94. Push the **Filled Contours** radio button
95. Under Color Mesh, Select a Displacement result for any cycle
96. Push the **Up** button

Creating an Animation

97. Push the **Animation** button
98. Select **Shape Change** for the **Deform Result Type**
99. Select **Shell Stress/Strain** for the **Color Result Type**
100. Push **Next>**
101. Use the Shift key and select all the shape changes for the Deform Mesh Frames
102. Push **Next>**
103. Use the Shift key and select all the Shell Stress for the Color Mesh Frames
104. Push **Next>**
105. Push the **Save Animation...** button
106. Enter `SHDSG003.gif` for the name
107. Push the **Save** button

    By default, the animation will be saved in the working directory unless the user changes it.

Quit Design Studio

108. From the main menu bar, select **File** → **Quit**
109. Push the **Don’t Save** button

6.4 Design using 2-D (QUAD) Domain - Quadratic Variation

Introduction

The purpose of this example is to learn how to create a basic shape optimization data. The analysis model is a cantilevered I-beam modeled with shell elements.

In this problem, a 2-dimensional domain is used to associate the grids for defining the shape changes. One QUAD domain is created along the length of the beam. Perturbations are applied on this domain to design the height at the free end and the curvature along the length of the beam.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

VonMises Stress $\leq$ 10.0

Displacement at the end $\leq$ 0.2

Designable region:

Height of the I-Beam at the free end

Curvature of the beam along its length

Example ID

`SHDSG004`

Special Features:

In this example, mid-side perturbations are applied such that the shape is a quadratic variation along the edge of the QUAD domain. This quadratic variation is achieved by applying a perturbation to a grid close to the middle of the edge.

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG004.dat`	`Input data`	Provided: Contains the finite element mesh of a cantilevered I-Beam modeled plate with applied load and boundary conditions.
`SHDSG004_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `SHDSG004.dat`
`SHDSG004_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SHDSG004_dsg.SHP`	`Shape Change data`	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization
`SHDSG004_ref.dat`	`Input data`	Provided. Reference result input file. Should be nearly identical to `SHDSG004_dsg.dat`. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG004.dat`

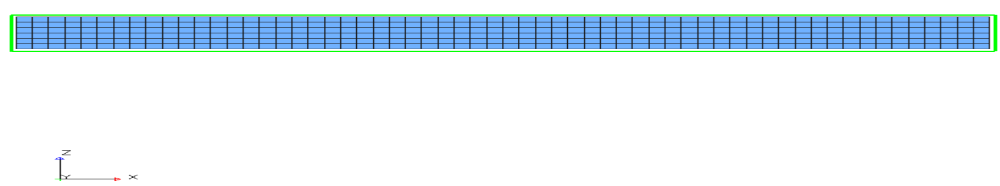
Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase
5. View the loading and boundary conditions on the beam in the Viewport

Create a 2-D (QUAD) Shape Domains

6. From the **Design** category chooser, select **Shape Domains**
7. Push the **New Domain** button from the Edit menu toolbar
8. Enter `Quad` for the Name
9. Check the **New Domains Quick Setup** radio button
10. Push **Next**>
11. Select `Create New Domain Group` item
12. Push **Next>**
13. Select the **Quads** icon

14. Select the **XZ** icon for **Pick points on workplane**
15. From the Viewport window, change the view to be the XZ plane (Left view)
16. Click-drag-click to create a rectangular domain on the Viewport such that the entire beam is enclosed within the domain as shown in the figure
17. Push the **Finish** button.



Create End Perturbations

18. From the **Design** category chooser, select **Shape Morphing Sets**
19. Push the **New Shape Set** button from the Edit menu toolbar
20. Enter `End_Height` in the Name field
21. Select the **Domain Morphing Set** radio button
22. Push **Next>**
23. Select `Quad` from the **Select Domains to Act Upon** list
24. Push **Next>**
25. Push the **Select None** button
26. From the Viewport, select one of the corners of the QUAD domain at the free end of the beam
27. Enter `0.0`, `0.0`, `1.0` for the **X, Y, Z** to define the direction of the perturbation
28. Enter `45.0` for the **Magnitude**
29. Push the **Add Perturbation** button

    Verify that there is “**1 perturbations on 1 grids”** at the bottom of the window
30. Push the **Select None** button
31. From the Viewport, select the other corner at the free end
32. Change the **Z** value to `-1.0` for the direction of perturbation

33. Push the **Add Perturbation** button

    Verify that there is "**2 perturbations on 2 grids"** at the bottom of the window

34. Push the **Finish** button

Create Mid-Side Perturbations

35. From the **Design** category chooser, select **Shape Morphing Sets**
36. Push the **New Shape Set** button from the Edit menu toolbar
37. Enter `Curvature` in the Name field
38. Select the **Domain Morphing Set** radio button
39. Push **Next>**
40. Select `Quad` from the **Select Domains to Act Upon** list
41. Push **Next>**
42. Push the **Select None** button
43. From the Viewport, select the a grid on the top surface near the middle of the beam
44. Enter `0.0`, `0.0`, `-1.0` for the **X, Y, Z** to define the direction of the perturbation
45. Enter `20.0` for the **Magnitude**
46. Push the **Add Perturbation** button

    Verify that there is "**1 perturbations on 1 grids"** at the bottom of the window

47. Push the **Select None** button
48. From the Viewport, select the grid on the bottom surface
49. Change the **Z** value to `1.0` for the direction of perturbation
50. Push the **Add Perturbation** button

    Verify that there is "**2 perturbations on 2 grids"** at the bottom of the window

51. Push the **Finish** button

Preview the shape changes

52. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
53. Select the **Oscillate** radio button
54. Under the **Deform Mesh**, select a **Shape Morphing Set Preview**

    Notice the preview of the shape change for DVAR 2. Since in this case perturbations are applied on the grids near the middle of a QUAD edge, the variation is quadratic.

55. Push the **Up** button

Define the Design Objective

56. From the **Design** category chooser, select **Objectives**
57. Push the **New Objective** button from the Edit menu toolbar
58. Enter `Mass` for the Name
59. Make sure the **Mass** radio button is selected
60. Push **Finish**

Define the Stress Design Constraint

61. From the **Design** category chooser, select **Constraints**
62. Push the **New Constraint** button from the Edit menu toolbar
63. Enter `Stress` for the Name
64. Make sure the **Stress** response is selected and accept the default **Selected Groups**
65. Enter `10.0` as **Upper Bound**
66. Push **Next>**
67. Push the **Select** button to select both the existing groups
68. Push **Next>**
69. Make sure the **Stress Components** are the **vonMises Top & Bottom**
70. Push **Next>**
71. Select the existing loadcase (`VERTICAL LOAD`)
72. Push **Finish**

    Note that there is 2 response in the constraints list. One constraint for vonMises on the top layer and one for the bottom layer.

Define the Displacement Design Constraint

73. From the **Design** category chooser, select **Constraints**
74. Push the **New Constraint** button from the Edit menu toolbar
75. Enter `End_Displacement` for the Name
76. Select the **Displacement** radio button
77. Enter `0.2` as **Upper Bound**
78. Push **Next>**

79. From the Viewport, select a grid at the free end of the beam
80. Select the **Translation Magnitude** radio button for the component
81. Push **Next>**
82. Select the existing loadcase (`VERTICAL LOAD`)
83. Push **Finish**

Optimize the Structure Using Genesis

84. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button

Import the Shape Changes File

85. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
86. Select the `SHDSG004_dsg.SHP` file
87. Push the **Open** button

Post-Processing the Results (Shape Changes)

88. Select the **Post** tab
89. Push the **Deform/Mesh Color Mesh** button
90. Select a Shape Change for the last design cycle
91. Push the **Filled Contours** radio button
92. Select a Shape Change for the last design cycle in the **Color Mesh** frame
93. Push the **Up** button

Quit Design Studio

94. From the main menu bar, select **File** → **Quit**
95. Push the **Don’t Save** button

6.5 Design using Multiple 2-D (QUAD) Domains

Introduction

The purpose of this example is to learn how to create basic shape optimization data. The analysis model is a cantilevered I-beam modeled with shell elements.

In this problem, multiple 2-dimensional domains are used to associate the grids for defining the shape changes. Three QUAD domains are created along the length of the beam. Perturbations are applied on these domains to design the beam heights along the length of the beam.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

Displacement at the end $\leq$ 1.0

Designable region:

Height of the I-Beam along the length of the beam

Example ID

`SHDSG005`

Special Features:

In this example, end perturbations are applied on the intersection edges of multiple domains such that there is a linear variation along all the QUAD domains that are formed with that edge.

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG005.dat`	Input data	Provided: Contains the finite element mesh of a cantilevered I-Beam modeled plate with applied load and boundary conditions.

SHDSG005_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in SHDSG005.dat
SHDSG005_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
SHDSG005_dsg.SHP	Shape Change data	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization
SHDSG005_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SHDSG005_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG005.dat`

Review the Loadcase

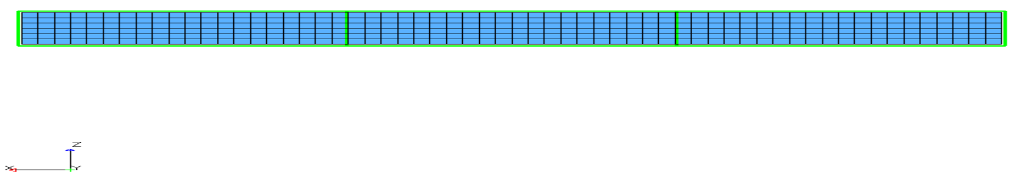
3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase
5. View the loading and boundary conditions on the beam in the Viewport

Create Multiple 2-D (QUAD) Shape Domains

6. From the **Design** category chooser, select **Shape Domains**
7. Push the **New Domain** button from the Edit menu toolbar
8. Enter `Multiple Quads` for the Name
9. Check the **New Domains Quick Setup** radio button
10. Push **Next**>
11. Select `Create New Domain Group` item
12. Push **Next>**
13. Select the **Quads** icon
14. Select the **XZ** icon for **Pick points on workplane**
15. From the Viewport window, change the view to be the XZ plane (Left view)
16. Click-drag-click to create a rectangular domain on the Viewport such that the entire beam is enclosed within the domain as shown in the figure
17. Select the Quad created
18. Push the **Subdivide** button

19. Enter `3` for the Elements in 1st dimension
20. Enter `1` for the Elements in 2nd dimension
21. Push **Next>**
22. Push the **Finish** button.

    In this step, three QUAD domains were created by creating a big domain and dividing them into three equal domains. The created domains should look like the picture below.



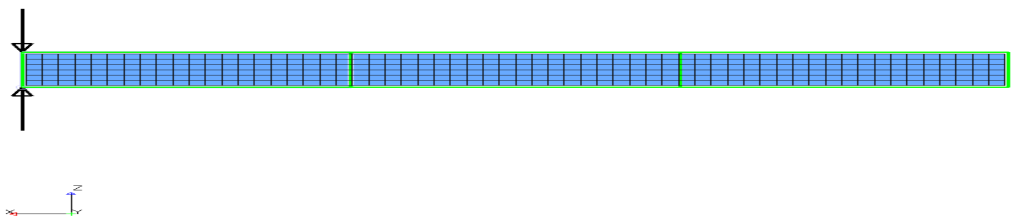
Create Perturbations on Domain Edges

23. From the **Design** category chooser, select **Shape Morphing Sets**
24. Push the **New Shape Set** button from the Edit menu toolbar
25. Enter `Height1` in the Name field
26. Select the **Domain Morphing Set** radio button
27. Push **Next>**
28. Select `Quad ID=1` from the **Select Domains to Act Upon** list
29. Push **Next>**
30. Push the **Select None** button
31. From the Viewport, select one of the corners of the QUAD domain at the fixed end of the beam
32. Enter `0.0`, `0.0`, `1.0` for the **X, Y, Z** to define the direction of the perturbation
33. Enter `45.0` for the **Magnitude**
34. Push the **Add Perturbation** button

    Verify that there is "**1 perturbations on 1 grids"** at the bottom of the window
35. Push the **Select None** button
36. From the Viewport, select the other fixed corner of the QUAD edge
37. Change the **Z** value to `-1.0` for the direction of perturbation

38. Push the **Add Perturbation** button

    Verify that there is “**2 perturbations on 2 grids”** at the bottom of the window. The applied perturbations will look like the picture below.



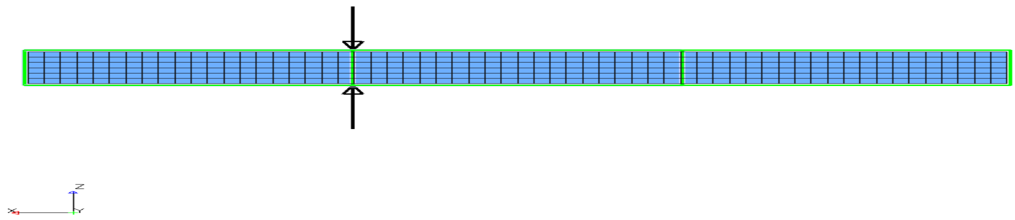
39. Push the **Finish** button

    Now we will create another perturbation to design the intersection edge of QUADs with ID=1 and ID=2.

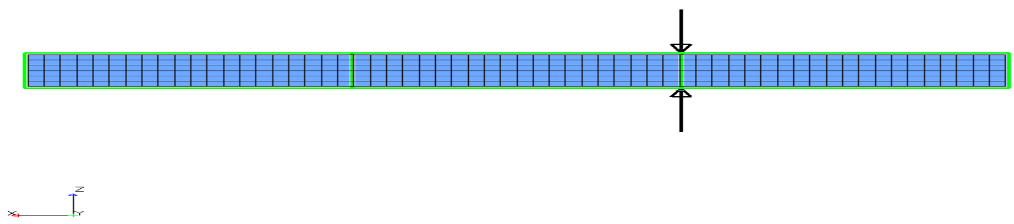
40. Push the **New Shape Set** button from the Edit menu toolbar
41. Enter `Height2` in the Name field
42. Select the **Domain Morphing Set** radio button
43. Push **Next>**
44. Select `Quad ID=1` and `Quad ID=2` from the **Select Domains to Act Upon** list
45. Push **Next>**
46. Push the **Select None** button
47. From the Viewport, select the one of the corners of the intersection edge of the QUAD domains
48. Enter `0.0`, `0.0`, `1.0` for the **X, Y, Z** to define the direction of the perturbation
49. Enter `45.0` for the **Magnitude**
50. Push the **Add Perturbation** button

    Verify that there is “**1 perturbations on 1 grids”** at the bottom of the window
51. Push the **Select None** button
52. From the Viewport, select the other corner of the intersection edge
53. Change the **Z** value to `-1.0` for the direction of perturbation
54. Push the **Add Perturbation** button

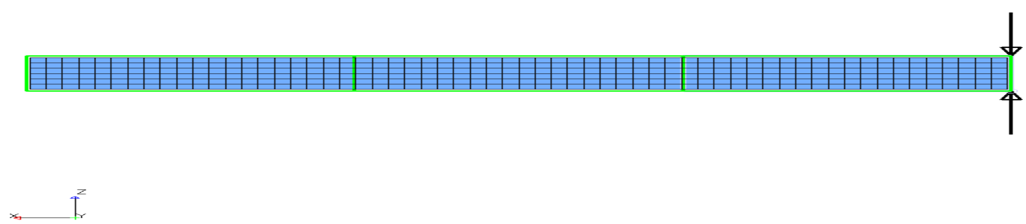
    Verify that there is “**2 perturbations on 2 grids”** at the bottom of the window. The applied perturbations will look like the picture below.



55. Push the **Finish** button
56. Repeat steps 40 through 55 to apply perturbations on the intersection edge of QUADs with `ID=2` and `ID=3`. While repeating step 41, enter `Height3` in the Name field. While repeating step 44, select `Quad ID=2` and `Quad ID=3` from the **Select Domains to Act Upon** list. The perturbations will look like the figure below.



57. Repeat steps 24 through 39 to apply perturbations on the outer edge of QUAD with `ID=3`. While repeating step 25, enter `Height4` in the Name field. While repeating step 28, select `Quad ID=3` from the **Select Domains to Act Upon** list. The perturbations will look like the figure below.



Preview the shape changes

58. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
59. Select the **Oscillate** radio button
60. Under the **Deform Mesh**, select a **Shape Morphing Set Preview**

    Notice the preview of the shape change for DVAR 2 and DVAR 3. Since in this case perturbations are applied on the intersection edges of two QUADs, the perturbation affects both the domains.

61. Push the **Up** button

Define the Design Objective

62. From the **Design** category chooser, select **Objectives**
63. Push the **New Objective** button from the Edit menu toolbar
64. Enter `Mass` for the Name
65. Make sure the **Mass** radio button is selected
66. Push **Finish**

Define the Displacement Design Constraint

67. From the **Design** category chooser, select **Constraints**
68. Push the **New Constraint** button from the Edit menu toolbar
69. Enter `End_Displacement` for the Name
70. Select the **Displacement** radio button
71. Enter `1.0` as **Upper Bound**
72. Push **Next>**
73. From the Viewport, select a grid at the free end of the beam
74. Select the **Translation Magnitude** radio button for the component
75. Push **Next>**
76. Select the existing loadcase (`VERTICAL LOAD`)
77. Push **Finish**

Optimize the Structure Using Genesis

78. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button

Import the Shape Changes File

79. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
80. Select the `SHDSG005_dsg.SHP` file
81. Push the **Open** button

Post-Processing the Results (Shape Changes)

82. Select the **Post** tab
83. Push the **Deform/Mesh Color Mesh** button
84. Select a Shape Change for the last design cycle
85. Push the **Filled Contours** radio button
86. Select a Shape Change for the last design cycle in the **Color Mesh** frame
87. Push the **Up** button

Quit Design Studio

88. From the main menu bar, select **File** → **Quit**
89. Push the **Don’t Save** button

6.6 Design using 3-D (HEXA) Domains

Introduction

The purpose of this example is to learn how to create a basic shape optimization data. The analysis model is a cantilevered beam modeled with solid elements.

In this problem, multiple 3-dimensional domains are used to associate the grids for defining the shape changes. Two HEXA domains are created along the length of the beam. Perturbations are applied on these domains to design the cross-section at two locations along the length of the beam.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

Displacement at the end $\leq$ 0.2

Vonmises Stress $\leq$ 330

Designable region:

Cross-section of the Beam at two locations along the length of the beam

Example ID

`SHDSG006`

Special Features:

In this example, 3-D HEXA domains are created by dragging out the domain and moving the domain in space so that it encloses the entire beam. Design Studio has the ability to move the entire domain in space to place it at the appropriate location. Also the user has the freedom of selecting a face or edge or vertex of the domain to resize the domain. After the domains are created, end perturbations are applied on one face of the HEXA domain such that there is a linear variation along all the length of the HEXA domain.

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SHDSG006.dat	Input data	Provided: Contains the finite element mesh of a cantilevered beam modeled using solid elements with applied load and boundary conditions.
SHDSG006_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in SHDSG006.dat
SHDSG006_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
SHDSG006_dsg.SHP	Shape Change data	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization
SHDSG006_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SHDSG006_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: SHDSG006.dat

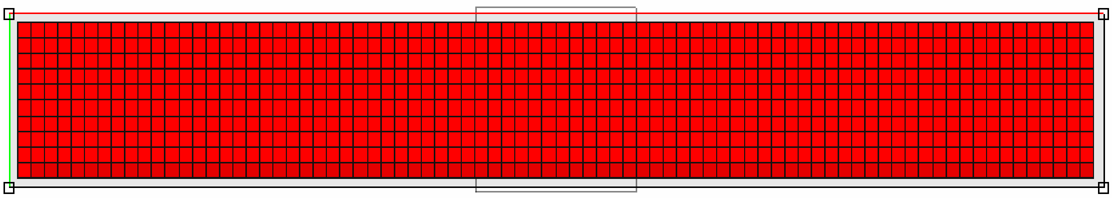
Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase
5. View the loading and boundary conditions on the beam in the Viewport

Create Multiple 3-D (HEXA) Shape Domains

6. From the **Design** category chooser, select **Shape Domains**
7. Push the **New Domain** button from the Edit menu toolbar
8. Enter Two Hexas for the Name
9. Check the **New Domains Quick Setup** radio button
10. Push **Next**>
11. Select the Create New Domain Group item
12. Push **Next>**
13. From the Viewport window, change the view to be the XZ plane (Left view)
14. Select the **Hexas** icon

15. Select the **SCR** icon (XY plane of screen) for **Pick points on workplane**
16. Click-drag-release to create a rectangular domain on the Viewport such that the entire beam is enclosed within the domain as shown in the figure below.

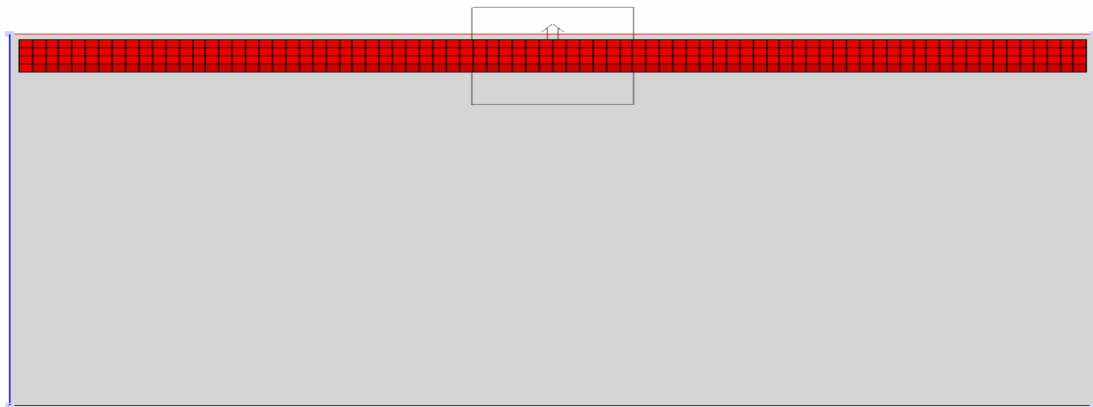


Now that we have created a Hexa domain we are going to move/ resize it so that the entire beam is enclosed in the domain.

17. Select the **Adjust Regions** (arrow icon) button from the **Region Definition Options**
18. From the Viewport window, change the view to be the XY plane (Top view)

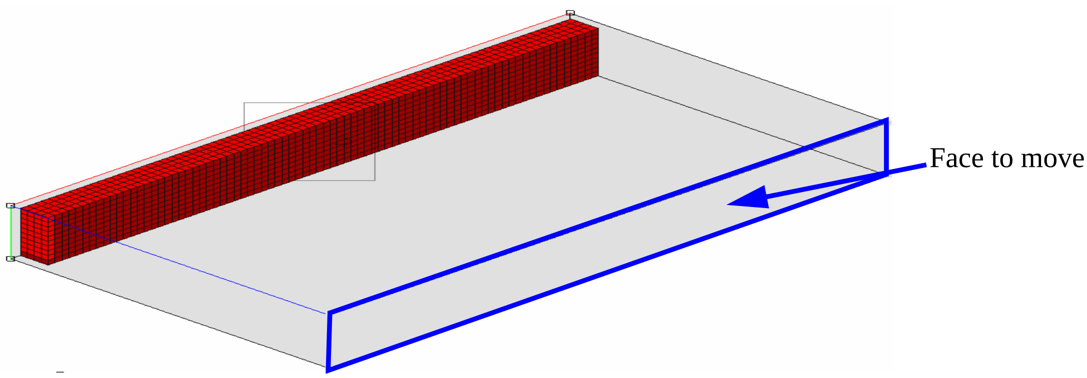
Notice that the beam is not fully enclosed in the domain.

19. To move the entire domain, click-and-hold on the domain and move it in the positive Y-direction so that it encloses the entire beam as shown in the figure.



20. From the Viewport window, change the view to be the Iso Front-Left-Top view

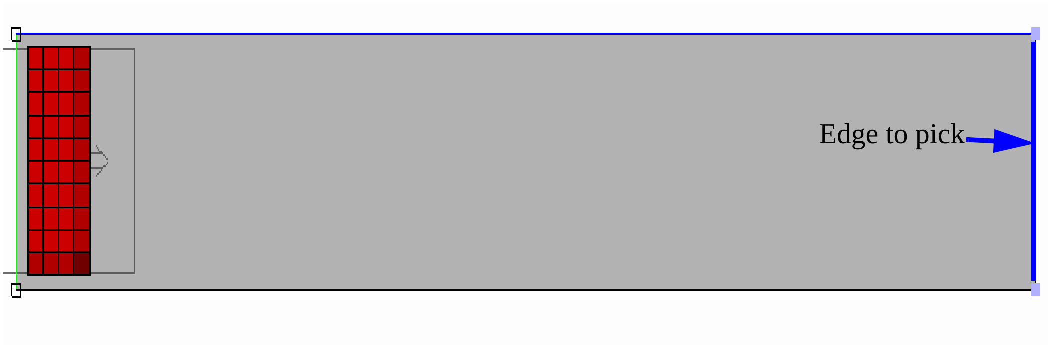
Now we need to resize the domain so that it will enclose only the beam. To do this we need to select the entire face (shown in the figure below) and move it.



21. To select the face, double click on the face shown in the picture above

    Notice that once the face is selected, the small rectangular boxes at the corners of the face are filled.
    Now that you have selected the face, we will move the entire face in the positive Y-direction.

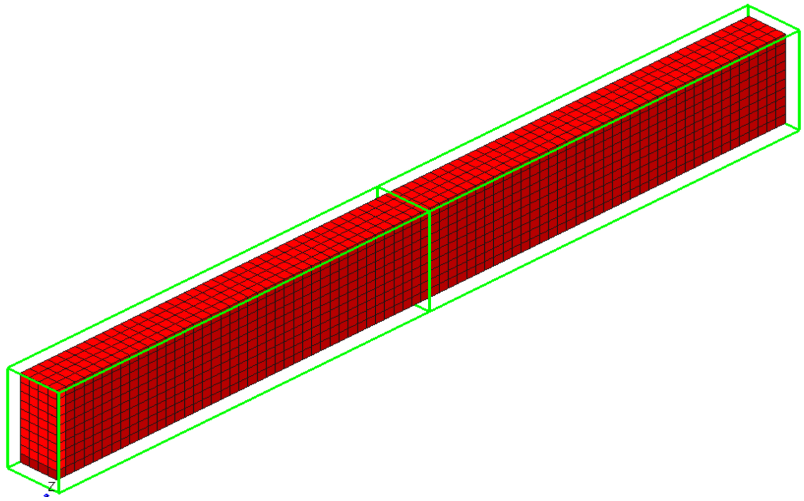
22. From the Viewport window, change the view to be the YZ plane(Front view) as shown in the figure below.



23. To resize the domain, click-and-hold on the line (representing the face selected) and move the line to the appropriate size so that the entire beam is enclosed as shown in the figure below.

24. Click the different view in the Viewport window and make sure the entire beam is enclose in the domain.
25. Select the Hexa created
26. Push the **Subdivide** button
27. Enter `2` for the Elements in 1st dimension
28. Enter `1` for the Elements in 2nd dimension
29. Enter `1` for the Elements in 3rd dimension
30. Push **Next>**
31. Push the **Finish** button.

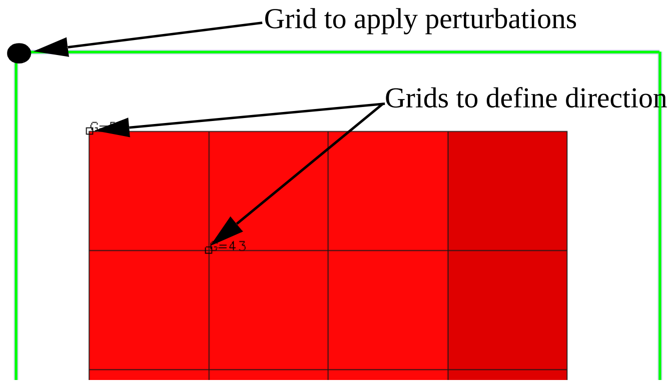
    In this step, two HEXA domains were created by creating a domain moving/resizing it to enclose the entire beam and dividing it into two equal domains. The created domains should look like the picture below.



Create Perturbations to Design Cross-section Size

32. From the **Design** category chooser, select **Shape Morphing Sets**
33. Push the **New Shape Set** button from the Edit menu toolbar
34. Enter `End_Crossection` in the Name field
35. Select the **Domain Morphing Set** radio button
36. Push **Next>**
37. Select `Hexa ID=1` from the **Select Domains to Act Upon** list
38. Push **Next>**
39. From the Viewport window, change the view to be the YZ plane(Front view)
40. Push the **Select None** button
41. From the Viewport, select one of the corners of the HEXA domain at the free end of the beam
42. Push the **By 2 Grids...** button

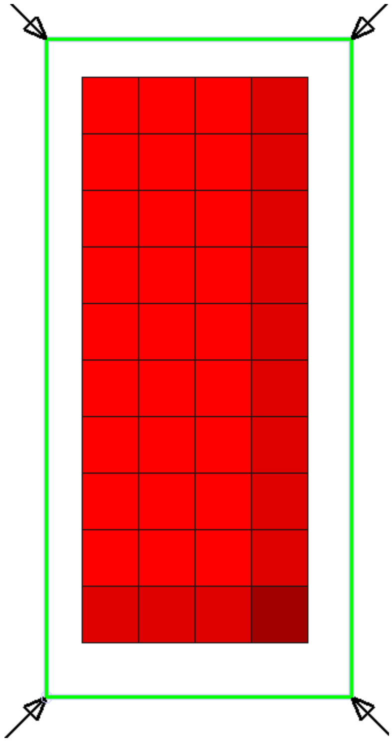
43. Select two grids along the diagonal to define the direction of the perturbation as shown in the figure below



44. Push **Next>**
45. Enter `3.0` for the **Magnitude**
46. Push the **Add Perturbation** button

    Verify that there is "**1 perturbations on 1 grids**" at the bottom of the window
47. Push the **Select None** button
48. Repeat the process to apply perturbations at all four corners of the HEXA face.

    Verify that there is "**4 perturbations on 4grids**" at the bottom of the window. The applied perturbations will look like the picture below.

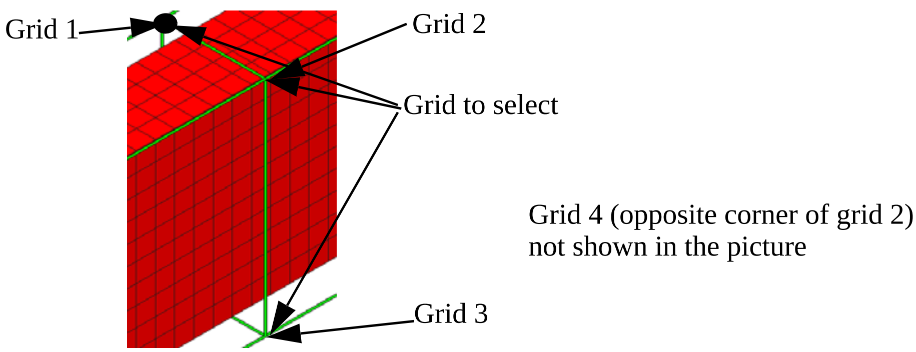


49. Push the **Finish** button

Now we will create another perturbation to on the intersection face of the two HEXAs.

50. Push the **New Shape Set** button from the Edit menu toolbar
51. Enter `Center_Crossection` in the Name field
52. Select the **Domain Morphing Set** radio button
53. Push **Next>**
54. Select `Hexa ID=1` and `Hexa ID=2` from the **Select Domains to Act Upon** list
55. Push **Next>**
56. Push the **Select None** button
57. From the Viewport, select Grid 1 (one corner of the intersection face of the HEXA domains) as shown in the figure below

    To view the grids for the domain, one can go to the **Display** tab and hide the PSOLID in the **Show/Hide Groups**

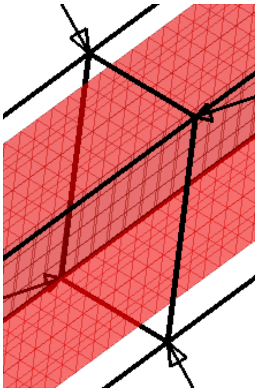


58. Enter `0.0`, `-0.7071`, `-0.7071` for the **X, Y, Z** to define the direction of the perturbation
59. Enter `3.0` for the **Magnitude**
60. Push the **Add Perturbation** button

61. Repeat steps 56 through 60, to apply perturbations on the remaining three corners of the face. Use the data in the table below, for the perturbation directions

Grid Label	Direction
Grid 1	`0.0, -0.7071, -0.7071`
Grid 2	`0.0, 0.7071, -0.7071`
Grid 3	`0.0, 0.7071, 0.7071`
Grid 4	`0.0, -0.7071, 0.7071`

Verify that there is “**4 perturbations on 4 grids”** at the bottom of the window. The applied perturbations will look like the picture below.



62. Push the **Finish** button

Preview the shape changes

63. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
64. Select the **Oscillate** radio button
65. Under the **Deform Mesh**, select a **Shape Morphing Set Preview**
66. Push the **Up** button

Define the Design Objective

67. From the **Design** category chooser, select **Objectives**
68. Push the **New Objective** button from the Edit menu toolbar
69. Enter `Mass` for the Name
70. Make sure the **Mass** radio button is selected

71. Push **Finish**

Define the Displacement Design Constraint

72. From the **Design** category chooser, select **Constraints**
73. Push the **New Constraint** button from the Edit menu toolbar
74. Enter `End_Displacement` for the Name
75. Select the **Displacement** radio button
76. Enter `0.2` as **Upper Bound**
77. Push **Next>**
78. From the Viewport window, change the view to be the XZ plane(Left view)
79. From the Viewport, select all the grids at the free end of the beam
80. Select the **Translation Magnitude** radio button for the component
81. Push **Next>**
82. Select the existing loadcase (`Loadcase 1`)
83. Push **Finish**

Define the Stress Design Constraint

84. Push the **New Constraint** button from the Edit menu toolbar
85. Enter `Stress` for the Name
86. Select the **Stress** radio button
87. Enter `330.0` as **Upper Bound**
88. Push **Next>**
89. Select the existing PSOLID from the **Choose Stress Groups** list
90. Push **Next>**
91. Select vonMises from the dropdown listbox for PSOLID Stress
92. Push **Next>**
93. Select the existing loadcase (`Loadcase 1`)
94. Push **Finish**

Optimize the Structure Using Genesis

95. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button

Import the Shape Changes File

96. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
97. Select the `SHDSG006_dsg.SHP` file
98. Push the **Open** button

Post-Processing the Results (Shape Changes)

99. Select the **Post** tab
100. Push the **Deform/Mesh Color Mesh** button
101. Select a Shape Change for the last design cycle
102. Push the **Filled Contours** radio button
103. Select a Shape Change for the last design cycle in the **Color Mesh** frame
104. Push the **Up** button

Quit Design Studio

105. From the main menu bar, select **File** → **Quit**
106. Push the **Don’t Save** button

6.7 Design using 3-D (HEXA) and 2-D (QUAD) Domains

Introduction

The purpose of this example is to learn how to create shape optimization data. The analysis model is a cantilevered hollow beam modeled with solid elements.

In this problem, multiple 2-dimensional domains are used to associate the grids for defining the shape changes on the cross-section of the beam. As the beam is hollow, several QUAD domains are created and the domains that are not needed are deleted. A HEXA domain is also created along the length of the beam. Perturbations are applied on these domains to design a uniform wall thickness of the hollow beam and also the taper of the beam.

The following optimization problem will be created, solved and post-processed:

Minimize Displacement at the free end

Subject to:

Mass $\leq$ 4E-6

Designable region:

Wall thickness of the hollow beam

Taper of the beam along its length

Example ID

`SHDSG007`

Special Features:

In this example, multiple 2-D QUAD domains are created by creating a big domain and splitting it into smaller domains. As the beam is hollow, the domains within the interior of the beam that have no grids associated with them are deleted. Design Studio has the capability of creating each of these smaller domains individually. It is easier to create the smaller domains by splitting a bigger one and deleting the extra domains.

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SHDSG007.dat	Input data	Provided: Contains the finite element mesh of a cantilevered beam modeled using solid elements with applied load and boundary conditions.
SHDSG007_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in SHDSG007.dat
SHDSG007_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
SHDSG007_dsg.SHP	Shape Change data	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization
SHDSG007_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SHDSG007_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG007.dat`

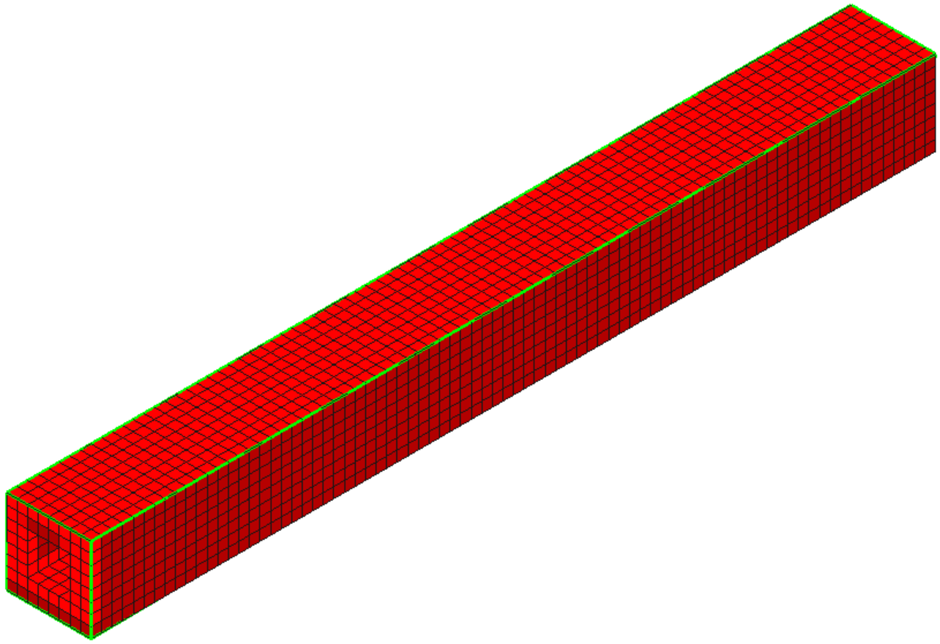
Create Multiple 3-D (HEXA) Shape Domain

3. From the **Design** category chooser, select **Shape Domains**
4. Push the **New Domain** button from the Edit menu toolbar
5. Enter `Hexa` for the Name
6. Check the **New Domains Quick Setup** radio button
7. Push **Next**>
8. Select the `Create New Domain Group` item
9. Push **Next>**
10. Select the **Point-by-Point** radio button
11. Select the **Hexas** icon
12. Select the **Pick existing grids** radio button
13. Select the eight corners grids of the beam to create the Hexa domain

    Use the right-hand rule to pick the bottom (or left side) four grids first and the follow the same order for the top (or right side) four grids.

14. Push the **Finish** button.

    The created domain should look like the picture below.



Review the Shape Domains

15. From the **Design** category chooser, select **Shape Domains**
16. Select the existing HEXA domain
17. Push the **Modify Domain** button from the Edit Menu toolbar

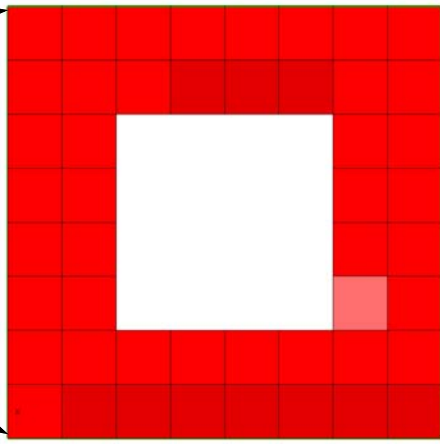
    Notice that the grids controlled by the HEXA domains are selected.

18. Push the **Cancel** button
19. Right click on the Viewport window and select **Clear** → **All** to clear the selection

Create Perturbations to Design Taper of the Beam

20. From the **Design** category chooser, select **Shape Morphing Sets**
21. Push the **New Shape Set** button from the Edit menu toolbar
22. Enter `Taper` in the Name field
23. Select the **Domain Morphing Set** radio button
24. Push **Next>**
25. Select `Hexa ID=1` from the **Select Domains to Act Upon** list
26. Push **Next>**

27. From the Viewport window, change the view to be the YZ plane(Back view)
28. Push the **Select None** button
29. From the Viewport, select one of the corners of the HEXA domain (Grid 1 in the figure below) at the fixed end of the beam
30. Enter `0.0`, `-0.7071`, `0.7071` for the **X, Y, Z** to define the direction of the perturbation

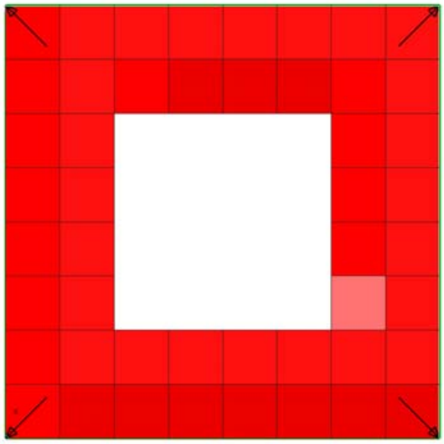


31. Enter `1.0` for the **Magnitude**
32. Push the **Add Perturbation** button

    Verify that there is "**1 perturbations on 1 grids"** at the bottom of the window
33. Repeat steps 23 through 27 to apply perturbations at all four corners of the HEXA face.The direction of the perturbations are given in the table below

Grid Label	Direction
Grid 1	`0.0, -0.7071, 0.7071`
Grid 2	`0.0, 0.7071, 0.7071`
Grid 3	`0.0, 0.7071, -0.7071`
Grid 4	`0.0, -0.7071, -0.7071`

Verify that there is "**4 perturbations on 4grids"** at the bottom of the window. The applied perturbations will look like the picture below.



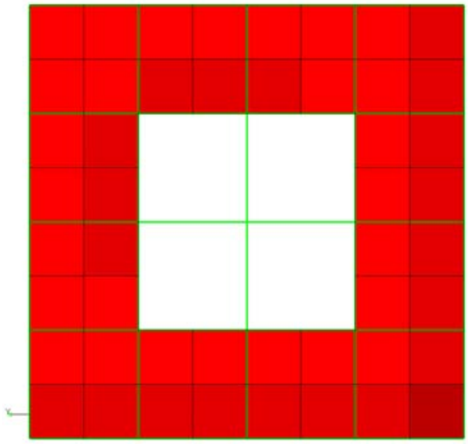
34. Push the **Finish** button

Create Multiple 2-D (QUAD) Shape Domains

35. From the **Design** category chooser, select **Shape Domains**
36. Push the **New Domain** button from the Edit menu toolbar
37. Enter `Quads` for the Name
38. Check the **New Domains Quick Setup** radio button
39. Push **Next**>
40. Select the `Create New Domain Group` item
41. Push **Next>**
42. From the Viewport window, change the view to be the YZ plane(Front view)
43. Select the **Point-by-Point** radio button
44. Select the **Quads** icon
45. Select the **Pick existing grids** radio button
46. Select the four corners grids on the front face of the beam
47. Select the Quad created
48. Push the **Subdivide** button
49. Enter 4 for the Elements in 1st dimension
50. Enter 4 for the Elements in 2nd dimension
51. Push **Next>**

52. Push the **Finish** button.

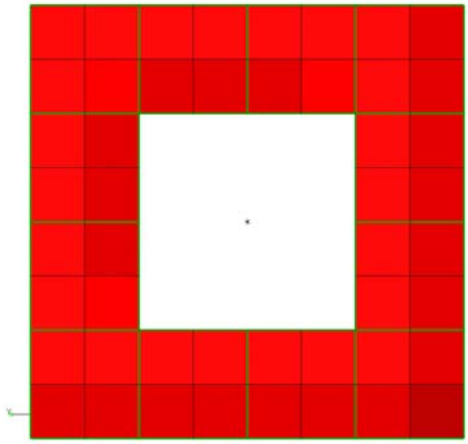
    The created domain should look like the picture below.



Deleting Shape Domains

53. From the **Design** category chooser, select **Shape Domains**
54. From the Viewport window, select the four domains that are in the hole in the beam

    Verify that there is “4 domains selected” in bottom of the main window
55. From the main menu bar, select **Edit** → **Delete Domain**



Deleting the Free Grid

56. From the **Analysis** category chooser, select **Grids**
57. From the main menu bar, select **Edit** → **Select All**
58. From the Edit menu toolbar, select **Delete Grids** button

Review the Shape Domains

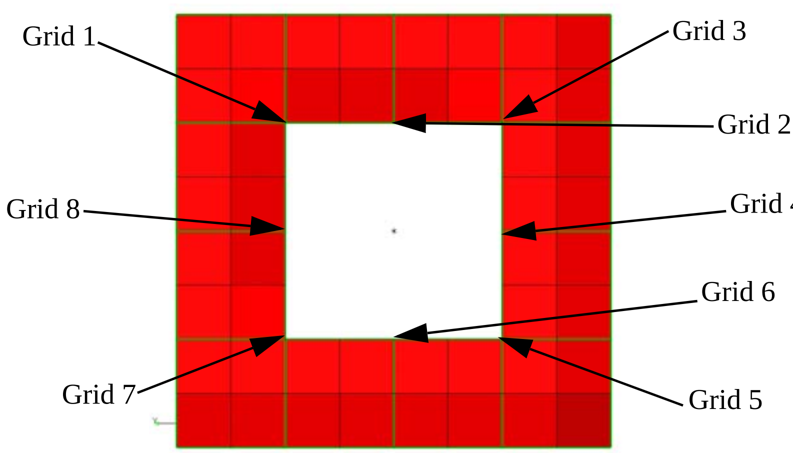
59. From the **Design** category chooser, select **Shape Domains**
60. Select one of the existing QUAD domains
61. Push the **Modify Domain** button from the Edit Menu toolbar

    Notice that the grids controlled by the QUAD domains are selected.
62. Push the **Cancel** button
63. Right click on the Viewport window and select **Clear** → **All** to clear the selection

Create Perturbations to Design Wall Thickness

64. From the **Design** category chooser, select **Shape Morphing Sets**
65. Push the **New Shape Set** button from the Edit menu toolbar
66. Enter `Wall Thickness` in the Name field
67. Select the **Domain Morphing Set** radio button
68. Push **Next>**
69. Select all the existing QUAD domains from the **Select Domains to Act Upon** list

    Verify that there is "**12 domains selected**'" at the bottom of the window.
70. Push **Next>**
71. From the Viewport window, change the view to be the YZ plane(Front view)
72. Push the **Select None** button
73. From the Viewport, select one of the corners of the HEXA domain (Grid 1 in the figure below) at the fixed end of the beam



74. Enter `0.0`, `0.7071`, `0.7071` for the **X, Y, Z** to define the direction of the perturbation

75. Enter `1.0` for the **Magnitude**

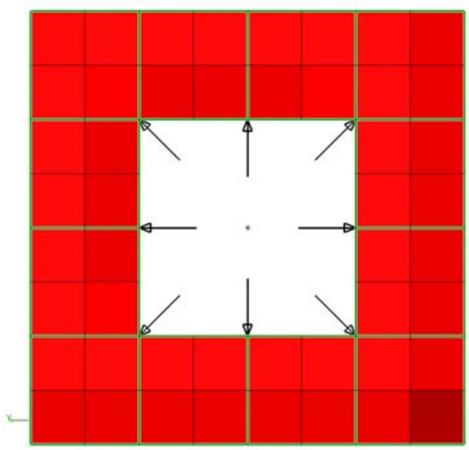
76. Push the **Add Perturbation** button

    Verify that there is “**1 perturbations on 1 grids”** at the bottom of the window

77. Repeat steps 72 through 76 to apply perturbations at all eight corners of the QUAD domains along the hole.The magnitude and direction of the perturbations are given in the table below

Grid Label	Magnitude	Direction
Grid 1	`1.0`	`0.0, 0.7071, 0.7071`
Grid 2	`0.7071`	`0.0, 0.0, 1.0`
Grid 3	`1.0`	`0.0, -0.7071, 0.7071`
Grid 4	`0.7071`	`0.0, -1.0, 0.0`
Grid 5	`1.0`	`0.0, -0.7071, -0.7071`
Grid 6	`0.7071`	`0.0, 0.0, -1.0`
Grid 7	`1.0`	`0.0, 0.7071, -0.7071`
Grid 8	`0.7071`	`0.0, 1.0, 0.0`

Verify that there is “**8 perturbations on 8 grids”** at the bottom of the window. The applied perturbations will look like the picture below.



78. Push the **Finish** button

Preview the shape changes

79. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
80. Select the **Oscillate** radio button
81. Under the **Deform Mesh**, select a **Shape Morphing Set Preview**
82. Push the **Up** button

Define the Displacement Design Objective

83. From the **Design** category chooser, select **Objectives**
84. Push the **New Objective** button from the Edit menu toolbar
85. Enter `Displacement` for the Name
86. Select the **Displacement** radio button
87. Push **Next>**
88. Push the **Select None** button
89. From the Viewport, select the grid where the load is applied (middle of the top surface of the beam at the free end)
90. Select the **Translation Magnitude** radio button for the component
91. Push **Next>**
92. Select the existing loadcase
93. Push **Finish**

Define the Mass Design Constraint

94. From the **Design** category chooser, select **Constraints**
95. Push the **New Constraint** button from the Edit menu toolbar
96. Enter `Mass` for the Name
97. Select the **Mass** radio button
98. Enter `4E-6` as **Upper Bound**
99. Push **Finish**

Optimize the Structure Using Genesis

100. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button

Import the Shape Changes File

101. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
102. Select the `SHDSG007_dsg.SHP` file
103. Push the **Open** button

Post-Processing the Results (Shape Changes)

104. Select the **Post** tab
105. Push the **Deform/Mesh Color Mesh** button
106. Select a Shape Change for the last design cycle
107. Push the **Filled Contours** radio button
108. Select a Shape Change for the last design cycle in the **Color Mesh** frame
109. Push the **Up** button

Quit Design Studio

110. From the main menu bar, select **File** → **Quit**
111. Push the **Don’t Save** button

6.8 Design a Feature - Using 2-D (TRIA & QUAD) Domains

Introduction

The purpose of this example is to learn how to use shape optimization to design geometric features of a structure. The analysis model is a shell element FEM of a bracket.

A bracket with a triangular hole is modeled using shell elements. Shape optimization is used to design the size and location of the hole. In this example, transition domains are used to minimize mesh distortions.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

vonMises Stress in the bracket $\leq$ 75.0

Designable region:

Location of the triangular hole in the bracket

Size of the triangular hole

Example ID

`SHDSG008`

Special Features:

In this example, perturbations are applied so that a feature with the structure could be designed. In this example, 2-D TRIA domains are used to design the location and size of a triangular hole in a bracket.

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SHDSG008.dat	Input data	Provided: Contains the finite element mesh of a bracket with applied load and boundary conditions.

`SHDSG008_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `SHDSG008.dat`
`SHDSG008_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SHDSG008_dsg.SHP`	`Shape Change data`	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization
`SHDSG008_ref.dat`	`Input data`	Provided. Reference result input file. Should be nearly identical to `SHDSG008_dsg.dat`. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG008.dat`

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase
5. View the loading and boundary conditions on the beam in the Viewport
6. From the main menu, select **Edit** → **Deselect All** to clear the loadcase selection

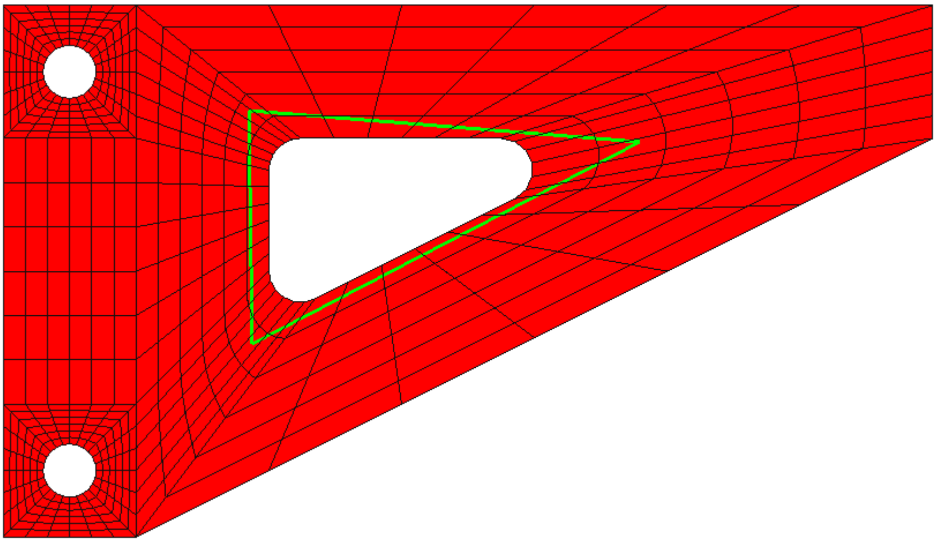
Create Multiple 2-D Shape Domains

In this step, 2 TRIA and 3 QUAD domains are created to enclose the bracket. Even though only the hole is being designed, domains are created on the entire bracket. The domains surrounding the hole are used as transition domains to avoid mesh distortions while designing the hole.

7. From the Viewport window, change the view to be the XY plane (Top view)
8. From the **Design** category chooser, select **Shape Domains**
9. Push the **New Domain** button from the Edit menu toolbar
10. Enter `Hole Domain` for the Name
11. Check the **New Domains Quick Setup** radio button
12. Push **Next>**
13. Select the `Create New Domain Group` item
14. Push **Next>**
15. Select the **Trias** icon
16. Select the **XY** icon for **Pick points on workplane**

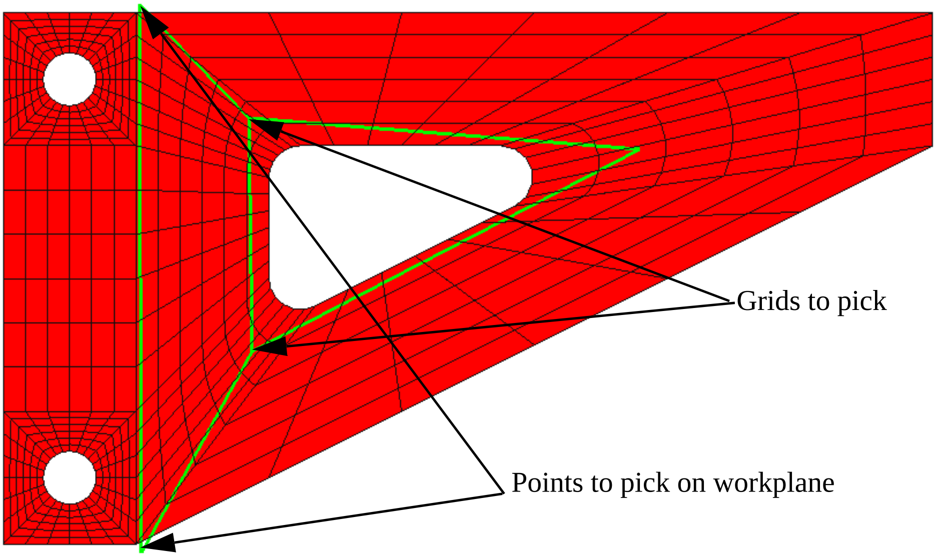
17. Click-drag-release to create a triangular domain on the Viewport such that the hole is enclosed within the domain as shown in the figure

    Once the triangle is created, design studio has the capability to move/resize the domain. For resizing, one can select edges or vertices of the triangle.



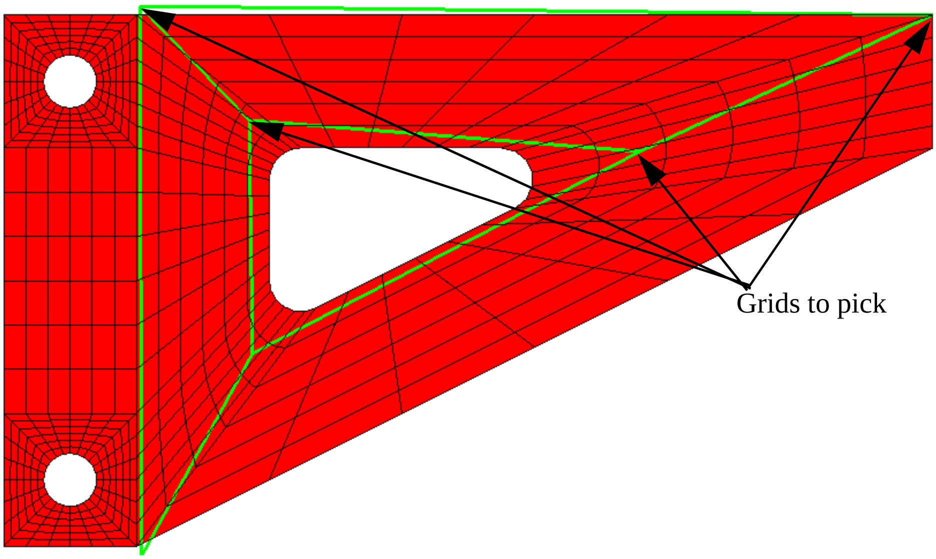
18. Push the **Finish** button.
19. Push the **New Domain** button from the Edit menu toolbar
20. Enter `Transition Domain 1` for the Name
21. Check the **New Domains Quick Setup** radio button
22. Push **Next**>
23. Select the existing group to create the Domains
24. Push **Next>**
25. Select the **Point-by-Point** radio button
26. Select the **Quads** icon
27. Select the **XY** icon for **Pick points on workplane**
28. Click on the viewport to select the first two points (shown in the figure below) of the quad domain
29. Select the **Pick existing grids radio button**
30. Pick the two corners of the adjacent triangular domain that attaches to this quad Hole domain
31. Push the **Finish** button.

    The QUAD domain created should look like the figure below. In creating the QUAD domain, two new grids were created by picking points on the workplane. Alternatively the user, can select the existing grids in the vicinity by using the **Pick existing grids** option.



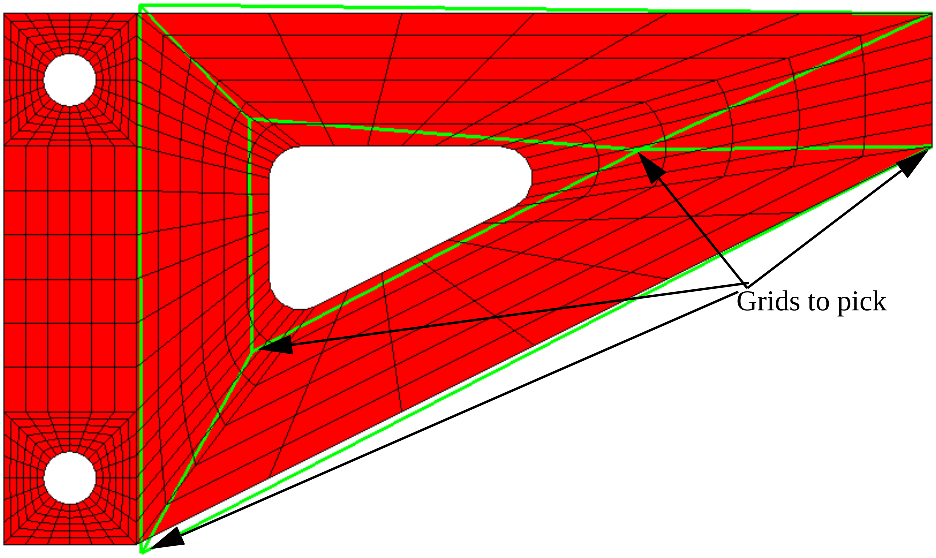
32. Push the **New Domain** button from the Edit menu toolbar
33. Enter `Transition Domain 2` for the Name
34. Check the **New Domains Quick Setup** radio button
35. Push **Next**>
36. Select the existing group to create the Domains
37. Push **Next>**
38. Select the **Point-by-Point** radio button
39. Select the **Quads** icon
40. Select the **Pick existing grids** radio button

41. Pick the four existing grids, as shown in the figure below, for the corners of the quad domain



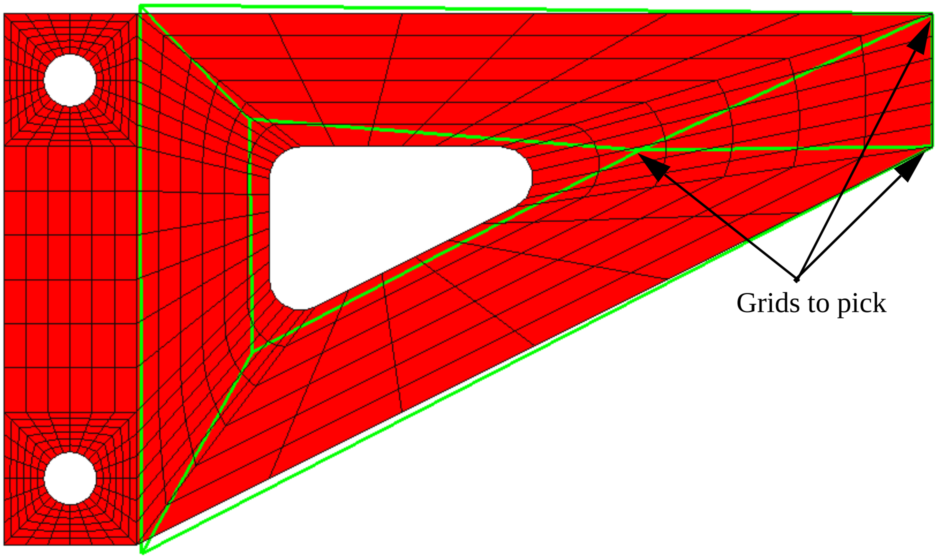
42. Push the **Finish** button.
43. Push the **New Domain** button from the Edit menu toolbar
44. Enter `Transition Domain 3` for the Name
45. Check the **New Domains Quick Setup** radio button
46. Push **Next**>
47. Select the existing group to create the Domains
48. Push **Next>**
49. Select the **Point-by-Point** radio button
50. Select the **Quads** icon
51. Select the **Pick existing grids** radio button

52. Pick the four existing grids, as shown in the figure below, for the corners of the quad domain



53. Push the **Finish** button.
54. Push the **New Domain** button from the Edit menu toolbar
55. Enter `Transition Domain 4` for the Name
56. Check the **New Domains Quick Setup** radio button
57. Push **Next**>
58. Select the existing group to create the Domains
59. Push **Next>**
60. Select the **Point-by-Point** radio button
61. Select the **Trias** icon
62. Select the **Pick existing grids** radio button

63. Pick the three existing grids, as shown in the figure below, for the corners of the quad domain

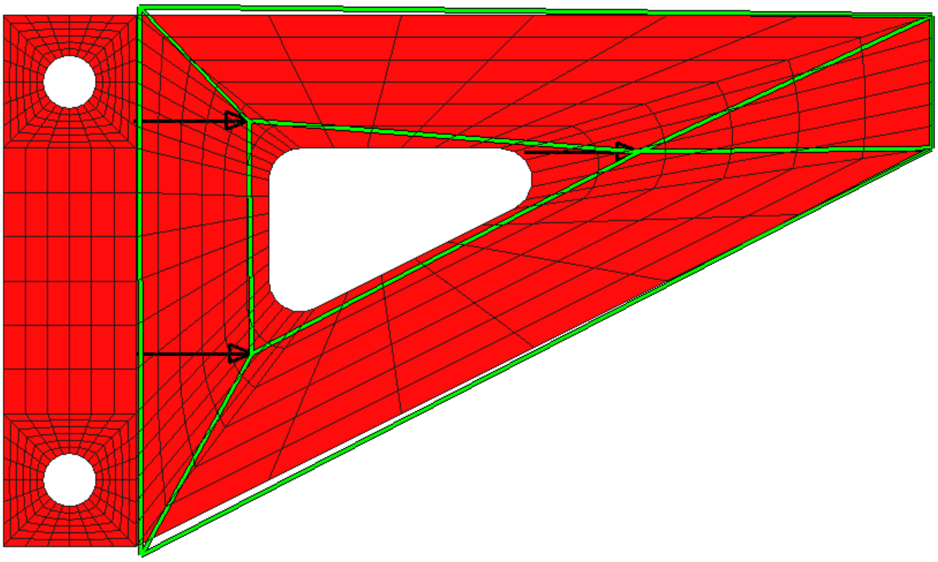


64. Push the **Finish** button.

Create Perturbations to Design Hole Location

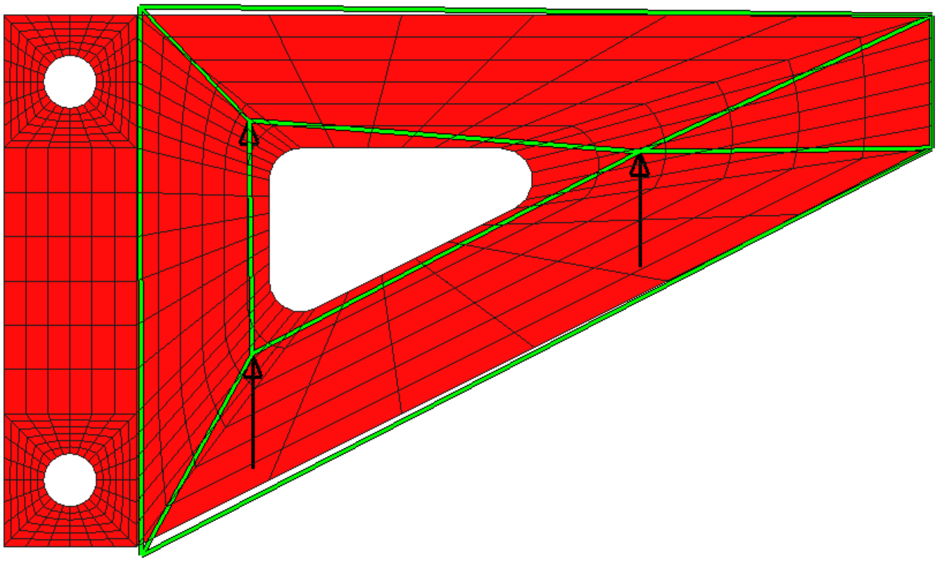
65. From the **Design** category chooser, select **Shape Morphing Sets**
66. Push the **New Shape Set** button from the Edit menu toolbar
67. Enter `X-Location` in the Name field
68. Select the **Domain Morphing Set** radio button
69. Push **Next>**
70. Select all the existing domains from the **Select Domains to Act Upon** list
71. Push **Next>**
72. Push the **Select None** button
73. From the Viewport, select the three corners of the TRIA domain that encloses the hole
74. Enter `1.0`, `0.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation
75. Enter `10.0` for the **Magnitude**
76. Push the **Add Perturbation** button

Verify that there is “**3 perturbations on 3 grids”** at the bottom of the window. The applied perturbations will look like the picture below.



77. Push the **Finish** button
78. Push the **New Shape Set** button from the Edit menu toolbar
79. Enter `Y-Location` in the Name field
80. Select the **Domain Morphing Set** radio button
81. Push **Next>**
82. Select all the existing domains from the **Select Domains to Act Upon** list
83. Push **Next>**
84. Push the **Select None** button
85. From the Viewport, select the three corners of the TRIA domain that encloses the hole
86. Enter `0.0`, `1.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation
87. Enter `10.0` for the **Magnitude**
88. Push the **Add Perturbation** button

Verify that there is “**3 perturbations on 3 grids”** at the bottom of the window. The applied perturbations will look like the picture below.



89. Push the **Finish** button

Create Perturbations to Design Hole Size

90. From the **Design** category chooser, select **Shape Morphing Sets**
91. Push the **New Shape Set** button from the Edit menu toolbar
92. Enter `Hole Size-X` in the Name field
93. Select the **Domain Morphing Set** radio button
94. Push **Next>**
95. Select all the existing domains from the **Select Domains to Act Upon** list
96. Push **Next>**
97. Push the **Select None** button
98. From the Viewport, select the two left corners of the TRIA domain that encloses the hole
99. Enter `1.0`, `0.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation
100. Enter `10.0` for the **Magnitude**
101. Push the **Add Perturbation** button

Verify that there is “**2 perturbations on 2 grids”** at the bottom of the window.

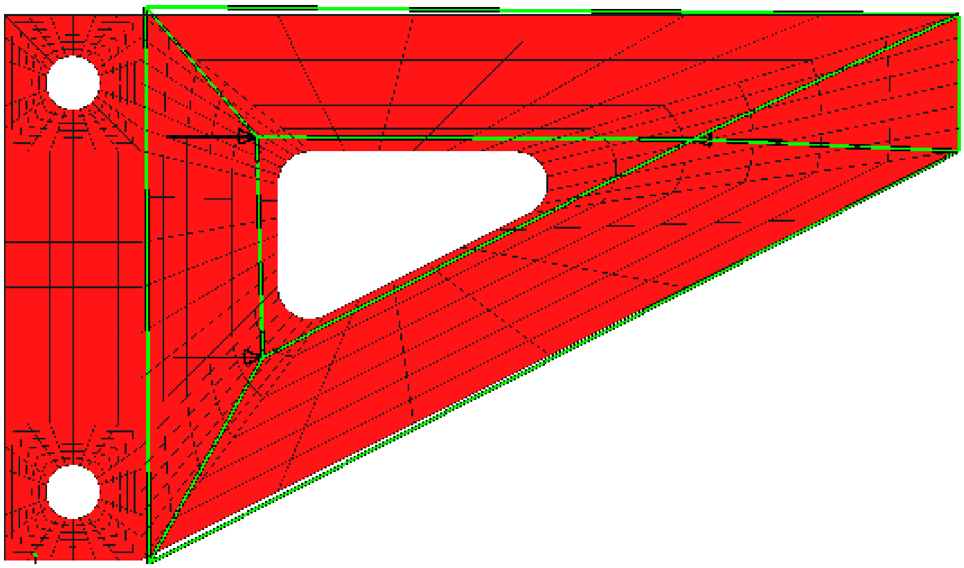
102. Push the **Select None** button
103. From the Viewport, select the third corner of the TRIA domain that encloses the hole

104. Enter `-1.0`, `0.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation

105. Enter `10.0` for the **Magnitude**

106. Push the **Add Perturbation** button

    Verify that there is “**3 perturbations on 3 grids”** at the bottom of the window. The applied perturbations will look like the picture below.



107. Push the **Finish** button

108. Push the **New Shape Set** button from the Edit menu toolbar

109. Enter `Hole Size-Y` in the Name field

110. Select the **Domain Morphing Set** radio button

111. Push **Next>**

112. Select all the existing domains from the **Select Domains to Act Upon** list

113. Push **Next>**

114. Push the **Select None** button

115. From the Viewport, select the two top corners of the TRIA domain that encloses the hole

116. Enter `0.0`, `-1.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation

117. Enter `10.0` for the **Magnitude**

118. Push the **Add Perturbation** button

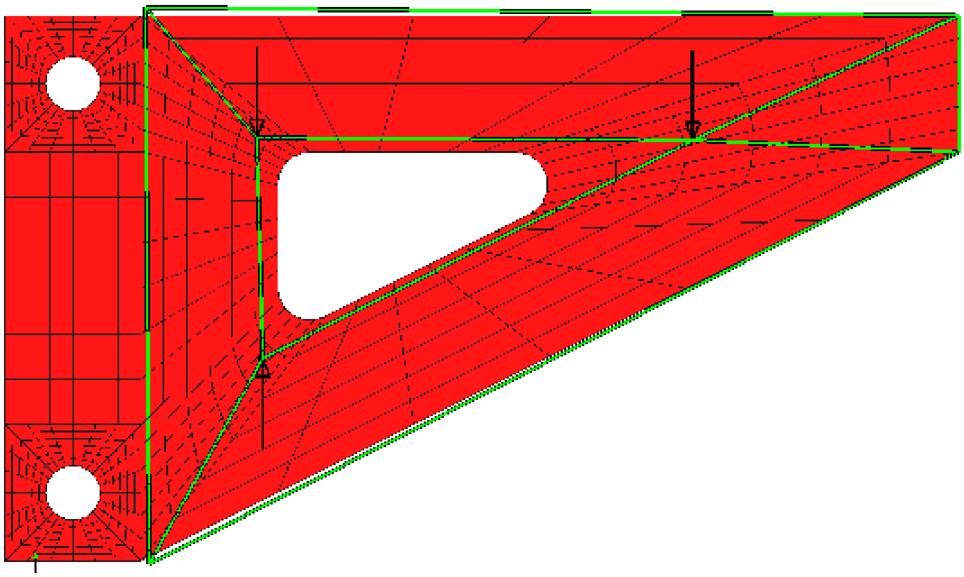
    Verify that there is “**2 perturbations on 2 grids”** at the bottom of the window.

119. Push the **Select None** button

120. From the Viewport, select the third corner of the TRIA domain that encloses the hole

121. Enter `0.0`, `1.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation
122. Enter `10.0` for the **Magnitude**
123. Push the **Add Perturbation** button

    Verify that there is "**3 perturbations on 3 grids**" at the bottom of the window. The applied perturbations will look like the picture below.



124. Push the **Finish** button

Preview the shape changes

125. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
126. Select the **Oscillate** radio button
127. Under the **Deform Mesh**, select a **Shape Morphing Set Preview**
128. Push the **Up** button

Define the Design Objective

129. From the **Design** category chooser, select **Objectives**
130. Push the **New Objective** button from the Edit menu toolbar
131. Enter `Mass` for the Name
132. Make sure the **Mass** radio button is selected
133. Push **Finish**

Define the Stress Design Constraint

134. From the **Design** category chooser, select **Constraints**
135. Push the **New Constraint** button from the Edit menu toolbar
136. Enter `Stresses` for the Name
137. Make sure the **Stress** response is selected and accept the default **Selected Groups**
138. Enter `75.0` as **Upper Bound**
139. Push **Next>**
140. Push the **Select** button to select the existing groups
141. Push **Next>**
142. Make sure the **Stress Components** are the **vonMises Top & Bottom**
143. Push **Next>**
144. Select the existing loadcase
145. Push **Finish**

Add option for Mesh Smoothing

146. From the main menu bar, select **Genesis** → **Options...**
147. Select the **Design Control** tab
148. Push the **Advanced...** button
149. Select the **Misc.** tab
150. Check **Mesh Smoothing**
151. From the category chooser, select **Yes**

    If MSMOOTH = YES, then mesh smoothing is performed on 2D planar surfaces and 3D elements.

152. Push the **Close** button
153. Push the **Apply** button

Optimize the Structure Using Genesis

154. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button

Import the Shape Changes File

155. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**

156. Select the `SHDSG008_dsg.SHP` file
157. Push the **Open** button

Post-Processing the Results (Shape Changes)

158. Select the **Post** tab
159. Push the **Deform/Mesh Color Mesh** button
160. Select a Shape Change for the last design cycle
161. Push the **Filled Contours** radio button
162. Select a Shape Change for the last design cycle in the **Color Mesh** frame
163. Push the **Up** button

Quit Design Studio

164. From the main menu bar, select **File** → **Quit**
165. Push the **Don’t Save** button

6.9 Design a Feature - Using 2-D (QUAD) Domains

Introduction

The purpose of this example is to review and understand how domains and perturbations can be used in designing a feature in a model. The provided input file contains both the design and analysis data required for shape optimization of a plate with a hole. The size, location, and shape of the hole are designed. Also the taper of the plate is also designed.

The following optimization problem will be solved and post-processed:

Minimize Volume

Subject to:

VonMises Stress $\leq$ 100000.0

Designable region:

Design of the hole location, size and shape

Design the taper of the plate

Example ID

`SHDSG009`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG009.dat`	`Input data`	Provided: Contains the finite element mesh and design data for shape optimization of a plate with a hole
`SHDSG009_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains the data similar to the one in `SHDSG009.dat`
`SHDSG009_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SHDSG009_dsg.SHP`	`Shape Change data`	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG009.dat`

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase
5. View the loading and boundary conditions on the plate in the Viewport
6. From the main menu, select **Edit** → **Deselect All** to clear the loadcase selection

Preview the shape changes

7. From the Viewport window, change the view to be the XY plane(Top view)
8. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
9. Select the **Oscillate** radio button
10. Under the **Color Mesh**, select the **Filled Contours** radio button
11. Select each **Shape Morphing Set Preview** under **Color Mesh**

    Review each shape morphing set and how the perturbations are used to design the hole
12. Push the **Up** button

Request to change Maximum Design Cycles

13. From the main menu bar, select **Genesis** → **Options...**
14. Select the **Design Control** tab
15. For **Maximum Design Cycles**, enter `15`
16. Push the **Apply** button

Optimize the Structure Using Genesis

17. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button

Import the Shape Changes File

18. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
19. Select the `SHDSG0009_dsg.SHP` file

20. Push the **Open** button

Post-Processing the Results (Shape Changes)

21. Select the **Post** tab
22. Push the **Deform/Mesh Color Mesh** button
23. Push the **Filled Contours** radio button
24. Select a Shape Change for the last design cycle in the **Color Mesh** frame

    Notice that there are two Shape Change output for each design cycle. This is due to the use of mesh smoothing. For each design cycle, the first one is before mesh smoothing and the second one is after mesh smoothing.
    Observe for a difference between the two outputs in each design cycle to see what mesh smoothing did.

Creating an Animation

25. Select the **Post** tab
26. Push the **Animation** button
27. Select **Shape Change** for the **Deform Result Type**
28. Select **Shape Change** for the **Color Result Type**
29. Push **Next>**
30. Use the Shift key and select all the shape changes for the Deform Mesh Frames
31. Push **Next>**
32. Use the Shift key and select all the shape changes for the Color Mesh Frames
33. Push **Next>**
34. Push the **Save Animation...** button
35. Enter `SHDSG009.gif` for the name
36. Push the **Save** button

    By default, the animation will be saved in the working directory unless the user changes it. Thie GIF file could be used in a powerpoint presentation to show an animation of the shape changes

Quit Design Studio

37. From the main menu bar, select **File** → **Quit**
38. Push the **Don’t Save** button

6.10 Design of a Steering Knuckle - Review of Shape Domains

Introduction

The purpose of this example is to review and understand how domains and perturbations can be used in designing complex shaped structures. The provided input file contains both the design and analysis data required for shape optimization of a steering knuckle.

The following optimization problem will be solved and post-processed:

Minimize Mass

Subject to:

VonMises Stress ≤ 5.0

Designable region:

Shape design of the features of the steering knuckle

Example ID

`SHDSG010`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG010.dat`	`Input data`	Provided: Contains the finite element mesh and design data for shape optimization of a steering knuckle.
`SHDSG010_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains the data similar to the one in `SHDSG010.dat`
`SHDSG010_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SHDSG010_dsg.SHP`	`Shape Change data`	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG010.dat`

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase
5. View the loading and boundary conditions on the beam in the Viewport

Preview the shape changes

6. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
7. Select the **Oscillate** radio button
8. Under the **Deform Mesh**, select each **Shape Morphing Set Preview**

   Review each shape morphing set and how the perturbations are used to design the features in the knuckle
9. Push the **Up** button

Optimize the Structure Using Genesis

10. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button

Import the Shape Changes File

11. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
12. Select the `SHDSG010_dsg.SHP` file
13. Push the **Open** button

Post-Processing the Results (Shape Changes)

14. Select the **Post** tab
15. Push the **Deform/Mesh Color Mesh** button
16. Push the **Filled Contours** radio button
17. Select a Shape Change for the last design cycle in the **Color Mesh** frame

Creating an Animation

18. Select the **Post** tab

19. Push the **Animation** button
20. Select **Shape Change** for the **Deform Result Type**
21. Select **Shape Change** for the **Color Result Type**
22. Push **Next>**
23. Use the Shift key and select all the shape changes for the Deform Mesh Frames
24. Push **Next>**
25. Use the Shift key and select all the shape changes for the Color Mesh Frames
26. Push **Next>**
27. Push the **Save Animation...** button
28. Enter `SHDSG010.gif` for the name
29. Push the **Save** button

    By default, the animation will be saved in the working directory unless the user changes it.

Quit Design Studio

30. From the main menu bar, select **File** → **Quit**
31. Push the **Don’t Save** button

6.11 Using Different Domains to Perform Similar Design

Introduction

The purpose of this example is to review and understand how different domains can be used to achieve the same design. The provided input file contains some of the design and analysis data required for shape optimization of two solid hollow beams. The shape domains are already defined. This example also goes through the creation of multiple objectives and constraints.

The following optimization problem will be solved and post-processed:

Minimize Displacements

Subject to:

Mass $\leq$ 5.0E-6

Designable region:

Wall thickness of the hollow beam

Example ID

`SHDSG011`

Special Features:

In this example, different types of domains are used in two identical beams to obtain similar shape changes. While applying perturbations, one should select only the domains that the perturbations should act upon. In this case when applying perturbations on the 1-D BAR domains, one should select only the BAR domains where the perturbations should act. Also, multiple objectives are defined in this problem.

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG011.dat`	`Input data`	Provided: Contains the finite element mesh and some design data for shape optimization of two hollow beams.

SHDSG011_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains the data similar to the one in `SHDSG011.dat` along with data created in the example
SHDSG011_dsg.SHP	Shape Change data	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization
SHDSG011_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to `SHDSG011_dsg.dat`. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG011.dat`

Review the Shape Domains

3. From the **Design** category chooser, select **Shape Domains**
4. Select one of the existing BAR domains
5. Push the **Modify Domain** button from the Edit Menu toolbar

   Notice that the grids controlled by the BAR domains are selected.
6. Push the **Cancel** button
7. Repeat steps 4 - 6 to study all the other BAR domains

   Notice that the grids controlled by each BAR domain extend along the length of the beam. Also notice that some grids are controlled by multiple BAR domains.
8. Select one of the existing QUAD domains
9. Push the **Modify Domain** button from the Edit Menu toolbar

   Notice that the grids controlled by the QUAD domains are selected.
10. Push the **Cancel** button
11. Repeat steps 8 - 10 to study all the other QUAD domains

    Notice that the grids controlled by each QUAD domain extend along the length of the beam. Also, notice that none of the grids are controlled by multiple domains.

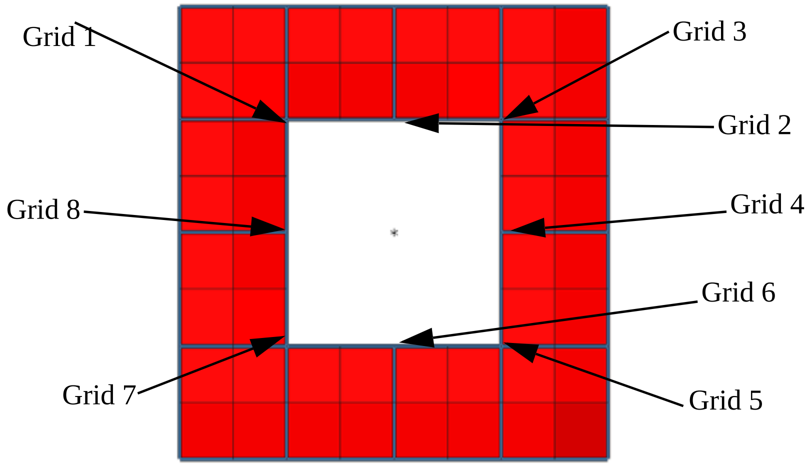
Create Perturbations on QUAD domains

12. From the **Design** category chooser, select **Shape Morphing Sets**
13. Push the **New Shape Set** button from the Edit menu toolbar
14. Enter `Quads` in the Name field
15. Select the **Domain Morphing Set** radio button
16. Push **Next>**

17. Select all the existing QUAD domains from the **Select Domains to Act Upon** list

    Verify that there is "**12 domains selected'"** at the bottom of the window.

18. Push **Next>**
19. From the Viewport window, change the view to be the YZ plane(Front view)
20. Push the **Select None** button
21. From the Viewport, select the one corners of the HEXA domain (Grid 1 in the figure below) at the fixed end of the beam



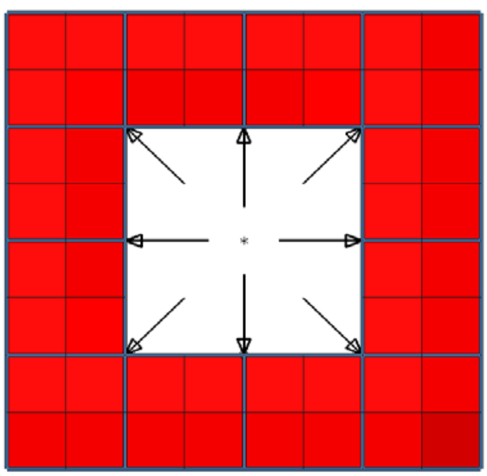
22. Enter `0.0`, `0.7071`, `0.7071` for the **X, Y, Z** to define the direction of the perturbation
23. Enter `1.0` for the **Magnitude**
24. Push the **Add Perturbation** button

    Verify that there is "**1 perturbations on 1 grids"** at the bottom of the window

25. Repeat steps 20 through 24 to apply perturbations at all eight corners of the QUAD domains along the hole.The magnitude and direction of the perturbations are given in the table below

Grid Label	Magnitude	Direction
Grid 1	`1.0`	`0.0, 0.7071, 0.7071`
Grid 2	`0.7071`	`0.0, 0.0, 1.0`
Grid 3	`1.0`	`0.0, -0.7071, 0.7071`
Grid 4	`0.7071`	`0.0, -1.0, 0.0`
Grid 5	`1.0`	`0.0, -0.7071, -0.7071`
Grid 6	`0.7071`	`0.0, 0.0, -1.0`
Grid 7	`1.0`	`0.0, 0.7071, -0.7071`
Grid 8	`0.7071`	`0.0, 1.0, 0.0`

Verify that there is “**8 perturbations on 8 grids”** at the bottom of the window. The applied perturbations will look like the picture below.



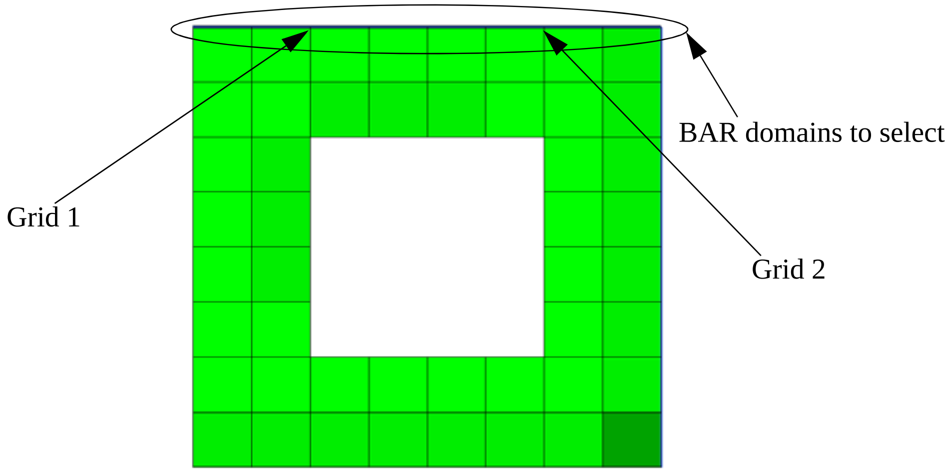
26. Push the **Finish** button

Create Perturbations on BAR domains

27. Push the **New Shape Set** button from the Edit menu toolbar
28. Enter `Bar_1` in the Name field
29. Select the **Domain Morphing Set** radio button

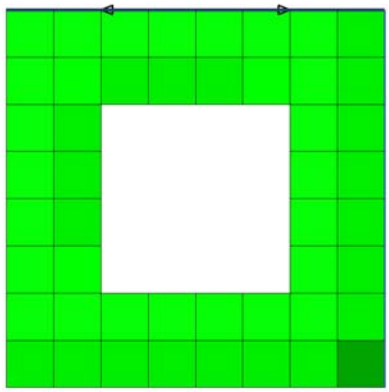
30. Push **Next>**
31. Select four existing BAR domains along the width of the beam (as shown in the figure below) from the **Select Domains to Act Upon** list

    Care should be taken to select only the horizontal BAR elements and not all the BAR elements as the perturbation should act on only these domains. If all the BAR domains are selected, Genesis prompts an error. This is because the grids where the perturbations are applied are also grids controlled by the horizontal BAR domains and are not corner or mid-size grids.



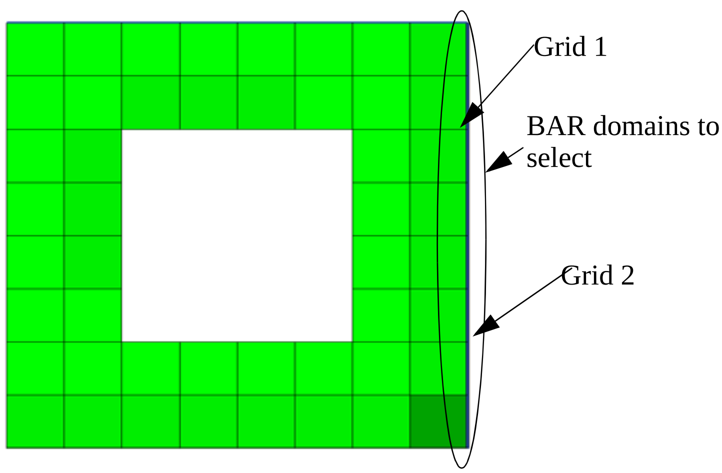
32. Push **Next>**
33. Push the **Select None** button
34. Select one of the grids (Grid 1 shown in the figure above) to apply the perturbation
35. Enter `0.0`, `1.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation
36. Enter `0.7071` for the **Magnitude**
37. Push the **Add Perturbation** button
38. Push the **Select None** button
39. Select the other grid (Grid 2 shown in the figure above) to apply the perturbation
40. Enter `0.0`, `-1.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation
41. Enter `0.7071` for the **Magnitude**
42. Push the **Add Perturbation** button

Verify that there is “**2 perturbations on 2 grids”** at the bottom of the window



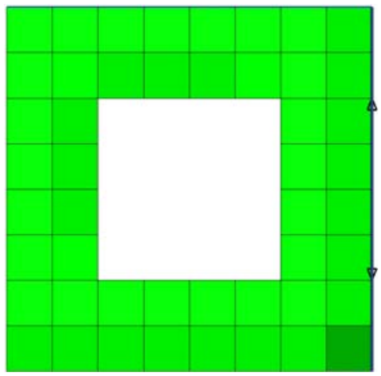
43. Push the **Finish** button
44. Push the **New Shape Set** button from the Edit menu toolbar
45. Enter `Bar_2` in the Name field
46. Select the **Domain Morphing Set** radio button
47. For the **Design Variable**, select `Shape2` (the design variable created while defining the `Bar_1` perturbations)

    The same design variable controls the shape change for both the perturbations resulting in a uniform wall thickness of the beam
48. Push **Next>**
49. Select the four BAR domains along the height of the beam (as shown in the figure below) from the **Select Domains to Act Upon** list



50. Push **Next>**
51. Push the **Select None** button
52. Select one of the grids (Grid 1 shown in the figure above) to apply the perturbation

53. Enter `0.0`, `0.0`, `1.0` for the **X, Y, Z** to define the direction of the perturbation
54. Enter `0.7071` for the **Magnitude**
55. Push the **Add Perturbation** button
56. Select the other grid (Grid 2 shown in the figure above) to apply the perturbation
57. Enter `0.0`, `0.0`, `-1.0` for the **X, Y, Z** to define the direction of the perturbation
58. Enter `0.7071` for the **Magnitude**
59. Push the **Add Perturbation** button
60. Verify that there is "**2 perturbations on 2 grids"** at the bottom of the window



61. Push the **Finish** button

Preview the shape changes

62. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
63. Select the **Oscillate** radio button
64. Under the **Deform Mesh**, select each **Shape Morphing Set Preview**

    Review each shape morphing set and how the perturbations are used to design uniform wall thickness of both the beams
65. Push the **Up** button

Define the Two Displacement Design Objective

66. From the **Design** category chooser, select **Objectives**
67. Push the **New Objective** button from the Edit menu toolbar
68. Enter `Displacement_1` for the Name
69. Select the **Displacement** radio button
70. Push **Next>**

71. Push the **Select None** button
72. From the Viewport, select the grid where the load is applied (middle of the top surface at the free end) on the beam with QUAD domains
73. Select the **Translation Magnitude** radio button for the component
74. Push **Next>**
75. Select the existing loadcase (`STAT ID = 1`)
76. Push **Finish**
77. Push the **New Objective** button from the Edit menu toolbar
78. Enter `Displacement_2` for the Name
79. Select the **Displacement** radio button
80. Push **Next>**
81. Push the **Select None** button
82. From the Viewport, select the grid where the load is applied (middle of the top surface at the free end) on the beam with BAR domains
83. Select the **Translation Magnitude** radio button for the component
84. Push **Next>**
85. Select the existing loadcase (`STAT ID = 2`)
86. Push **Finish**

    While running Genesis, design studio checks whether multiple objective are defined. If so, it would use the DINDEX input format of Genesis to create a objective that is a weighted sum of the individual objectives.

Define the Mass Design Constraint

87. From the **Design** category chooser, select **Constraints**
88. Push the **New Constraint** button from the Edit menu toolbar
89. Enter `Mass` for the Name
90. Select the **Mass** radio button
91. Enter `5E-6` as **Upper Bound**
92. Push **Finish**

Optimize the Structure Using Genesis

93. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button
    Notice that the objective starts from a value of 2.0 as it is the normalized sum of the two individual objectives.

Import the Shape Changes File

94. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
95. Select the `SHDSG011_dsg.SHP` file
96. Push the **Open** button

Post-Processing the Results (Shape Changes)

97. Select the **Post** tab
98. Push the **Deform/Mesh Color Mesh** button
99. Push the **Filled Contours** radio button
100. Select a Shape Change for the last design cycle in the **Color Mesh** frame
101. From the Viewport window, change the view to be the YZ plane(Front view)

     Notice that both the beams have identical wall thickness as both they are identical beams.
102. Push the **Up** button

Quit Design Studio

103. From the main menu bar, select **File** → **Quit**
104. Push the **Don’t Save** button

6.12 Making Copies of Existing Domains - Wheel Example

Introduction

The purpose of this example is to learn how to make copies of existing domains to design similar features in a structure. The provided input file contains the analysis model of a wheel. It also contains a set of domains that are defined along with the perturbations. In this exercise, new domains are created by copying this existing domains set. perturbations are also applied to the copied domains.

The following optimization problem will be solved and post-processed:

Maximize the 9th Natural Frequency

Designable region:

Shape changes in the wheel

Example ID

`SHDSG012`

Special Features:

In this example, multiple domains are created by making copies of an existing set of domains. The perturbations in all the domains are controlled by the same design variable to maintain the symmetry of the structure.

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG012.dat`	`Input data`	Provided: Contains the finite element mesh and some design data for shape optimization of wheel.
`SHDSG012_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains the data similar to the one in `SHDSG012.dat` along with data created in the example
`SHDSG012_dsg.SHP`	`Shape Change data`	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization

SHDSG012_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SHDSG012_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG012.dat`

Review existing Shape Domains

3. From the **Design** category chooser, select **Shape Domains**
4. Study the existing HEXA domains

   12 HEXA domains are created to design the shape of the shell.

Review existing Shape Morphing Set

5. From the **Design** category chooser, select **Shape Morphing Sets**
6. Select the first morphing set
7. Push the **Modify Shape Set** button from the Edit Menu toolbar
8. Study the perturbations applied on the domains. Please verify that there are four perturbations applied.
9. Push the **Cancel** button.
10. Repeat the steps above to review the second morphing set.

Preview the associated shape changes

Two morphing sets are defined to that change the shape of the shell in the Y-direction.

11. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
12. Select the **Oscillate** radio button
13. Under the **Color Mesh**, select **Filled Contours** radio button
14. Under the **Color Mesh**, select each **Shape Morphing Set Preview**

    Review each shape morphing set and how the perturbations are used to design the shape of the shell
15. Push the **Up** button

Create Copies of Existing Domains

In this step, the existing domains on one of the spokes is used to make copies on the other four spokes along the perimeter of the wheel.

16. From the **Design** category chooser, select **Shape Domains**
17. Push the **New Domain** button from the Edit menu toolbar
18. Select the **Duplicate Selected Domains (use new rotated grids)** radio button

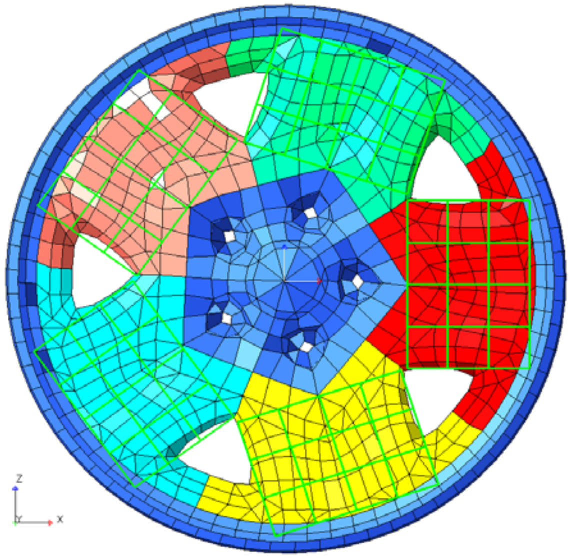
    When domains are copied, the new domains are created with newly defined grids. While applying perturbations care should be taken to select the domain grids. If there are existing grids in the same location as the newly created grid one might need to merge the coincident grids using **Analysis** → **Grids** → **Select All** → **Merge Coincident**

19. Push **Next>**
20. Select all the existing HEXA domains from the **Select Domains to Copy** list

    Verify that there is “**12 domains selected‘”** at the bottom of the window.

21. Push **Next>**
22. Enter `4` for the **Number of copies**
23. Push the **Change** button to change the **Ref. Coord System**
24. Select the `Center` coordinate system from the **Choose Coordinate System** list
25. Push **Next>**
26. For the Rotational Axis, Select **Y**
27. Enter `72` for the **Angle Increment**
28. Push **Next>**
29. Select the existing group from the **Choose Group for New Domains** list
30. Push the **Finish** button

    Notice that this operation has created four sets of HEXA domains covering the other shells in the wheel as shown in the figure below.



Modify existing Morphing Sets

31. From the **Design** category chooser, select **Shape Morphing Sets**
32. Select the `Top` Morphing Set
33. Push the **Modify Shape Set** button from the Edit Menu toolbar
34. Push **Next>**
35. Select all the existing HEXA domains from the **Select Domains to Act Upon** list

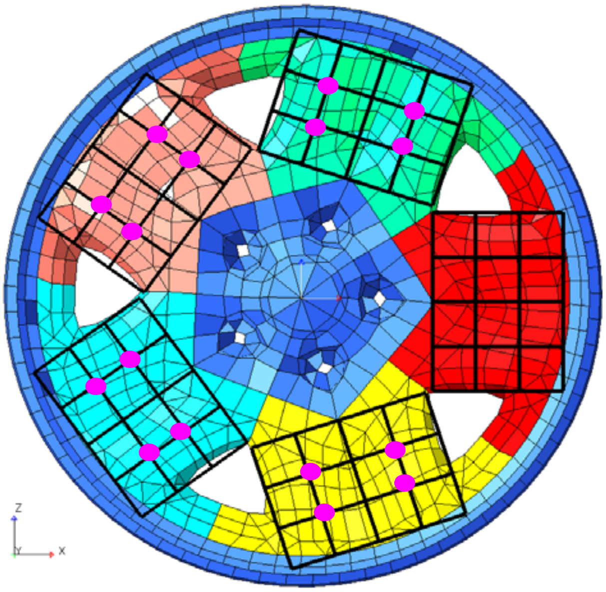
    Verify that there is “**60 domains selected‘”** at the bottom of the window.

36. Push **Next>**

    Notice that “**4 perturbations on 4 grids**” exists at the bottom of the window because perturbations are already applied on one of the domain sets

37. From the Viewport window, change the view to be the ZX plane(Left view)
38. Push the **Select None** button

39. Select the 16 grids from the Viewport as shown in the figure below



Grids to Select

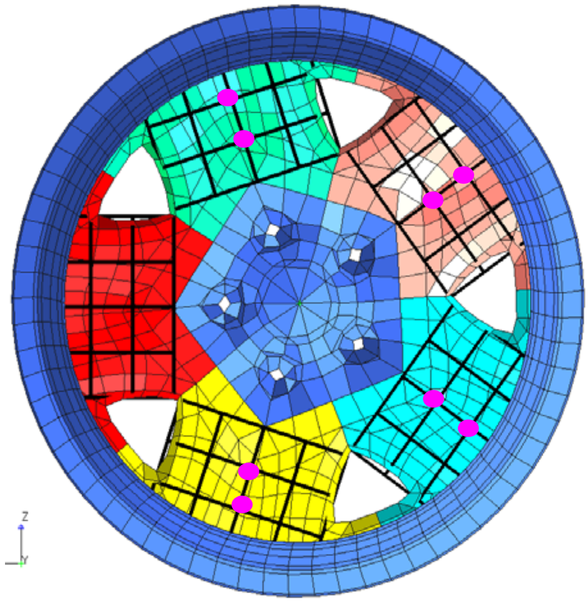
40. Enter `0.0, -1.0, 0.0` for the **X, Y, Z** to define the direction of the perturbation
41. Enter `20.0` for the **Magnitude**
42. Push the **Add Perturbation** button

    Verify that there is “**20 perturbations on 20 grids”** at the bottom of the window
43. Push the **Finish** button
44. Select the `Bottom` Morphing Set
45. Push the **Modify Shape Set** button from the Edit Menu toolbar
46. Push **Next**>
47. Select all the existing HEXA domains from the **Select Domains to Act Upon** list

    Verify that there is “**60 domains selected‘”** at the bottom of the window.
48. Push **Next**>

    Notice that “**2 perturbations on 2 grids**” exists at the bottom of the window because perturbations are already applied on one of the domain sets
49. From the Viewport window, change the view to be the XZ plane(Right view)
50. Push the **Select None** button

51. Select the 8 grids from the Viewport as shown in the figure below



● Grids to Select

52. Enter `0.0`, `1.0`, `0.0` for the **X, Y, Z** to define the direction of the perturbation
53. Enter `10.0` for the **Magnitude**
54. Push the **Add Perturbation** button

    Verify that there is "**10 perturbations on 10 grids"** at the bottom of the window

55. Push the **Finish** button

Preview the shape changes

56. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
57. Select the **Oscillate** radio button
58. Under the **Deform Mesh**, select each **Shape Morphing Set Preview**

    Notice that in each of the morphing sets, as the perturbations in each of the domain sets are controlled by the same design variable, the shape change is the same along all the members.

59. Push the **Up** button

Define the Design Objective

60. From the **Design** category chooser, select **Objectives**
61. Push the **New Objective** button from the Edit menu toolbar
62. Enter `9th_Frequency` for the Name

63. Select the **Frequency Mode Number** radio button
64. Enter `9` in the adjacent textbox
65. Select **Max** for the **Objective Definition Switch**
66. Push **Next>**
67. Select the existing loadcase (`FREQ ID = 2`)
68. Push **Finish**

Optimize the Structure Using Genesis

69. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History** charts; when done, push the **Close** button
    Study the **Genesis Console Output**; when done, push the **Close** button

Import the Shape Changes File

70. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
71. Select the `SHDSG012_dsg.SHP` file
72. Push the **Open** button

Post-Processing the Results (Shape Changes)

73. Select the **Post** tab
74. Push the **Deform/Mesh Color Mesh** button
75. Push the **Filled Contours** radio button
76. Select a Shape Change for the last design cycle in the **Color Mesh** frame
77. Push the **Up** button

Quit Design Studio

78. From the main menu bar, select **File** → **Quit**
79. Push the **Don’t Save** button

6.13 Design Location of Member - Using Synthetic Responses

Introduction

The purpose of this example is to review and understand how shape optimization can be used to design the location of a member within the structure. The provided input file contains all the necessary data. Synthetic responses are used to define the objective as well as some constraints for the optimization.

The following problem will be solved:

Maximize (3*Freq 7 + Freq 8 + Freq 9 + Freq 10)/6

Subject to:

Freq 7 ≥ 6.2 Hz (standard constraint)
Freq 8 ≥ 9.0 Hz (standard constraint)
Freq 9 ≥ 15.0 Hz (standard constraint)
Freq 10 ≥ 19.0 Hz (standard constraint)
Freq 8 - Freq 7 ≥ 2.0 Hz (synthetic constraint, to separate frequencies)
Freq 9 - Freq 8 ≥ 2.0 Hz (synthetic constraint, to separate frequencies)
Freq 10 - Freq 9 ≥ 2.0 Hz (synthetic constraint, to separate frequencies)

Example ID

SHDSG013

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SHDSG013.dat	Input data	Provided: Contains the finite element mesh and design data for shape optimization
SHDSG013_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains the data similar to the one in SHDSG013.dat
SHDSG013_dsg.SHP	Shape Change data	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization

Start Design Studio

1. Load the Genesis data file: `SHDSG013.dat`

   This file has all the data needed for the optimization.

Preview the applied shape changes

2. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
3. Select the **Oscillate** radio button
4. Under the **Color Mesh**, select **Filled Contours** radio button
5. Under the **Color Mesh**, select each **Shape Morphing Set Preview**

   Notice how each of the morphing sets is used to define the shape/location of the members in the frame.
6. Push the **Up** button

Optimize the Structure Using Genesis

7. From the main menu bar, select **Genesis** → **Optimize**

   Study the **Design History**, when done push the **Close** button
   Study the **Genesis Console Output**, when done push the **Close** button
   Open the output file. Study the values of the objective and constraints and complete the following table:

Type	Function definition	Reference Final Solution	Initial Value	Final Value	Gain%
Objective	(3*Freq7+Freq8+Freq9+Freq10)/6	12.289			
Constraint 1	Frequency 7	6.229			
Constraint 2	Frequency 8	11.140			
Constraint 3	Frequency 9	15.680			
Constraint 4	Frequency 10	28.225			
Constraint 5	Freq 8 - Freq 7	4.911			
Constraint 6	Freq 9 - Freq 8	4.539			
Constraint 7	Freq 10 - Freq 9	12.545			

Import the Shape Changes File

8. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**

9. Select the `SHDSG013_dsg.SHP` file
10. Push the **Open** button

Post-Processing the Results (Shape Changes)

11. Select the **Post** tab
12. Push the **Deform/Mesh Color Mesh** button
13. Push the **Filled Contours** radio button
14. Select a Shape Change for the last design cycle in the **Color Mesh** frame
15. Push the **Up** button

Quit Design Studio

16. From the main menu bar, select **File** → **Quit**
17. Push the **Don’t Save** button
18. Optional studies: Import into Design Studio the `SHDSG013_dsgxx.pch` files to study the mode shapes.

6.14 Design of a Solid Cantilever Beam using DOMAINs

Introduction

The purpose of this exercise is to setup a shape optimization problem.

Problem Statement:

Minimize the mass of a cantilevered beam subject to three loadcases consisting of tip shear, tip moment and end twisting moment.

Minimize mass; 9 shape design variables; 1 stress constraint; 2 geometric constraints; 1 frequency constraint

In this problem, you will create 4 of the 9 shape perturbations. All loadcase and boundary condition data is provided.

This optimization is done using perturbation vectors. Three HEXA domains are used to define 9 basis vectors. The first domain covers the entire beam.

Four perturbations are generated with this domain. The first two are linear tapering in the y- and z- directions, respectively. The next two allow quadratic variations in shape along the entire length of the beam. The other two domains each cover half of the beam. The last 5 basis vectors are generated using these two domains.

Example ID

`SHDSG014`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG014.dat`	`Input data`	Provided: Contains the finite element mesh of the structure
`SHDSG014_ref.dat`	`Input data`	Provided: Reference input file ready to be optimized
`SHDSG014_dsgxx.pch`	`DSG file`	Created: Punch file containing the temperature results for each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG014.dat`

Check the Loadcases

3. From the **Analysis** category chooser, select **Loadcases**
4. Select each loadcase and study it in the Viewport

Define the Design Objective

5. From the **Design** category chooser, select **Objectives**
6. Push the **New Objective** button from the Edit menu toolbar
7. Enter `MASS` in the name field
8. Select the **Mass** radio button as Response Type
9. Check the **Min** radio button as Objective Definition Switch
10. Push the **Finish** button

Define the Grid Stress Constraints

11. From the **Design** category chooser, select **Constraints**
12. Push the **New Constraint** button from the Edit menu toolbar
13. Enter `MISES` in the **Name** field
14. Select the **More Response Type** radio button as the Response Type
15. Enter `260.0` as **Upper Bound**
16. Push **Next>**
17. Select the **Grid Stress** radio button in Additional Responses
18. Push **Next>**
19. Push the **Select None** button

    We want to apply the constraints to a grid set already created. We select the grid set then we will come back to this **Constraint** menu. Do not push **Cancel** at this point.

20. From the **Analysis** category chooser, select **Grid Sets**
21. Select `Grid Set Constraint` from the list
22. Select the **Design** tab
23. Push **Next>**
24. Select the three static loadcases
25. Push the **Finish** button

26. Right click on the Viewport window and select **Clear** → **All**

Define the Length Constraints

27. From the **Design** category chooser, select **Constraints**
28. Push the **New Constraint** button from the Edit menu toolbar
29. Enter `TIP-Y` in the name field
30. Select the **More Response Type** radio button as the Response Type
31. Enter `10.0` as **Lower Bound**
32. Push **Next>**
33. Select the **Geometric** radio button in Additional Responses
34. Push **Next>**
35. Enter `130`, `132` in the **Select by Grid ID** field
36. Push the **Enter** button on the keyboard
37. Select the **Length** radio button in Geometric Response
38. Push the **Finish** button
39. Right click on the Viewport window and select **Clear** → **All**
40. Push the **New Constraint** button from the Edit menu toolbar
41. Enter `TIP-Z` in the name field
42. Select the **More Response Type** radio button as the Response Type
43. Enter `10.0` as **Lower Bound**
44. Push **Next>**
45. Select the **Geometric** radio button in Additional Responses
46. Push **Next>**
47. Enter `132`, `124` in the **Select by Grid ID** field
48. Push the **Enter** button on the keyboard
49. Select the **Length** radio button in Geometric Response
50. Push the **Finish** button
51. Right click on the Viewport window and select **Clear** → **All**

Define the Frequency Mode Number Constraint

52. From the **Design** category chooser, select **Constraints**

53. Push the **New Constraint** button from the Edit menu toolbar
54. Enter `FREQ4` in the name field
55. Select the **Frequency Mode Number** radio button as the Response Type
56. Enter `4` in the **Frequency Mode Number** field
57. Enter `700.0` as **Lower Bound**
58. Push **Next>**
59. Select the frequency loadcases named `MODES`
60. Push the **Finish** button

Check Independent Design Variables for the Perturbation Vectors

61. From the **Design** category chooser, select **Design Variables**
62. Select each independent design variable and study it quickly

Create a Hexa Shape Domain

63. From the **Design** category chooser, select **Shape Domains**
64. Push the **New Domain** button from the Edit menu toolbar
65. Enter `Domain 11` in the name field
66. Check the **Define A New Original Domain** radio button
67. Push **Next**>
68. Select `Create New Domain Group` item
69. Push **Next>**
70. Select the **Hexa** radio button
71. Push **Next>**
72. Enter `1` in the select by Grid ID field
73. Push the Enter key from your keyboard

    Repeat steps 72 and 73 respectively to select the following grids `3`, `12`, `10`, `132`, `124`, `128` and `130`.

74. Push **Next>**
75. Enter `133` in the **Select by Grid ID** field
76. Push the **Add** button

    Repeat steps 75 and 76 respectively to select the following grids `134`, `135`, `136`, `137` and `138`.

77. Push the **Invert** button
78. Push the **Finish** button

Create a Perturbation associated to the set of Domains

79. From the **Design** category chooser, select **Shape Morphing Sets**
80. Push the **New Shape Set** button from the Edit menu toolbar
81. Enter `TAPER-Y` in the name field
82. Select the **Domain Morphing Set** radio button
83. Select `TAPER-Y` in the **Design Variable** category chooser
84. Push **Next>**
85. Select `Domain 11` from the **Select Act Upon** list.
86. Push **Next>**
87. Push the **Select None** button
88. Enter `124` in the **Select by Grid ID** field
89. Push the **Add** button
90. Enter `132` in the **Select by Grid ID** field
91. Push the **Add** button

    Verify that there are 2 grids selected. The perturbation will be applied on those 2 grids.
92. Push the **By 2 Grids...** button
93. Enter `124`, `128`

    This defines the direction of the perturbation from grid 124 to grid 128
94. Push the **Enter** button on the keyboard
95. Push **Next>**
96. Enter `1.0` for the **Magnitude**
97. Push the **Add Perturbation** button

    Verify that there is "**2 perturbations on 2 grids"** at the bottom of the window
98. Push the **Select None** button
99. Enter `128` in the **Select by Grid ID** field
100. Push the **Add** button
101. Enter `130` in the **Select by Grid ID** field
102. Push the **Add** button

Verify that there are 2 grids selected. The perturbation will be applied on those 2 grids.

103. Push the **Enter** button on the keyboard
104. Push the **By 2 Grids...** button
105. Enter `128, 124`

This defines the direction of the perturbation from grid 128 to grid 124

106. Push **Next>**
107. Enter `1.0` for the **Magnitude**
108. Push the **Add Perturbation** button

Verify that there is “**4 perturbations on 4 grids”** at the bottom of the window

109. Push the **Finish** button
110. Right click on the Viewport window and select **Clear** → **All**

Repeat steps 80 to 110 to create the 3 other perturbation vectors with the following table:

Shape Morphing Set Name	Associated Design Variable Name (*)	Domains to Act Upon	Perturbated Grid ID	Direction of the perturbation
TAPER-Z	TAPER-Z	Domain 11	124 128	-Z
		Domain 11	132 130	Z
CURVE-Y	CURVE-Y	Domain 11	64 72	Y
		Domain 11	68 70	-Y
CURVE-Z	CURVE-Z	Domain 11	64 68	-Z
		Domain 11	72 70	Z

Set the Analysis Print (APRINT) to Output all the Design Cycle Responses

111. From the **Genesis** menu, select **Options...**
112. Select the **Output Control** tab
113. Check the **Analysis Output** check box, and select **All Cycles** from the pull down menu
114. Push the **Apply** button

Save the Design Studio file

115. From the main menu bar, select **File** → **Save As...**
116. Enter `SHDSG014` as the Filename and push **Save (**as a Design Studio File)

Optimize the structure using Genesis

117. From the main menu bar, select **Genesis** →**Optimize**
118. Study the **Genesis Console Output** window

Import the Post Processing Files

119. From the **Genesis Console Output** window, select the **Import Post...** button
120. Select `SHDSG014_dsg.SHP`
121. Use the Shift/Ctrl key to select all the `SHDSG014_dsgxx.pch`, files where `xx` is the design cycle number
122. Push the **Import** button
123. From the **Genesis Console Output** window, select the **Close** button

Postprocessing the Results

124. Select the **Post** tab

     You can view individual frames with **Deform Mesh/Color Mesh** or you can make an animation with **Animation**. You can also animate shape changes and element results (e.g. stresses) simultaneously
125. Push **Deform Mesh/Color Mesh...**
126. In the **Deform Mesh** window, select **Ramp** (try also **Static** and **Oscillate**)
127. In the **Color Mesh** window, select **Filled Contours** radio button
128. Select each proposed Cycle to check the Grid Stress
129. Push **Up** when finished
130. From the **Post** tab, push the **Animation** button
131. Select **Shape Change** from the **Deform Result Type** menu
132. Select **Solid Stress/Strain** from the **Color Result Type** menu
133. Check the **Filled Contours** radio button
134. Push **Next>**

135. Select all the Cycles for Shape Change from the list
136. Push **Next>**
137. Select all the Cycles for Loadcase 1 Grid Stress from the list
138. Push **Next>**
139. Push **Finish**

Quit Design Studio

140. From the main menu bar, select **File** →**Quit**
141. Push the **Don’t Save** button

6.15 Design of a Solid Cantilever Beam using Natural Perturbation Vectors

Introduction

The purpose of this exercise is to setup natural perturbation vectors in a shape optimization.

This exercise is divided into two parts:

In the first part you will learn how to generate natural perturbation vectors.

In the second part you will learn how to perform a shape optimization using these natural perturbation vectors.

Design Problem:

Minimize mass; 12 shape design variables; 1 stress constraint; 2 geometric constraints; 1 frequency constraint

Example ID

SHDSG015

Special Features:

In this example, the displacements of the structure are used to create natural perturbations vectors for shape optimization.

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SHDSG015_1.dat	Input data	Provided: Contains the finite element mesh of the structure
SHDSG015_2.dat	Input data	Provided: Contains the FE mesh as along with the loading for optimization.
SHDSG015_ref1.dat	Input data	Provided: Reference input file ready to be analyzed for Part 1
SHDSG015_ref2.dat	Input data	Provided: Reference input file ready to be optimized for part 2

6.15.1 Part 1

Introduction

The purpose of this part is to learn how to generate natural perturbation vectors.

Problem Statement:

Create natural perturbation vectors for the shape design of a beam.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG015_1.dat`

Create Element Pressure Static Loads

3. From the **Analysis** category chooser, select **Static Loads**
4. Push the **New Load Set** button from the Edit menu toolbar
5. Enter `Load Set 4` in the name field
6. Check the **Force, Moment, Pressure, SPCD** radio button
7. Push **Next>**
8. Push **Next>**
9. Push the **Select None** button
10. Enter a `110` in the **Select by Element ID** field
11. Push the **Add** button
12. Enter `-5.0` in the **Pressure** field
13. Push the **Add Pressure** button
14. Push the **Finish** button

Repeat steps 4 to 14 to create 7 other Static Loads using the following table

Static Load Name	ElementID loaded	Pressure
Load Set 5	125	-5.0
Load Set 6	195	-5.0
Load Set 7	180	-5.0

Load Set 8	135 145	-5.0
Load Set 9	140 150	-5.0
Load Set 10	160 170	-5.0
Load Set 11	155 165	-5.0

Create Grid Force Static Loads

15. From the **Analysis** category chooser, select **Static Loads**
16. Push the **New Load Set** button from the Edit menu toolbar
17. Enter `Load Set 12` in the name field
18. Check the **Force, Moment, Pressure, SPCD** radio button
19. Push **Next>**
20. Push the **Select None** button
21. Enter a `123`, `124`, `126`, `128` in the **Select by Grid ID** field
22. Push the **Add** button
23. Enter `-1.0` in the Z field

    It means the direction of the force is in the -Z direction
24. Enter `20.0` in the **Magnitude** field
25. Push the **Add Force** button
26. Push the **Finish** button

    Repeat these steps to create 3 other Static Loads using the following table

Static Load Name	ElementID loaded	Force Magnitude	Direction
Load Set 13	121 124 132	20.0	Y
Load Set 14	127 128 130	20.0	-Y

Load Set 15	129 130 131 132	20.0	Z

Create a Static Loadcase

27. From the **Analysis** category chooser, select **Loadcases**
28. Push the **New Loadcase** button from the Edit menu toolbar
29. Enter `LOAD4` in the name field
30. Check the **Static** radio button
31. Push **Next>**
32. From the **SPC** category chooser, select `2 SPC Set 2`
33. Push **Next>**
34. From the **Load Set** category chooser, select `Load Set 4`
35. Push **Next>**
36. From the first **Displacement** category chooser, select **Post**
37. Push the **Finish** button

Repeat these steps to create 11 other loadcases using the following table

Loadcase Name	SPC	Static Loads
TOP2	3 SPC Set 3	Load Set 5
BOT1	4 SPC Set 4	Load Set 6
BOT2	5 SPC Set 5	Load Set 7
SIDE1	6 SPC Set 6	Load Set 8
SIDE2	7 SPC Set 7	Load Set 9
SIDE3	8 SPC Set 8	Load Set 10
SIDE4	9 SPC Set 9	Load Set 11
TOP	10 SPC Set 10	Load Set 12
SIDE	11 SPC Set 11	Load Set 13
SIDE5	11 SPC Set 11	Load Set 14
BOT3	10 SPC Set 10	Load Set 15

Analyze the structure using Genesis

38. From the main menu bar, select **Genesis** →**Single Analysis**
39. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

40. From the main menu bar, select **File** →**Import** →**Punch/Output2 Results...**
41. Select the `SHDSG015_1_dsg00.pch` file
42. Push the **Open** button

Check the Displacement Results

43. Select the **Post** tab
44. Push the **Deform Mesh/Color Mesh**
45. In the **Color Mesh** window, select **Filled Contours**
46. Select each loadcase from the **Color Mesh** window and study it
47. Push **Up** when finished

Quit Design Studio

48. From the main menu bar, select **File** →**Quit**
49. Push the **Don’t Save** button

6.15.2 Part 2

Introduction

The purpose of this part is to perform a shape optimization using the natural perturbation vectors created in Part 1.

Genesis example ID:

D063

Problem Statement:

Minimize the mass of a cantilevered beam subject to three loadcases consisting of tip shear, tip moment and end torque.

Start Design Studio

1. Start Design Studio
2. Import the Genesis input file: `SHDSG015_2.dat`

Import the Post-Processing Files

3. From the main menu bar, select **File** →**Import** →**Punch/Output2 Results...**
4. Select the `SHDSG015_1_dsg00.pch` file

   If you do not see this file, change the files of type to **All Files**.
   If you do not have this file, run a single analysis with the file named `SHDSG015_ref1.dat`

5. Push the **Open** button

Create a Natural Perturbation from the displacement result

6. From the **Design** category chooser, select **Shape Morphing Sets**
7. Push the **New Shape Set** button from the Edit menu toolbar
8. Enter `Raw Shape 10` in the name field
9. Select the **Raw Morphing Set** radio button
10. Push **Next>**
11. Push the **Select None** button

    We want to apply the perturbation vector to a grid set already created. We select the grid set then we will come back to this **Shape Morphing Sets** menu. Do not push **Cancel** at this point.

12. From the **Analysis** category chooser, select **Grid Sets**
13. Select `Grid Set 5_6` from the list
14. Select the **Design** tab
15. Push the **Convert Deform Result...** button
16. Select `Cycle 0 Loadcase 2 Displacement` from the deform results list
17. Enter `10894` in the **Scale Factor** field
18. Push **Next>**
19. Push the **Finish** button

Repeat these steps to create 11 other Loadcases Sets using the following table

Shape Morphing Set Name	Grid Set to be perturbated	Loadcase	Scale Factor
Raw Shape 20	Grid Set 5_6	Loadcase 3	10894
Raw Shape 30	Grid Set 7_8	Loadcase 4	10894
Raw Shape 40	Grid Set 7_8	Loadcase 5	10894
Raw Shape 50	Grid Set 9_10	Loadcase 6	7343
Raw Shape 60	Grid Set 9_10	Loadcase 7	7343
Raw Shape 70	Grid Set 11_12	Loadcase 8	7343
Raw Shape 80	Grid Set 11_12	Loadcase 9	7343
Raw Shape 90	Grid Set 13	Loadcase 10	466887
Raw Shape 100	Grid Set 14	Loadcase 11	1043136
Raw Shape 110	Grid Set 15	Loadcase 12	1043136
Raw Shape 120	Grid Set 16	Loadcase 13	466887

Define the Design Objective

20. From the **Design** category chooser, select **Objectives**
21. Push the **New Objective** button from the Edit menu toolbar
22. Enter `MASS` in the name field
23. Select the **Mass** radio button as Response Type
24. Check the **Min** radio button as Objective Definition Switch

25. Push the **Finish** button

Define the Grid Stress Constraints

26. From the **Design** category chooser, select **Constraints**
27. Push the **New Constraint** button from the Edit menu toolbar
28. Enter `MISES` in the name field
29. Select the **More Response Type** radio button as the Response Type
30. Enter `260.0` as **Upper Bound**
31. Push **Next>**
32. Select the **Grid Stress** radio button in Additional Responses
33. Push **Next>**
34. Push the **Select None** button

    We want to apply the perturbation vector to a grid set already created. We select the grid set then we will come back to this **Constraint** menu. Do not push **Cancel** at this point.

35. From the **Analysis** category chooser, select **Grid Sets**
36. Select `Grid Set Constraint` from the list
37. Select the **Design** tab
38. Push **Next>**
39. Select the three static loadcases
40. Push the **Finish** button
41. Right click on the Viewport window and select **Clear** → **All**

Define the Length Constraints

42. From the **Design** category chooser, select **Constraints**
43. Push the **New Constraint** button from the Edit menu toolbar
44. Enter `TIP-Y` in the name field
45. Select the **More Response Type** radio button as the Response Type
46. Enter `12.0` as **Lower Bound**
47. Push **Next>**
48. Select the **Geometric** radio button in Additional Responses
49. Push **Next>**

50. Enter a `130` in the **Select by Grid ID** field
51. Push the Enter key from your keyboard
52. Enter a `132` in the **Select by Grid ID** field
53. Push the Enter key from your keyboard
54. Select the **Length** radio button in Geometric Response
55. Push the **Finish** button
56. Right click on the Viewport window and select **Clear** → **All**
57. Push the **New Constraint** button from the Edit menu toolbar
58. Enter `TIP-Z` in the name field
59. Select the **More Response Type** radio button as the Response Type
60. Enter `12.0` as **Lower Bound**
61. Push **Next>**
62. Select the **Geometric** radio button in Additional Responses
63. Push **Next>**
64. Enter a `132` in the **Select by Grid ID** field
65. Push the Enter key from your keyboard
66. Enter a `124` in the **Select by Grid ID** field
67. Push the Enter key from your keyboard
68. Select the **Length** radio button in Geometric Response
69. Push the **Finish** button

Define the Frequency Mode Number Constraints

70. From the **Design** category chooser, select **Constraints**
71. Push the **New Constraint** button from the Edit menu toolbar
72. Enter `FREQ4` in the name field
73. Select the **Frequency Mode Number** radio button as the Response Type
74. Enter `4` in the Frequency Mode Number field
75. Enter `700.0` as **Lower Bound**
76. Push **Next>**
77. Select the frequency loadcases named `MODES`
78. Push the **Finish** button

Set the Analysis Print (APRINT) to Output all the Design Cycle Responses

79. From the **Genesis** menu, select **Options...**
80. Select the **Output Control** tab
81. Check the **Analysis Output** check box, and select **All Cycles** from the pull down menu
82. Push the **Apply** button

Optimize the structure using Genesis

83. From the main menu bar, select **Genesis** →**Optimize**
84. Study the **Genesis Console Output** window

Import the Post Processing Files

85. From the **Genesis Console Output** window, select the **Import Post...** button
86. Select `SHDSG015_2_dsg.SHP`
87. Use the Shift/Ctrl key to select all the `SHDSG015_2_dsgxx.pch`, files where `xx` is the design cycle number
88. Push the **Import** button
89. From the **Genesis Console Output** window, select the **Close** button

Postprocessing the Results

90. Select the **Post** tab

    You can view individual frames with **Deform Mesh/Color Mesh** or you can make an animation with **Animation**. You can also animate shape changes and element results (e.g. stresses) simultaneously

91. Push **Deform Mesh/Color Mesh...**
92. In the **Deform Mesh** window, select **Ramp** (try also **Static** and **Oscillate**)
93. In the **Color Mesh** window, select **Filled Contour**
94. Select each proposed Cycle to check the Grid Stress
95. Push **Up** when finished
96. From the **Post** tab, push the **Animation** button
97. Select **Shape Change** from the **Deform Results Type** menu

98. Select **Solid Stress/Strain** from the **Color Results Type** menu
99. Check the **Filled Contours** radio button
100. Push **Next>**
101. Select all the Cycles for Shape Change from the list
102. Push **Next>**
103. Select all the Cycles for Loadcase 1 Grid Stress from the list
104. Push **Next>**
105. Push the **Finish** button

Quit Design Studio

106. From the main menu bar, select **File** →**Quit**
107. Push the **Don’t Save** button

6.16 Using Hexa Domains

Introduction

The main purpose of this exercise is to create a finite element mesh and set up basic shape optimization data.

The following problem is solved:

Minimize Mass

Subject to:

Von mises Stress $\leq$ 330 MPa

Deflection $\leq$ 0.2mm

The problem is divided into four parts that will help you to learn the following tasks:

1) How to create a simple mesh with Design studio

2) How to create a Hexa Shape Domain and a Shape Morphing Set in order to perform a Shape optimization

3)How to create and export a file with CAD geometry that represents the results of the Shape optimization

4)How to create and to export an updated input data file which contains the new grids location of the optimized structure

Example ID

SHDSG016

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SHDSG016_ref1.dat	Input data	Provided: Contains the finite element mesh of the structure, and the loadcase
SHDSG016_ref2.dat	Input data	Provided: Contains the finite element mesh of the structure and the sets of domain
SHDSG016_ref3.dat	Input data	Provided: Contains the updated file resulting from the shape optimization

Part 1

The purpose of this part is to learn how to create a simple mesh with Design studio and create a Loadcase. If you are familiar with this task, you can skip this part and go to Part 2.

When you finish this example, you should have created a file named: `SHDSG016.dat`

Start Design Studio

1. Start Design Studio

Create a Meshed Structure

First you need to define the material properties.

2. Select the **Analysis** tab
3. From the category chooser, select **Materials**
4. Push the **New Material** button from the Edit menu toolbar
5. Check the **Isotropic** radio button
6. Push **Next**>
7. For the Young modulus **E**, enter `210000`
8. For the Poisson coefficient **Nu**, enter `0.28`
9. For the density **Rho**, enter `7.8E-9`
10. Push the **Finish** button

Now, you need to define the group properties.

11. From the **Analysis** category chooser, select **Group Properties**
12. Push the **New Group Property** button from the Edit menu toolbar
13. From the **Type** category chooser, select PSOLID
14. For Name, enter `Beam`
15. Push **Next**>
16. Check that the Material you have just created is selected
17. Push the **Finish** button

Now you will create a meshed beam with the following features: 10*4*80

18. From the category chooser, select **Elements**
19. Select the **Iso Right-Front-Top** view icon in the Viewport

20. Push the **New Elements** button from the Edit menu toolbar
21. Check the **Define New Original Elements** radio button
22. Push **Next**>
23. Select the PSOLID group from the list
24. Push **Next**>
25. Select the **Point by Point** radio button
26. Select the **Hexas** icon as **Region Definition option**
27. Enter `0.0` for **X**, `0.0` for **Y** and `0.0` for **Z**
28. Push the Enter key from your keyboard

Repeat steps 27 and 28 to define the other corners of the Hexas in order of the following table

Corner	X	Y	Z
2	0.0	0.0	10.0
3	0.0	4.0	10.0
4	0.0	4.0	0.0
5	80.0	0.0	0.0
6	80.0	0.0	10.0
7	80.0	4.0	10.0
8	80.0	4.0	0.0

29. In the list, select the Hexa you just created
30. Push the **Subdivide** button
31. Enter `10` as Elements in 1st Dimension
32. Enter `4` as Elements in 2nd Dimension
33. Enter `80` as Elements in 3rd Dimension

The first dimension is displayed by a red line on the shape domain in the Viewport window.
The second dimension is displayed by a green line.
The third dimension is displayed by a blue line.

34. Push **Next>**
35. Push the **Finish** button

Create a Grid-Component-Set

36. From the **Analysis** category chooser, select **Grid-Component-Sets**
37. Push the **New Grid-Component Set** button from the Edit menu toolbar
38. Check the **Single-Point Constraints** radio button
39. Push **Next>**
40. Select the **Right** view (XZ plane) icon in the Viewport window to change the view
41. Select the first left vertical column of grids

    You should have 55 grids selected
42. For **Components**, enter `123`
43. Push the **Set Components** button

    You should have 165 total dof on 55 grids
44. Push the **Finish** button

Create a Static Load

45. From the **Analysis** category chooser, select **Static Loads**
46. Push the **New Load Set** button from the Edit menu toolbar
47. Check the **Force, Moment, Pressure,SPCD** radio button
48. Select the **Iso Right-Front-Top** view icon in the Viewport window to change the view
49. Push **Next>**
50. Push the **Select None** button
51. Enter `33` in the **Select by Grid ID** field
52. Push the **Add** button
53. Enter `0.0` as **X** direction, `0.0` as **Y** direction and `-1.0` as **Z** direction.

    This defines the direction of the perturbation (the positive Z direction)
54. Enter `20.0` for the **Magnitude**
55. Push the **Add Force** button
56. Push the **Finish** button

Create a Loadcase

57. From the **Analysis** category chooser, select **Loadcases**
58. Push the **New Loadcase** button from the Edit menu toolbar

59. Check the **Static** radio button
60. Push **Next>**
61. From the **SPC** category chooser, select the Grid-Component-Set you have just created
62. Push **Next>**
63. From the **Load Set** category chooser, select the Static Load you have just created
64. Push **Next>**
65. Select **Post** and **All** for the Element Stress options
66. Push the **Finish** button

Export the Input File

67. From the main menu bar select **File** → **Export** → **Input Data...**
68. Enter `SHDSG016`
69. Push the **Save** button

Quit Design Studio

70. From the main menu bar, select **File** → **Quit**
71. Push the **Don't Save** button

Part 2

The purpose of this part is to learn how to create HEXA domains and how to create shape morphing sets.

You can work using your own file you have created in the Part1 `SHDSG016.dat` or use the reference file `SHDSG016_ref1.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG016.dat`

Create Multiple Shape Domains

We are going to create 5 HEXA domains as shown below.

3. Select the **Design** tab
4. From the **Design** category chooser, select **Shape Domains**
5. Push the **New Domain** button from the Edit menu toolbar
6. Enter the Name `Beam`
7. Check the **New Domains Quick Setup** radio button
8. Push **Next**>
9. Select `Create New Domain Group` item
10. Push **Next>**
11. Select the **Iso Right-Front-Top** view icon in the viewport window
12. Check the **Point by Point** radio button
13. Check the **Pick existing grids** radio button
14. Select the **Hexas** icon as **Region Definition Option**
15. Enter `0.0` for **X**, `0.0` for **Y** and `0.0` for **Z**
16. Push the Enter key from your keyboard

    Repeat steps 15 and 16 to define the other corners of the Hexas in order of the following table

Corner	X	Y	Z
2	0.0	0.0	10.0
3	0.0	4.0	10.0

4	0.0	4.0	0.0
5	80.0	0.0	0.0
6	80.0	0.0	10.0
7	80.0	4.0	10.0
8	80.0	4.0	0.0

17. In the list, select the Hexa you just created
18. Push the **Subdivide** button
19. Enter `1` as Elements in 1st Dimension
20. Enter `1` as Elements in 2nd Dimension
21. Enter `5` as Elements in 3rd Dimension
22. Push **Next**>
23. Push the **Finish** button

    Verify that there are 5 domains listed
24. Right click on the Viewport window and select **Clear** → **All**

Create the Perturbations associated to the first set of Domains

We are going to create 10 perturbations on the HEXAS domains.

25. From the **Design** category chooser, select **Shape Morphing Sets**
26. Push the **New Shape Set** button from the Edit menu toolbar
27. Enter the Name `Beam_Shape1`
28. Check the **Domain Morphing Set** radio button
29. Push **Next>**
30. Select the 5 Hexa Domains from the **Select Act Upon** list.
31. Push **Next>**
32. Push the **Select None** button
33. Enter `55` in the **Select by Grid ID** field
34. Push the **Add** button
35. Enter `11` in the **Select by Grid ID** field
36. Push the **Add** button

    Verify that there are 2 grids selected. The perturbation will be applied on these grids**.**

37. Enter `0.0` as **X** direction, `0.0` as **Y** direction and `-1.0` as **Z** direction.

    This defines the direction of the perturbation (the negative Z direction)

38. Enter `4.0` for the **Magnitude**

39. Push the **Add Perturbation** button

    Verify that there is “**2 perturbations on 2 grid”** at the bottom of the window

40. Push the **Select None** button

41. Enter `1` in the **Select by Grid ID** field

42. Push the **Add** button

43. Enter `45` in the **Select by Grid ID** field

44. Push the **Add** button

    Verify that there are 2 grids selected. The perturbation will be applied on these grid**.**

45. Enter `0.0` as **X** direction, `0.0` as **Y** direction and `1.0` as **Z** direction.

    This defines the direction of the perturbation (the positive Z direction)

46. Enter `4.0` for the **Magnitude**

47. Push the **Add Perturbation** button

    Verify that there is “**4 perturbations on 4 grids”** at the bottom of the window

48. Push the **Finish** button

49. Right click on the Viewport window and select **Clear** → **All**

    Repeat steps 26 to 49 to create 4 others Shape Morphing Sets using the following table

Shape Morphing Sets Name	Perturbated grids ID	Direction of the perturbation	Magnitude of the perturbation
Beam_Shape2	935 891	-Z	4.0
	925 881	Z	4.0
Beam_Shape3	1815 1771	-Z	4.0
	1805 1761	Z	4.0
Beam_Shape4	2695 2651	-Z	4.0
	2685 2641	Z	4.0

Beam_Shape5	3575 3531	-Z	4.0
	3565 3521	Z	4.0

Checking Perturbations

It is recommended to check each perturbation vector after creating them because it helps to prevent bad element deformations. At this point if you have any problem you can compare your file with the reference file `SHDSG016_ref2.dat`

50. Select the **Post** tab
51. Push the **Deform Mesh/Color Mesh**
52. In the Deform Mesh window, select **Ramp** (try also **Static** and **Oscillate**).
53. In the **Color Mesh** window, select **Filled Contours**.
54. Select each **Perturbation** from the **Color Mesh** window and study it.
55. Push **Up** when finished

Create the Objective

56. From the **Design** category chooser, select **Objectives**
57. Push the **New Objective** button from the Edit menu toolbar
58. Check the **Mass** radio button
59. Push the **Finish** button

Create the Constraints

You need to create the two constraints, first the stress constraint and then the deflection constraint.

60. From the category chooser, select **Constraints**
61. Push the **New Constraint** button from the Edit menu toolbar
62. Check the **Stress** radio button
63. From the **Stress** category chooser, select **Selected Groups**
64. Enter `330` as Upper Bound
65. Push **Next>**
66. Select PSOLID1 in the Group list

67. Push **Next>**
68. From the **PSOLID Stress** category chooser, select **von Mises**
69. Push **Next>**
70. Select the existing loadcase
71. Push the **Finish** button
72. Push the **New Constraint** button from the Edit menu toolbar
73. Check the **Displacement** radio button
74. Enter `0.2` as **Upper Bound**
75. Push **Next>**
76. Push the **Select None** button
77. Enter `23` in the **Select by Grid ID** field
78. Push the **Add** button

    Repeat the two last steps to select grids 24, 25, 26, 27, 28, 29, 30, 31, 32, 33

79. Check the **Translation Magnitude** radio button
80. Push **Next>**
81. Select the existing loadcase
82. Push the **Finish** button

Setting the number of design Cycle

83. From the main menu bar, select **Genesis** → **Option**
84. Select the **Design Control** tab in the Genesis Options window
85. Enter `20` as **Maximum Design Cycles**
86. Push the **Apply** button

Optimize the structure using Genesis

87. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History**, when done push the **Close** button
    Study the **Genesis Console Output**, when done push the **Close** button

Import the Post Processing Files

88. From the main menu bar select **File** → **Import** → **Punch/Output2 Results...**

89. Select the `SHDSG016_dsg.SHP` from the file browser and push **Open**
90. From the main menu bar select **File** → **Import** → **Punch/Output2 Results...**
91. Select one `SHDSG016_dsg00.pch`, and check the **Import Similar Results for All Design Cycles** check box
92. Push the **Open** button

Postprocessing the Results

93. Select the **Post** tab

    You can view individual frames with **Deform Mesh/Color Mesh** or you can make an animation with **Animation**. You can also animate shape changes and element results (e.g. stresses) simultaneously

94. Push **Deform Mesh/Color Mesh...**
95. In the **Deform Mesh** window, select **Ramp** (try also **Static** and **Oscillate**)
96. In the **Color Mesh** window, select **Filed Contour**
97. Select each proposed **Cycle** to check the Grid Stress
98. Push **Up** when finished]
99. From the **Post** tab, push the **Animation** button
100. Select **Shape Change** from the **Deform Results Type** menu
101. Select **Solid stress/strain** from the **Color Results Type** menu
102. Check the **Filled Contours** radio button
103. Push **Next>**
104. Select all the Cycles for Shape Change from the list
105. Push **Next>**
106. Select all the Cycles for Loadcase 1 Element Stress from the list
107. Push **Next>**
108. Push **Finish**

Quit Design Studio

109. From the main menu bar, select **File** → **Quit**
110. Push the **Don't Save** button

Study the Results

111. Open the output file. Study the values of the objective and complete the following table

	Reference file answers (1)	Your answers (2)
Objective value for the first Design cycle	2.4960E-5	
Objective value for the last Design cycle	1.3376E-5	

(1)Results from the run of `SHDSG016_ref2.dat`

(2)Results from your run

Part 3

The purpose of this part is to learn how to create and to export a file with a CAD geometry that represents the results of a shape optimization. You will create two IGES files, one with a coarse meshing and another with a refined meshing.

When you finish this example, you should have created four files named:
```
SHDSG016_RefinedMesh.igs
SHDSG016_CoarseMesh.igs
SHDSG016_RefinedMesh.dat
SHDSG016_CoarseMesh.dat
```

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG016_ref2.dat`

Optimize the structure using Genesis

3. From the main menu bar, select **Genesis** → **Optimize**

   Study the **Design History**, when done push the **Close** button
   Study the **Genesis Console Output**, when done push the **Close** button

Import the Post Processing Files

4. From the main menu bar select **File** → **Import** → **Punch/Output2 Results...**
5. Select the `SHDSG016_ref2_dsg.SHP` from the file browser and push **Open**
6. Push the **Open** button

Post-Processing the Results

7. Select the **Post** tab
8. Push the **Deform Mesh/Color Mesh** button
9. Select the Shape change of the last design cycle

Export a Coarse Surface Representation of the Shape Optimization Results using IGES Data Format

10. From the main menu bar select **File** → **Export** → **Coarsened Surface...**
11. For **Surface File Format**, select **IGES**

12. Enter `SHDSG016_CoarseMesh` as file name
13. Push the **Save** button

Export a Refined Surface Representation of the Shape Optimization Results using IGES Data Format

14. From the main menu bar select **File** → **Export** → **Coarsened Surface...**
15. For **Surface File Format**, select **IGES**
16. Move right the **Surface Mesh** slide to the **Fine** setting
17. Enter `SHDSG016_RefinedMesh` as file name
18. Push the **Save** button

Export a Coarse Surface Representation of the Shape Optimization Results using Input Data Format

Because Design Studio can not read IGES files, you will create the same mesh as above but using the Genesis input data format.

19. From the main menu bar select **File** → **Export** → **Coarsened Surface...**
20. Enter `SHDSG016_CoarseMesh` as file name
21. Push the **Save** button

Export a Refined Surface Representation of the Shape Optimization Results using Input Data Format

22. From the main menu bar select **File** → **Export** → **Coarsened Surface...**
23. Move right the **Surface Mesh** slide to the **Fine** setting
24. Enter `SHDSG016_RefinedMesh` as file name
25. Push the **Save** button
26. Push the **Up** button

Import the Coarse Surface Representation of the Shape Optimization Results

27. Import the Genesis data file: `SHDSG016_CoarseMesh.dat`
28. Select the **Display** tab
29. Push the **Show/Hide Groups** button

30. Hide PSOLID 1
31. Study the Viewport
32. Using the **Group Display Style** icons, select **Flat Shaded**

Import the Refined Surface Representation of the Shape Optimization Results

33. Import the Genesis data file: `SHDSG016_RefinedMesh.dat`

Compare the Coarse and the Refined Surface Representation of the Shape Optimization Results

34. Hide PSHELL 3

    Compare the two meshes by displaying and hiding PSHELL 3 and PSHELL 4.

35. Which surface is better?

    From the quality of the mesh point of view, the refined is clearly better. However, many CAD programs are slow to read refined meshes and therefore; sometimes, the coarse representation can be better. In this case you studied two levels, Design Studio allows you to select higher or lower coarse levels. You might select other coarse levels to find the one that is accurate enough and simultaneously small enough.

Counting Number of elements in each Mesh

36. Push the **Hide All** button
37. Select PSHELL 3 corresponding to the coarse mesh
38. Select the **Analysis** tab
39. From the category chooser, select **Elements**
40. Push the **Select All** button
41. Read the number of elements printed near the bottom of the Form
42. Push the **Select None** button
43. Repeat steps from 37 to 43 by selecting the PSHELL 4 corresponding to the refined mesh

Quit Design Studio

44. From the main menu bar, select **File** → **Quit**
45. Push the **Don't Save** button

Part 4

The purpose of this part is to demonstrate how to create and export an updated input data file which contains the new grid locations of the optimized structure. You will create a new data file: `SHDSG016Updated.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SHDSG016_ref2.dat`

Request the UPDATE File

The UPDATE file is a Genesis input file that contains the updated data entries.

3. From the main menu bar, select **Genesis** → **Options**
4. Select the **File Control** tab in the Genesis Options window
5. Check **Updated Input File**
6. Select **Last Cycle** from the **Updated Input File** category chooser

   The Last Cycle option will create one updated file corresponding to the last design cycle. The name of the file will be: `SHDSG016_ref2_dsgUPDATExx.dat` (where xx is the last design cycle number)

7. Push the **Apply** button

Optimize the structure using Genesis

8. From the main menu bar, select **Genesis** → **Optimize**

   Study the **Design History**, when done push the **Close** button
   Study the **Genesis Console Output**, when done push the **Close** button

Quit Design Studio

9. From the main menu bar, select **File** → **Quit**
10. Push the **Don't Save** button

Create a Genesis Input File

11. Copy `SHDSG016_ref2_dsgUPDATExx.dat` to `SHDSG016Updated.dat`

    The UPDATE file contains valid bulk data, but non valid executive and solution control data. To create a working file, you need to add valid Executive Control and Solution Control entries. You will do this in the next step.

12. In a text editor, load the two file `SHDSG016_ref2.dat` and `SHDSG016Updated.dat`
13. Replace the Executive Control Commands and the Solution Control Commands from `SHDSG016_ref2.dat` to `SHDSG016Updated.dat`

    The Executive Control Commands and the Solution Control Commands correspond to all data entries above the BEGIN BULK command
    The `SHDSG016Updated.dat` should now be a working file that can be used to run Genesis.
    You can compare your file with the reference file `SHDSG016_ref3.dat`

6.17 Using Multiple Quad Domains

Introduction

The purpose of this exercise is to get familiar with the creation of several shape domain morphing sets and to learn how to post process advanced shape optimization results.

The following optimization problem will be created and solved:

Minimize the mass of the structure

Subject to:

Von Mises Stress < 150MPa

Designable region:

Thickness of material web between pockets
Depth of material on longest edge

Example ID

`SHDSG017`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG017.dat`	`Input data`	Provided: Contains the finite element mesh of the structure
`SHDSG017_ref.dat`	`Input data`	Provided: Contains the finite element mesh of the structure and the set of domain

Start Design Studio

1. Start Design Studio
2. Load the Genesis data file: `SHDSG017.dat`

   The imported model will be shown in Viewport Window area.

Create Multiple Shape Domains

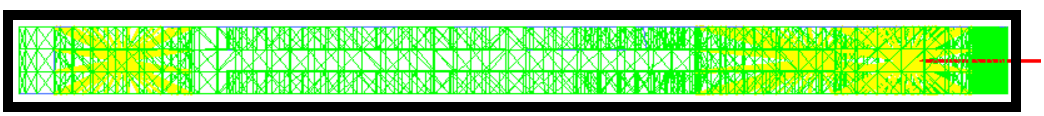
We are going to create 3 QUAD domains as shown below.

3. Select the **Left** view icon in the Viewport window

   Hint: To see the icon names, click on the Viewport window and let the cursor of the mouse few second on each icon

4. Select the **Design** tab
5. From the **Design** category chooser, select **Shape Domains**
6. Push the **New Domain** button from the Edit menu toolbar
7. Enter the Name `Thickness`
8. Check the **New Domains Quick Setup** radio button
9. Push **Next**>
10. Select `Create New Domain Group` item
11. Push **Next>**
12. Check the **Drag out size** radio button
13. Check the **Pick Points on Workplane** radio button
14. Select the **XZ plane of view coordinate system** icon
15. Select the **Quads** icon as **Region Definition Option**
16. Create a Domain as shown below on the picture

    To create the domain, you need to pick the 4 corners of the QUAD domain in the Viewport window by holding the mouse while dragging over the desired region



    Verify that there is 1 region listed

17. Select the Quad region you just created
18. Push the **Subdivide** button
19. Enter `1` as Elements in 1st Dimension
20. Enter `3` as Elements in 2nd Dimension

    The first dimension is displayed by a red line on the shape domain in the Viewport window. The second dimension is displayed by a green line.

21. Push **Next**>
22. Push the **Finish** button
23. Right click on the Viewport window and select **Clear** → **All**

Create the Perturbations associated to the first set of Domains

We are going to create 4 perturbations on the QUAD domains.

24. From the **Design** category chooser, select **Shape Morphing Sets**
25. Push the **New Shape Set** button from the Edit menu toolbar
26. Enter the Name `Thickness`
27. Check the **Domain Morphing Set** radio button
28. Push **Next>**
29. Select `Thickness` from the **Select Act Upon** list.

    There are 3 Quad domains listed. Select all.
30. Push **Next>**
31. Push the **Select None** button
32. Enter `7362` in the **Select by Grid ID** field
33. Push the **Add** button
34. Enter `7363` in the **Select by Grid ID** field
35. Push the **Add** button

    Verify that there are 2 grids selected. The perturbation will be applied on this grids.
36. Enter `0.0` as **X** direction, `0.0` as **Y** direction and `1.0` as **Z** direction.

    This defines the direction of the perturbation (the positive Z direction)
37. Enter `4.0` for the **Magnitude**
38. Push the **Add Perturbation** button

    Verify that there is “**2 perturbations on 2 grid”** at the bottom of the window
39. Push the **Select None** button
40. Enter `7364` in the **Select by Grid ID** field
41. Push the **Add** button
42. Enter `7365` in the **Select by Grid ID** field
43. Push the **Add** button

    Verify that there are 2 grids selected. The perturbation will be applied on this grid.
44. Enter `0.0` as **X** direction, `0.0` as **Y** direction and `-1.0` as **Z** direction.

    This defines the direction of the perturbation (the negative Z direction)
45. Enter `4.0` for the **Magnitude**
46. Push the **Add Perturbation** button

Verify that there is “**4 perturbations on 4 grids”** at the bottom of the window

47. Push the **Finish** button
48. Right click on the Viewport window and select **Clear** → **All**

Create a New View in the View Catalog

49. Select the **Top** view icon in the Viewport window
50. Push the **Rotate about an Axis** icon in the Viewport window
51. From the category chooser in the Rotate window, select **Z**
52. In the Rotate window, move the cursor from 0 to nearly 40

    You see the view of the structure changing in the Viewport window by turning around the Z axis
53. Push the **Close** button in the rotate window
54. Push the **View Catalog** icon in the Viewport window
55. Push the **New** button
56. Enter `Top_View` instead of `View 1`
57. Push the **Close** button

    You have created a view. It is useful when you need to work often but not in the common view plans XY, XZ or YZ

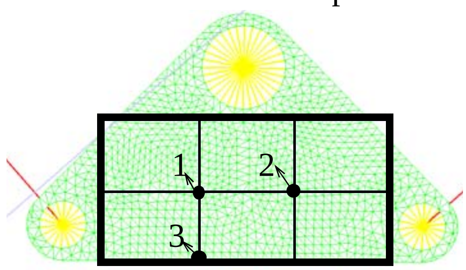
Create Multiple Shape Domains

We are going to create 6 QUAD domains.

58. From the **Design** category chooser, select **Shape Domains**
59. Push the **New Domain** button from the Edit menu toolbar
60. Enter the Name `Face_Movement`
61. Check the **New Domains Quick Setup** radio button
62. Push **Next**>
63. Select `Create New Domain Group` item
64. Push **Next>**
65. Push the **Relocate Origin** icon
66. Select the **Pick existing grids** option
67. In the Viewport window, push the **Front** view icon
68. Select a grid on the top of the structure

    This means all domains will be easily visible rather than hidden in the middle of the part

69. In the Viewport window, push the **View Catalog** icon
70. Select `Top_View` in the View Catalog and push the **Close** button
71. Check the **Drag out size** radio button
72. Check the **Pick Points on Workplane** radio button
73. Select the **SCR** icon (**XY Plan of screen)**
74. Select the **Quads** icon as **Region Definition Option**
75. Create a Domain as shown below on the picture



You just create the exterior rectangle, you will subdivide the rectangle in 6 parts in the followings steps

76. Select the Quad region you just created
77. Push the **Subdivide** button
78. Enter `3` as Elements in 1st Dimension
79. Enter `2` as Elements in 2nd Dimension

The first dimension is displayed by a red line on the shape domain in the Viewport window. The second dimension is displayed by a green line.

80. Push **Next**>
81. Push the **Finish** button
82. Right click on the Viewport window and select **Clear** → **All**

Create the Perturbations associated to the first set of Domains

We are going to create 2 perturbations on the QUAD domains.

83. From the **Design** category chooser, select **Shape Morphing Sets**
84. Push the **New Shape Set** button from the Edit menu toolbar
85. Enter the Name `Face_Movement`
86. Check the **Domain Morphing Set** radio button
87. Push **Next>**
88. Select `Face_Movement` domains from the **Select Act Upon** list.

There are 6 Quad domains listed. Select all.

89. Push **Next>**
90. Push the **Select None** button
91. Pick grids 1 and 2 from the previous picture

    Verify that there are 2 grids selected. The perturbation will be applied on this grids.
92. Push the **By 2 Grids...** button
93. Pick grids 1 and 3 from the previous picture

    This defines the direction of the perturbation from grid 1 to grid 3
94. Push **Next>**
95. Enter `7.0` for the **Magnitude**
96. Push the **Add Perturbation** button

    Verify that there is “**2 perturbations on 2 grid”** at the bottom of the window
97. Push the **Finish** button
98. Right click on the Viewport window and select **Clear** → **All**

Checking Perturbations

It is recommended to check each perturbation vector after creating them because it helps to prevent bad element deformations. At this point if you have any problem you can compare your file with the reference file `SHDSG017_ref.dat`

99. Select the **Post** tab
100. Push the **Deform Mesh/Color Mesh**
101. In the Deform Mesh window, select **Ramp** (try also **Static** and **Oscillate**).
102. In the **Color Mesh** window, select **Filled Contours**.
103. Select each **Perturbation** from the **Color Mesh** window and study it.
104. Push **Up** when finished

Checking the Objective

105. From the **Design** category chooser, select **Objectives**
106. Select the listed objective
107. Push the **Modify Objective** button from the Edit Menu toolbar and study the data
108. Push the **Cancel** button

Checking the Constraints

109. From the category chooser, select **Constraints**
110. Select the listed objective
111. Push the **Modify Constraint** button from the Edit Menu toolbar and study the data
112. Push **Next>** and study the data
113. Push **Next>** and study the data
114. Push **Next>** and study the data
115. Push the **Cancel** button

Optimize the structure using Genesis

116. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History**, when done push the **Close** button
    Study the **Genesis Console Output** window.

Import the Post Processing Files

117. From the **Genesis Console Output** window, select **Import Post...** button
118. Select the `SHDSG017_dsg.SHP` from the file browser
119. Select the files `SHDSG017_dsgxx.pch`, where `xx` is a design cycle number
120. Push the **Import** button
121. From the **Genesis Console Output** window, select the **Close** button

Postprocessing the Results

122. Select the **Post** tab

    You can view individual frames with **Deform Mesh/Color Mesh** or you can make an animation with **Animation**. You can also animate shape changes and element results (e.g. stresses) simultaneously

123. Push **Deform Mesh/Color Mesh...**
124. In the **Deform Mesh** window, select **Ramp** (try also **Static** and **Oscillate**)
125. In the **Color Mesh** window, select **Filled Contours**
126. Select each proposed **Cycle** to check the Grid Stress
127. Push **Up** when finished]
128. From the **Post** tab, push the **Animation** button

129. Select **Shape Change** from the **Deform Results Type** menu
130. Select **Solid stress/strain** from the **Color Results Type** menu
131. Check the **Filled Contours** radio button
132. Push **Next>**
133. Select all the Cycles for Shape Change from the list
134. Push **Next>**
135. Select all the Cycles for Loadcase 1 Element Stress from the list
136. Push **Next>**
137. Push **Finish**

Quit Design Studio

138. From the main menu bar, select **File** → **Quit**
139. Push the **Don't Save** button

Study the Results

140. Open the output file. Study the values of the objective and complete the following table

	Reference file answers (1)	**Your answers (2)**
Objective value for the first Design cycle	1.8784E-3	
Objective value for the last Design cycle	1.4773E-3	
Improvement (%)	21	

(1)Results from the run of `SHDSG017_ref.dat`
(2)Results from your run

6.18 Using Axisymmetric Bar Domains

Introduction

The purpose of this exercise is to get familiar with the creation of shape domain morphing sets and to learn how to post process simple advanced shape optimization results.

The following optimization problem will be created and solved:

Minimize Mass

Subject to:

Von mises Stress $\leq$ 120 MPa

Designable region:

Axisymmetric regions

Example ID

SHDSG018

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SHDSG018.dat	Input data	Provided: Contains the finite element mesh of the structure
SHDSG018_ref.dat	Input data	Provided: Contains the finite element mesh of the structure and the sets of domain

Start Design Studio

1. Start Design Studio
2. Load the Genesis data file: SHDSG018.dat

Create new Grids

We are going to create 3 grids which will help to create the shape domain.

3. Select the **Analysis** tab
4. From the category chooser, select **Grids**
5. Push the **New Grids** button from the Edit menu toolbar
6. Enter `1` as **Number of New Grids**
7. Accept the default option **Along a Line**
8. Push **Next**>
9. Enter `-7.0` as **Base X1**, `0.0` as **Base X2** and `0.0` as **Base X3**
10. Accept the default 1.0 as **Offset Scale Factor**
11. Push the **Finish** button

Repeat steps 5 to 11 to create 2 other grids with the data from the following table:

	Base X1	Base X2	Base X3
Second New Grid	1	0	0
Third New Grid	-7	-10	0

Create Multiple Shape Domains

We are going to create 2 BARX domains as shown below.

12. Select the **Design** tab
13. From the **Design** category chooser, select **Shape Domains**
14. Push the **New Domain** button from the Edit menu toolbar
15. Enter the **Name** `Inner_wall`
16. Check the **Define A New Original Domain** radio button
17. Push **Next**>
18. Select `Create New Domain Group` item
19. Push **Next>**
20. Check the **Axisymmetric Bar** radio button
21. Push **Next>**
22. Enter the 4 grids ID `10687`, `10686`, `10632`, `10449` (in order) in the **Select by Grid ID** field.

    To select a Grid, you need to push the *Enter* key on your keyboard to complete the selection. The last two grids define the symmetrical axis of rotation.
23. Push **Next>**

24. Enter `270` in the **Radius for non-3D Domains** field
25. Push the **Select Interior Grids** button

    Verify that there are 2563 grids selected
26. Push the **Finish** button
27. Right click on the Viewport window and select **Clear** → **All**

Repeat steps 14 to 27 to create the second domain with data from the following table:

Domain Name	Corner Grids ID (in order)	Grid ID for the symmetric axis	Radius	Number of grids selected
`Vessel_floor`	10686 10688	10632 10449	120	1207

Verify that the 2 Shape Domains you just created are listed

Create the Perturbations associated to the set of Domains

We are going to create 2 independent perturbations with the 2 BARX domains.

28. From the **Design** category chooser, select **Shape Morphing Sets**
29. Push the **New Shape Set** button from the Edit menu toolbar
30. Enter the **Name** `InnerWall`
31. Check the **Domain Morphing Set** radio button
32. From the **Design** category chooser, select **Design Variable**s

    We need to create a new **Design Variable** to associate it to the first perturbation vector then we will come back to this **Shape Morphing Sets** option. Do not push **Cancel** at this point.
33. Push the **New Design Variable** button from the Edit menu toolbar
34. Enter the **Name** `InnerWall_DV`
35. Accept the **Independent Design Variable** option.
36. Push **Next>**
37. Enter `0.0` as **Initial Value,** `-2.0` as **Lower bound** and `2.0` as **Upper bound.**
38. Push **Finish**
39. From the **Design** category chooser, select **Shape Morphing Sets**
40. Select `InnerWall_DV` in the Design Variable menu
41. Push **Next>**
42. Select `Inner_wall` from the **Select Act Upon** list.
43. Push **Next>**

44. Push the **Select None** button
45. Enter `10686` in the **Select by Grid ID** field
46. Push the **Add** button

    Verify that there is 1 grid selected. The perturbation will be applied on this grid.
47. Enter `-1.0` as **X** direction, `0.0` as **Y** direction and `0.0` as **Z** direction.

    This defines the direction of the perturbation (the negative X direction)
48. Enter `6.0` for the **Magnitude**
49. Push the **Add Perturbation** button

    Verify that there is “**1 perturbations on 1 grid”** at the bottom of the window
50. Push the **Finish** button
51. Right click on the Viewport window and select **Clear** → **All**

Repeat steps 29 to 51 to create the second perturbation vector with the following table:

Shape Morphing Set Name	Associated Design Variable Name (*)	Domains to Act Upon	Perturbated Grid ID	Direction of the perturbation	Magnitude
`vessel_floor`	`vesselFloor_DV`	`Vessel_floor`	10688	-Y	6.0

*For initial value, lower bound and upper bound, please keep the same values than the InnerWall_DB.

Checking Perturbations

It is recommended to check each perturbation vector after creating them because it helps to prevent bad element deformations. At this point if you have any problem you can compare your file with the reference file `SHDSG018_ref.dat`

52. Select the **Post** tab
53. Push the **Deform Mesh/Color Mesh**
54. In the **Deform Mesh** window, select **Ramp** (try also **Static** and **Oscillate**).
55. In the **Color Mesh** window, select **Filled Contours**.
56. Select each **Perturbation** from the **Color Mesh** window and study it.
57. Push **Up** when finished

Checking the Objective

58. From the **Design** category chooser, select **Objectives**
59. Select the listed objective

60. Push the **Modify Objective** button from the Edit Menu toolbar and study the data
61. Push the **Cancel** button

Checking the Constraints

62. From the category chooser, select **Constraints**
63. Select the listed constraint
64. Push the **Modify Constraint** button from the Edit Menu toolbar and study the data
65. Push **Next>** and study the data
66. Push **Next>** and study the data
67. Push **Next>** and study the data
68. Push the **Cancel** button

Optimize the structure using Genesis

69. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History**, when done push the **Close** button
    Study the **Genesis Console Output**, when done push the **Close** button

Import the Post Processing Files

70. From the main menu bar select **File** → **Import** → **Punch/Output2 Results...**
71. Select the `SHDSG018_dsg.SHP` from the file browser and push **Open**
72. From the main menu bar select **File** → **Import** → **Punch/Output2 Results...**
73. Select one `SHDSG018_dsg00.pch`, and check the **Import Similar Results for All Design Cycles** check box
74. Push the **Open** button

Postprocessing the Results

75. Select the **Post** tab

    You can view individual frames with **Deform Mesh/Color Mesh** or you can make an animation with **Animation**. You can also animate shape changes and element results (e.g. stresses) simultaneously
76. Push **Deform Mesh/Color Mesh...**
77. In the **Deform Mesh** window, select **Ramp** (try also **Static** and **Oscillate**)
78. In the **Color Mesh** window, select **Filled Contours**

79. Select each proposed **Cycle** to check the Grid Stress
80. Push **Up** when finished]
81. From the **Post** tab, push the **Animation** button
82. Select **Shape Change** from the **Deform Results Type** menu
83. Select **Solid stress/strain** from the **Color Results Type** menu
84. Check the **Filled Contours** radio button
85. Push **Next>**
86. Select all the Cycles for Shape Change from the list
87. Push **Next>**
88. Select all the Cycles for Loadcase 1 Grid Stress from the list
89. Push **Next>**
90. Push **Finish**

Quit Design Studio

91. From the main menu bar, select **File** → **Quit**
92. Push the **Don't Save** button

Study the Results

93. Open the output file. Study the values of the objective and complete the following table

	Reference file answers (1)	Your answers (2)
Objective value for the first Design cycle	7.7576.E-4	
Objective value for the last Design cycle	3.5168.E-4	
Improvement (%)	45.3	

(*)Results from the run of `SHDSG018_ref.dat`
(*)Results from your run

6.19 Using Axisymmetric Quad and Axisymmetric Bar Domains

Introduction

The purpose of this exercise is to learn how to create and use axis-symmetric domains. The following optimization problem will be created and solved:

Minimize the stresses of the structure

Subject to:

2 type of constraints (mass and stresses)

Designable region:

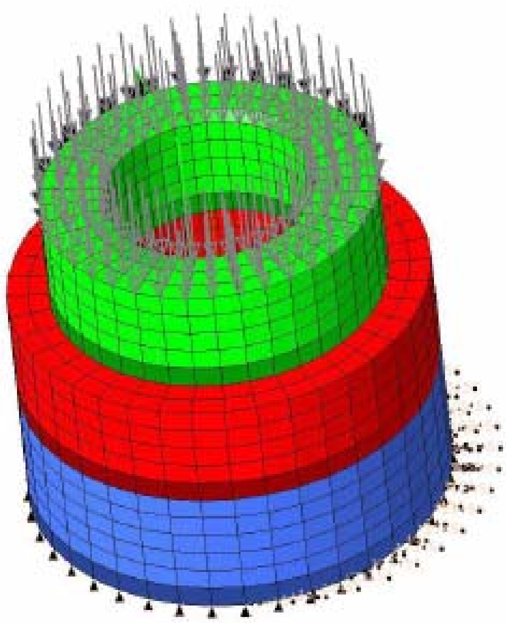
The three existing characteristic diameters

The height of the interface between 3 cylinders

Analysis Consideration:

The structure is subject to compressive loads on the top

The structure is fixed on the bottom



Note: Beta method

The objective function needs to be a single value response. To overcome this difficulty, we use the beta method: we introduce an artificial design variable `beta` and an additional constraint equation using the `beta` value. Genesis cannot directly minimize multiple stresses. The objective function is set to minimize a linear combination of the stresses value.

Here is how we create the appropriate equation:

If we want the stress under 250.

$$\sigma \le 250 \Leftrightarrow \frac{\sigma}{250} \le 1.0$$

Now, if we replace 1.0 with `beta`, we get

$$\frac{\sigma}{250} \le beta \Leftrightarrow 0 \le beta - \frac{\sigma}{250}$$

We minimize beta (with beta<=1.0) then, we minimize the stresses.

Note: Shape Optimization Data

In this example, you will be guided to:

Create 5 shape Domains

Name	Genesis Type	Description	Purpose
`RedCylinder`	QUADX	Axisymmetric Quad	Design vertical dimensions
`GreenCylinder`	QUADX	Axisymmetric Quad	Design vertical dimensions
`BlueCylinder`	QUADX	Axisymmetric Quad	Design vertical dimensions
BAR_1	BARX	Axisymmetric Bar	Design diameters
BAR_2	BARX	Axisymmetric Bar	Design diameters

Create 5 shape morphing sets:

Name	Magnitude	Purpose
`RedCylHeight_1`	0.5	Design height of interface between lower and intermediate cylinder
`RedCylHeight_2`	0.5	Design height of interface between upper and intermediate cylinder
`InnerDiam_1`	0.5	Design the inner diameter
`InnerDiam_2`	0.5	Design the intermediate diameter
`OuterDiam`	0.5	Design the outer diameter

Create 5 design variables

Name	Initial Value	Lower Bound	Upper Bound	Purpose: Scale factor for shape morphing set:
`RedCylHeight_1_DV`	0.0	-5.0	5.0	RedCylHeight_1
`RedCylHeight_2_DV`	0.0	-5.0	5.0	RedCylHeight_2
`InnerDiam_1_DV`	0.0	-2.0	2.0	InnerDiam_1
`InnerDiam_2_DV`	0.0	-2.0	2.0	InnerDiam_2
`OuterDiam_DV`	0.0	-2.0	2.0	OuterDiam

Example ID

`SHDSG019`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG019.dat`	`Input data`	Provided: Contains the finite element mesh of the structure
`SHDSG019_ref1.dat`	`Input data`	Provided: Contains the finite element mesh of the structure and the first sets of domain
`SHDSG019_ref2.dat`	`Input data`	Provided: Contains the finite element mesh of the structure and both first and second sets of domain

Start Design Studio and Load an Existing Genesis Input Data

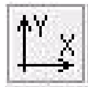
1. Start Design Studio
2. Load the Genesis data file: `SHDSG019.dat`

Change the view

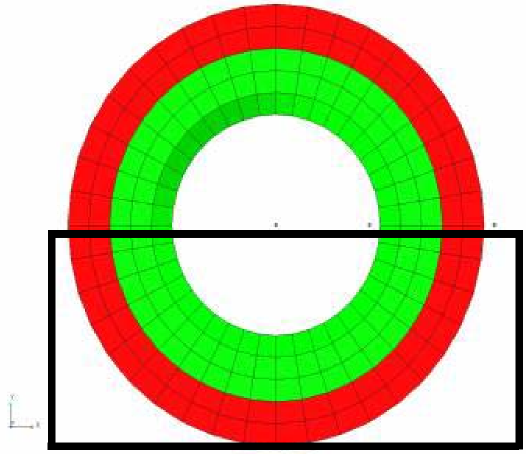
Here we will change the view to select domains easily:

3. Select the **Display** tab
4. From the **Group Display Style** category chooser, select the **Flat Shaded** icon
5. From the **Display** tab, push the **Show/Hide Elements** button

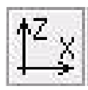
6. In the Viewport window, select the Top view icon (Y arrow up and X arrow right)



7. Select the bottom rows of elements as shown on the following picture.



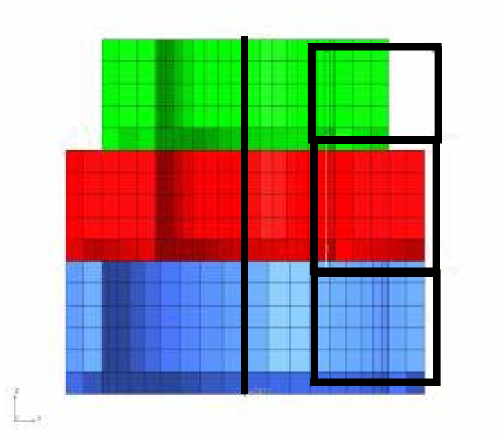
8. Push the **Hide Selected** button
9. In the Viewport window, select the Left view icon (Z arrow up and X arrow right)



10. Push the **Up** button

Create the first set of designable regions

We are going to create 3 QUADX domains as shown below.

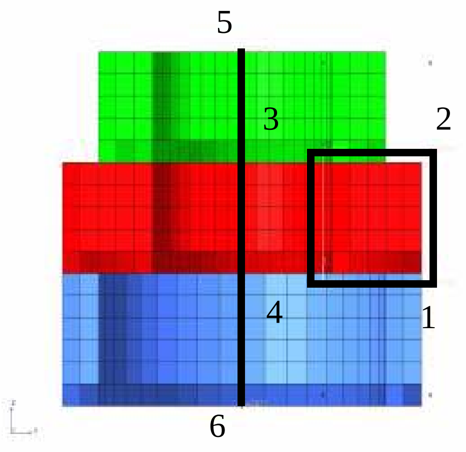


First Domain:

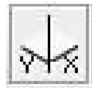
11. Select the **Design** tab
12. From the **Design** category chooser, select **Shape Domains**
13. Push the **New Domain** button from the Edit menu toolbar
14. Enter the **Name** `RedCylinder`
15. Check the **Define A New Original Domain** radio button
16. Push **Next**>
17. Select `Create New Domain Group` item
18. Push **Next>**
19. Check the **Axisymmetric Quad** radio button
20. Push **Next>**
21. Pick the 6 grids 1, 2, 3, 4, 5, 6 in order from the following picture.

    As a convention, the order for the first four grid is using the right hand rule. The last two grids are used to define the axis of symmetry. The free grids are shown by stars in the Viewport window.

    The order for choosing the grids in the axis is not important. You can also enter the number of each grid in the **Select By Grid ID.** Actual IDs are respectively 5005, 6720, 5260, 5263, 3978, 3977. Verify that there is **"6 grids chosen"** at the bottom of the tab.



22. Push **Next>**
23. In the Viewport window, select the Iso Front-Left-Top view icon



24. Push the **Select Interior Grids** button

    As only half of the model is being displayed, only half of the expected grids can now be selected. You need to select all the grids. Do not push **Finish** or **Cancel** at this point
25. Select the **Display** tab

26. Push the **Show/Hide Elements** button
27. Push the **All On** button

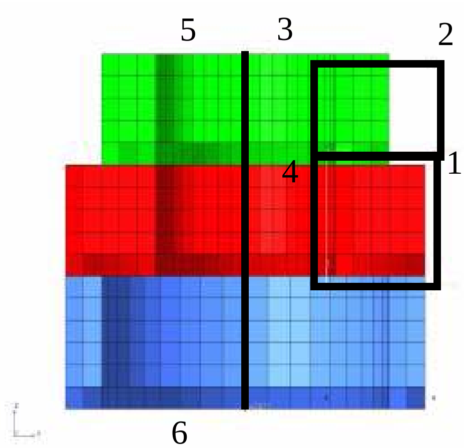
    Now the whole structure is displayed
28. Select the **Design** tab
29. Push the **Select Interior Grids** button

    Verify that there are 1444 grids selected
30. Push the **Finish** button
31. Right click on the Viewport window and select **Clear** → **All**

<u>Second Domain:</u>

32. From the **Design** category chooser, select **Shape Domains**
33. Push the **New Domain** button from the Edit menu toolbar
34. Enter the **Name** `GreenCylinder`
35. Check the **Define A New Original Domain** radio button
36. Push **Next**>
37. Select `Create New Domain Group` item
38. Push **Next>**
39. Check the **Axisymmetric Quad** radio button
40. Push **Next>**
41. Repeat steps from 5 to 11 to change the view
42. Pick the 6 grids 1, 2, 3, 4, 5, 6 in order from the following picture.

    You can also enter the number of each grid in the **Select By Grid ID.** Actual IDs are respectively 6720, 5267, 5261, 5260, 3978, 3977. Verify that there is **"6 grids chosen"** at the bottom of the tab



43. Push **Next>**
44. Repeat steps from 23 to 28 to see the whole structure
45. Push the **Select Interior Grids** button

    Verify that there are 644 grids selected
46. Push the **Finish** button
47. Right click on the Viewport window and select **Clear** → **All**

<u>Third Domain:</u>

Repeat steps 33 to 47 to create the last QUADX domain with the following table:

Domain Name	Corner Grids ID (in order)	Grid ID for the symmetric axis	Number of grids selected
`BlueCylinder`	6719 5005 5263 5266	3978 3977	604

Verify that the 3 Shape Domain you just created are listed

Create the perturbations associated to the first set of domains

<u>First set of perturbation vectors</u>

48. From the **Design** category chooser, select **Shape Morphing Sets**
49. Push the **New Shape Set** button from the Edit menu toolbar
50. Enter the **Name** `RedCylHeight_1`
51. Check the **Domain Morphing Set** radio button
52. From the **Design** category chooser, select **Design Variable**s

    We need to create a new **Design Variable** to associate it to the first perturbation vector then we will come back to this **Shape Morphing Sets** option. Do not push **Cancel** at this point.
53. Push the **New Design Variable** button from the Edit menu toolbar
54. Enter the **Name** `RedCylHeight_1_DV`
55. Accept the **Independent Design Variable** option.
56. Push **Next>**
57. Enter `0.0` as **Initial Value,** `-5.0` as **Lower bound** and `5.0` as **Upper bound.**
58. Push **Finish**

    The “*” means that this design variable is not used yet

59. From the **Design** category chooser, select **Shape Morphing Sets**
60. Select `RedCylHeight_1_DV` in the Design Variable menu
61. Push **Next>**
62. Select both `RedCylinder` and `BlueCylinder` from the **Select Act Upon** list.

    In this case, you could select all domains, but if you do so, Genesis will only use the RedCylind and the BlueCylinder domains, as the perturbations we will create will only share grids with these two.
63. Push **Next>**
64. Repeat steps from 5 to 11 to change the view
65. Push the **Select None** button
66. Enter `5263` in the **Select by Grid ID** field
67. Push the **Add** button
68. Enter `5005` in the **Select by Grid ID** field
69. Push the **Add** button

    Verify that there are 2 grids selected. The perturbation will be applied on those 2 grids.
70. Push the **By 2 Grids...** button
71. Pick grids `3978` and `3977`

    This defines the direction of the perturbation from grid 3978 to grid 3977
72. Push **Next>**
73. Enter `0.5` for the **Magnitude**
74. Push the **Add Perturbation** button

    Verify that there is “**2 perturbations on 2 grids”** at the bottom of the window
75. Push the **Finish** button
76. Right click on the Viewport window and select **Clear** → **All**

<u>Second perturbation vector</u>

Repeat steps 49 to 76 to create the second perturbation vector with the following table:

Shape Morphing Set Name	Associated Design Variable Name (*)	Domains to Act Upon	Perturbated Grid ID	Direction of the perturbation
`RedCylHeight_2`	`RedCylHeight_2 DV`	`RedCylinder GreenCylinder`	5260 6720	-Z

*For initial value, upper-bound et upper bound, please refer to the reference table at the beginning of the exercise

The reason we independently built those two sets of perturbation vectors is to allow for more freedom in the changing shape structure.

77. Repeat steps from 23 to 28 to see the whole structure

Objective and Constraints

The objective function (Minimize Beta) and the constraints are already provided in this example, you can check them by selecting **Objectives** and **Constraints** in the **Design** category chooser**.**

Check Perturbation

It is recommended to check each perturbation vector after creating them because it helps to prevent bad element deformations. At this point if you have any problem you can compare your file with the reference file `SHDSG019_ref1.dat`

78. Select the **Post** tab
79. Push the **Deform Mesh/Color Mesh**
80. In the Deform Mesh window, select **Ramp** (try also **Static** and **Oscillate**).
81. In the **Color Mesh** window, select **Filled Contours**
82. Select each Shape Morphing Set Preview from the **Color Mesh** window and study it.
83. Push **Up** when finished

Create a second set of designable regions

We are going to create 2 BARX domains. They will control 3 different diameters; 2 inner diameters and 2 outers ones.

84. From the **Design** category chooser, select **Shape Domains**
85. Push the **New Domain** button from the Edit menu toolbar
86. Enter the **Name** `BAR_1`
87. Check the **Define A New Original Domain** radio button
88. Push **Next**>
89. Select `Create New Domain Group` item
90. Push **Next>**
91. Check the **Axisymmetric Bar** radio button
92. Push **Next>**
93. Repeat steps from 5 to 11 to change the view

94. Enter the 4 grids ID `6722`, `6721`, `3978`, `3977` (in order) in the **Select by Grid ID** field.

    To select a Grid, you need to push the *Enter* key on your keyboard to complete the selection.
    The last two grids define the symmetrical axis.
    Verify that there is **“4 grids chosen”** at the bottom of the tab

95. Push **Next>**

96. Enter `22` in the **Radius for non-3D Domains** field

97. Repeat steps from 23 to 28 to see the whole structure

98. Push the **Select Interior Grids** button

    Verify that there are 1321 grids selected

99. Push the **Finish** button

100. Right click on the Viewport window and select **Clear** → **All**

Repeat steps 85 to 100 to create the last BARX domain with data from the following table:

Domain Name	Corner Grids ID (in order)	Grid ID for the symmetric axis	Radius	Number of grids selected
BAR_2	6723 6722	3978 3977	34	1646

Verify that the 2 Shape Domains you just created are listed

Create the Perturbations associated to the second set of Domains

We are going to create 3 independent perturbations with the 2 BARX domains.

First set of perturbation vectors

101. From the **Design** category chooser, select **Shape Morphing Sets**

102. Push the **New Shape Set** button from the Edit menu toolbar

103. Enter the **Name** `InnerDiam_1`

104. Check the **Domain Morphing Set** radio button

105. From the **Design** category chooser, select **Design Variable**s

     We need to create a new **Design Variable** to associate it to the first perturbation vector then we will come back to this **Shape Morphing Sets** option. Do not push **Cancel** at this point.

106. Push the **New Design Variable** button from the Edit menu toolbar

107. Enter the **Name** `InnerDiam_1_DV`

108. Accept the **Independent Design Variable** option.

109. Push **Next>**
110. Enter `0.0` as **Initial Value,** `-2.0` as **Lower bound** and `2.0` as **Upper bound.**
111. Push **Finish**
112. From the **Design** category chooser, select **Shape Morphing Sets**
113. Select `InnerDiam_1_DV` in the Design Variable menu
114. Push **Next>**
115. Select `BAR_1` from the **Select Act Upon** list.
116. Push **Next>**
117. Push the **Select None** button
118. Enter `6721` in the **Select by Grid ID** field
119. Push the **Add** button

     Verify that there is 1grid selected. The perturbation will be applied on this grid.
120. Push the **By 2 Grids...** button
121. Pick grids `6721` and `3978`

     This defines the direction of the perturbation from grid 6721 to grid 3978 (the positive X direction)
122. Push **Next>**
123. Enter `0.5` for the **Magnitude**
124. Push the **Add Perturbation** button

     Verify that there is “**1 perturbations on 1 grid”** at the bottom of the window
125. Push the **Finish** button
126. Right click on the Viewport window and select **Clear** → **All**

<u>Second set perturbation vector</u>

Repeat steps 102 to 126 to create the second perturbation vector with the following table:

Shape Morphing Set Name	Associated Design Variable Name (*)	Domains to Act Upon	Perturbated Grid ID	Direction of the perturbation
`InnerDiam_2`	`InnerDiam_2_DV`	`BAR_2`	6722	+X
`OuterDiam`	`OuterDiam_DV`	`BAR_2`	6723	+X

*For initial value, upper-bound et upper bound, please refer to the reference table at the beginning of the exercise

127. Repeat steps from 23 to 28 to see the whole structure

Checking Perturbations

It is recommended to check each perturbation vector after creating them because it helps to prevent bad element deformations. At this point if you have any problem you can compare your file with the reference file `SHDSG019_ref2.dat`

128. Repeat steps 78 to 83 to study the last 3 perturbations you just created.

Optimize the structure using Genesis

129. From the main menu bar, select **Genesis** → **Optimize**

   Study the **Design History**, when done push the **Close** button
   Study the **Genesis Console Output**, when done push the **Close** button

Import the Post Processing Files

130. From the main menu bar select **File** → **Import** → **Punch/Output2 Results...**
131. Select the `SHDSG019_dsg.SHP` from the file browser and push **Open**
132. From the main menu bar select **File** → **Import** → **Punch/Output2 Results...**
133. Select one `SHDSG019_dsg00.PCH`, and check the **Import Similar Results for All Design Cycles** check box
134. Push the **Open** button

Postprocessing the Results

135. Select the **Post** tab

   You can view individual frames with **Deform Mesh/Color Mesh** or you can make an animation with **Animation**. You can also animate shape changes and element results (e.g. stresses) simultaneously

136. Push **Deform Mesh/Color Mesh...**
137. In the **Deform Mesh** window, select **Ramp** (try also **Static** and **Oscillate**)
138. In the **Color Mesh** window, select **Filled Contour**
139. Select each proposed **Cycle** to check the Grid Stress
140. Push **Up** when finished]
141. From the **Post** tab, push the **Animation** button
142. Select **Shape Change** from the **Deform Results Type** menu
143. Select **Grid stress/strain** from the **Color Results Type** menu

144. Check the **Filled Contours** radio button
145. Push **Next>**
146. Select all the Cycles for Shape Change from the list
147. Push **Next>**
148. Select all the Cycles for Loadcase 1 Grid Stress from the list
149. Push **Next>**
150. Push the **Finish** button

Quit Design Studio

151. From the main menu bar, select **File** → **Quit**
152. Push the **Don't Save** button

6.20 Combining Shape and Sizing Optimization I

Introduction

The purpose of this exercise is to learn how to solve a simple shape optimization problem along with sizing optimization.The objectives, constraints and the sizing optimization data is already defined in the provided input file.

The following optimization problem will be created and solved:

Minimize Mass

Subject to:

Von mises Stress $\leq$ 300 MPa

Designable region:

The width of the back part of the structures using one independent design variable

The thickness of the three regions of the structure using two independent design variables

Example ID

SHDSG020

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

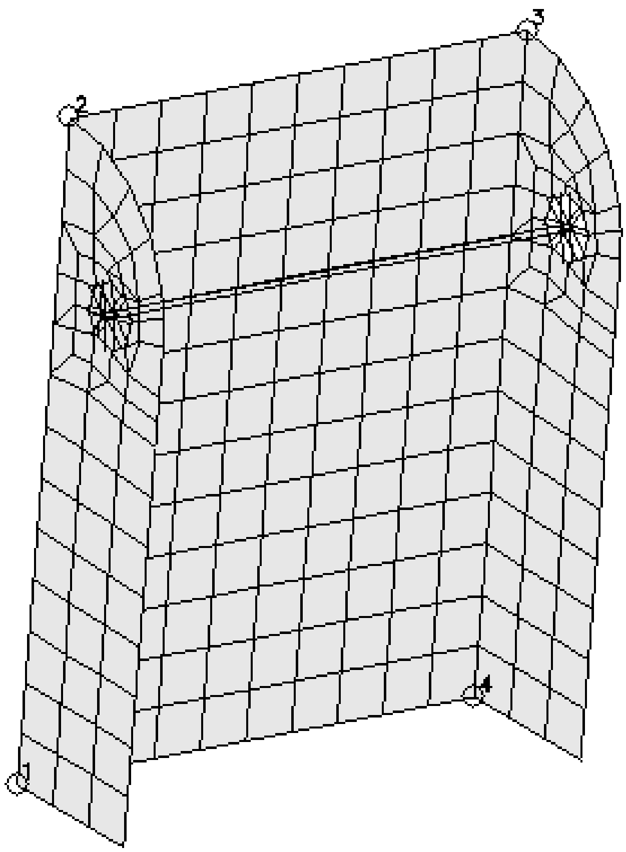
File Name	Type	Description
SHDSG020.dat	Input data	Provided: Contains the finite element mesh of the structure and some design data
SHDSG020_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in SHDSG020.dat.
SHDSG020_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
SHDSG020_dsg.SHP	Shape Change data	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization
SHDSG020_ref.dat	Input data	Provided. Reference result input file. Should be nearly identical to SHDSG020_dsg.dat. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Open the Design Studio data file: `SHDSG020.dat`

Creating a Shape Domain

3. Select **Design** tab.
4. From the **Design** category chooser, select **Shape Domains**
5. Push the **New Domain** button from the Edit menu toolbar to create a new Shape Domain.
6. Enter Name `width` and select the **Define A New Original Domain** radio button
7. Push **Next>**
8. Select `Create New Domain Group` item



9. Push **Next>**

10. Select **Quad** as Type
11. Push **Next>**
12. Pick the four corner grids 1, 2, 3, 4 in order from the graphics area.

    Note: actual Grid ids are: 167, 193, 14, 1
13. Push **Next**>
14. Push the **Select Interior Grids** button.
15. Push **Finish**

Creating Perturbations with Shape Morphing Sets

16. From the **Design** category chooser, select **Shape Morphing Sets**
17. Push the **New Shape Set** button from the Edit menu toolbar to create a new shape morphing set.
18. Enter Name `width`
19. Select **Domain Morphing Set** as Type:
20. Select `New Variable(-1.0, 1.0)`in the **Design Variable** menu.
21. Push **Next**>
22. Select `width` from the Shape Domain list.
23. Push **Next**>

    If any of the grids are preselected, deselect all the grids first by pushing the **Select None** button.
24. Pick grids 1 and 2 as shown in the picture above (this identifies where the perturbation will be applied).

    Verify that “2 grids selected” is shown under the grid selection panel. Otherwise, deselect all grids and repeat selection.
25. Push **By 2 Grids...** button
26. Pick grid 2 and then grid 3 as shown in the picture above

    This defines the direction of the perturbation -- from grid 2 to grid 3
27. Push **Next**>
28. Enter: `75.0` as the Magnitude of the perturbation
29. Push **Add Perturbation**

    Verify that “2 perturbations on 2 grids” appears at the bottom of the panel.
30. Push **Finish**

Checking Perturbations

It is always a good idea to check if the combination of shape domain and perturbations will change the structural shape as you intended before you start the optimization.

31. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
32. Select the **Oscillate** radio button
33. Under the **Deform Mesh**, select a **Shape Morphing Set Preview**

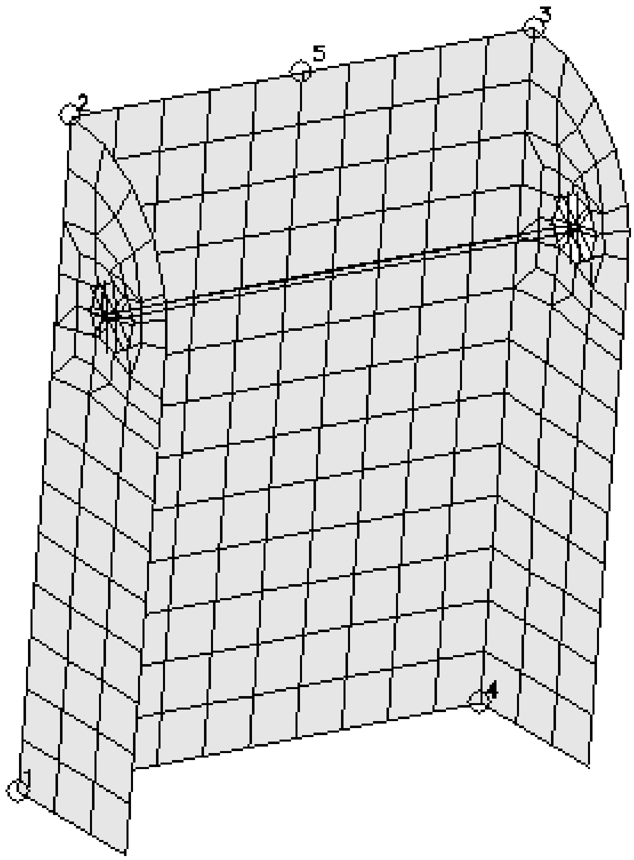
    Push **Static**, **Oscillate**, and **Ramp** buttons to test your preference. After the candidate shape change is confirmed as you intended, you will optimize the structure.
34. Push the **Up** button

Creating a Another Shape Domain

35. Select the **Design** tab.
36. From the **Design** category chooser, select **Shape Domains**
37. Push the **New Domain** button from the Edit menu toolbar
38. Enter Name `gouge` and check the **Define A New Original Domain** radio button
39. Push **Next>**
40. Select `Create New Domain Group` item
41. Push **Next>**
42. Select **Quad** as Type
43. Push **Next>**

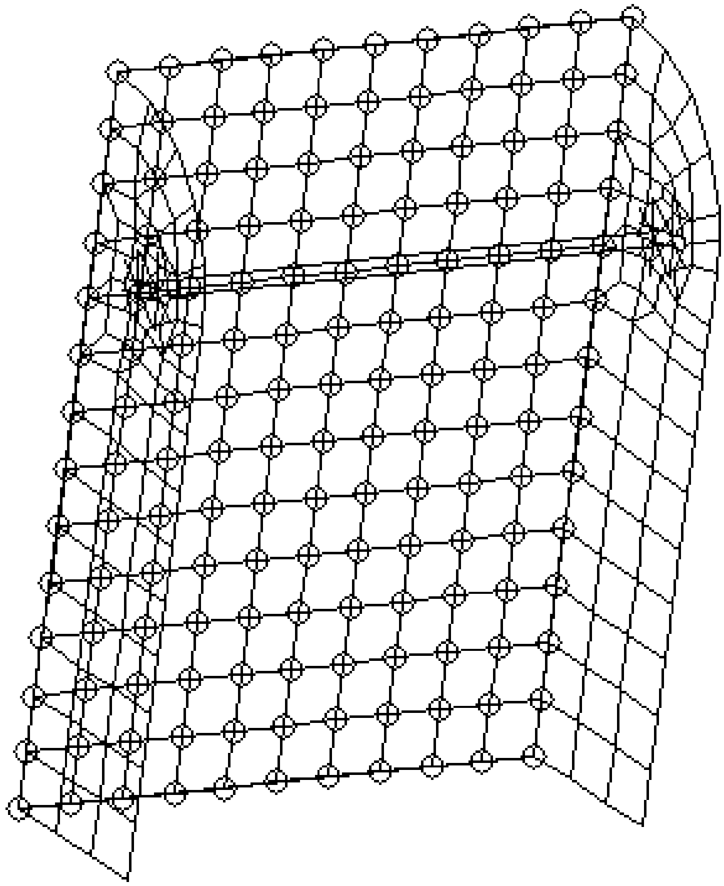
44. Pick nodes 1, 2, 3, 4 in order.

    (Node numbers are from the figure below.)



45. Push **Next>**
46. Enter `1.0` for the **Radius for non-3D Domains**

47. Push the **Select Interior Grids** button.



In this case, we only wanted grids in the plane of the domain to be selected. By setting the radius to a small number, the Select Interior Grids function does just that.

48. Push **Finish**

Creating a Midside Perturbation with Shape Morphing Sets

49. From the **Design** category chooser, select **Shape Morphing Sets**
50. Push the **New Shape Set** button from the Edit menu toolbar
51. Enter Name `gouge`
52. Select **Domain Morphing Set** as Type.
53. Select `New Variable(-1.0, 1.0)` in the **Design Variable** menu.
54. Push **Next**>
55. Select `gouge` from the Shape Domain list.
56. Push **Next**>

57. Push the **Select None** button
58. Pick grid 5 from the picture above (this identifies where the perturbation will be applied).

    Verify that "1 grid selected" is shown under the grid selection panel. Otherwise, deselect all grids and repeat selection.
59. Push **By 2 Grids...** button
60. Pick grid 2 and then grid 1 as defined in the picture above

    This defines the direction of the perturbation -- from grid 2 to grid 1
61. Push **Next**>
62. Enter `50.0` as the Magnitude of the perturbation
63. Push **Add Perturbation** button.

    Verify that "1 perturbations on 1 grids" appears at the bottom of the panel.
64. Push **Finish**

Optimizing and Viewing Results

65. From the main menu bar, select **Genesis** → **Optimize**
66. When the optimization is successfully done, from the **Genesis Console Output** window select the **Import Post...** button
67. Select `SHDSG020_dsg.SHP` from the **New Post-Processing Files** window

    You can view individual frames with **Deform Mesh/Color Mesh** or you can make an animation with **Animation**. You can also animate shape changes and element results (e.g. stresses) simultaneously.
68. Select `SHDSG020_dsgxx.pch` punch result files

    Make sure to all the punch files are selected
69. Push the **Import** button
70. Push **Animation** in the **Post** tab.
71. Select **Shape Change** from the **Deform Results Type** menu.
72. Select **Shell stress/strain** from the **Color Results Type** menu.
73. Select **Filled Contours**
74. Push **Next>**
75. Select all the Cycles for Shape Change from the list.
76. Push **Next>**

77. Select all the Cycles for Loadcase 1 Shell Stress from the list.

    Also test with different loadcases later.

78. Push **Next>**

79. Push **Finish**

Quit Design Studio

80. From the main menu bar, select **File** → **Quit**

81. Push the **Don't Save** button

Study the Results

82. Open the output file. Study the values of the objective and the constraint and complete the following table

Type	Initial Value	Final Value	change%
Objective			
Maximum Constraint Violations			

6.21 Combining Shape and Sizing Optimization II - Use of Domains and Grid Perturbations

Introduction

The purpose of this exercise is to learn how to solve a simple shape optimization problem along with sizing optimization. This problem also goes through the process of creating grid perturbations in Design Studio. The provided input file contains the finite element mesh along with the loading and boundary conditions of a truss tower modeled with 39 ROD elements divided into 5 PROD groups as shown in the figure below.

The following optimization problem will be created and solved:

Minimize Mass

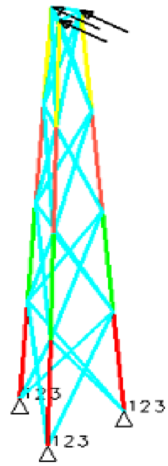
Subject to:

Sum of Displacements of the three grids at the tip of the tower $\leq$ 80.0

Designable region:

Cross-sectional areas of ROD elements using five independent design variables

Shape of the truss tower



Example ID

SHDSG021

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SHDSG021.dat`	`Input data`	Provided: Contains the finite element mesh of the truss tower along with loading and boundary conditions
`SHDSG021_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `SHDSG021.dat`
`SHDSG021_dsg.out`	`Output data`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`SHDSG021_dsg.SHP`	`Shape Change data`	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization
`SHDSG021_dsgOPOSTxx.pch`	`Punch File`	Generated using Genesis within Design Studio. This file contains the sizing optimization data for `xx` design cycle
`SHDSG021_ref.dat`	`Input data`	Provided. Reference result input file. Should be nearly identical to `SHDSG021_dsg.dat`. This file is provided to check your example.

Start Design Studio

1. Start Design Studio
2. Open the genesis input data file: `SHDSG021.dat`

Create the Sizing Optimization data

3. Select the **Design** tab
4. From the **Design** category chooser, select **Quick Setup Trails**
5. Push the **Quick Sizing Setup** button
6. Push the **Select** button to select all the existing groups
7. Push **Next>**
8. Enter `0.1` for **C2** and `0.0` for **M2** to define the lower bound of the design variables
9. Enter `10.0` for **C3** and `0.0` for **M3** to define the upper bound of the design variables
10. Push the **Finish** button

    Notice that in the summary, "**5 design variables defined**" and "**5 Sizing regions defined**" exist. During **Quick Sizing Setup**, design studio creates the 5 design variables and associates them to the areas of each of the 5 PROD groups selected.

Creating a Shape Domain

11. Select **Design** tab.
12. From the **Design** category chooser, select **Shape Domains**
13. Push the **New Domain** button from the Edit menu toolbar to create a new Shape Domain.
14. Enter `Vertical` for the **Name**
15. Make sure the **New Domains Quick Setup** radio button is selected
16. Push **Next>**
17. Select `Create New Domain Group` as Domain group.
18. Push **Next>**
19. Select the **Point by Point** radio button
20. Select the **Lines** icon
21. Enter `50.0`, `0.0`, `0.0` for **X**, **Y**, **Z** respectively
22. Push the `Enter` button on the keyboard

    Notice that the point (50.0, 0.0, 0.0) is picked on the Viewport with a small square around it
23. Again enter `50.0`, `0.0`, `400.0` for **X**, **Y**, **Z** respectively
24. Push the `Enter` button on the keyboard

    Notice that a line domain is created between the two points entered
25. Select the Line domain from the list
26. Push the **Subdivide** button
27. Enter `4` for the **Elements in the 1st dimension**
28. Push **Next>**
29. Push **Finish**

    Notice that there a four BAR domains created along the height of the tower

Review the Shape Domains

30. From the **Design** category chooser, select **Shape Domains**
31. Select one of the existing BAR domains
32. Push the **Modify Domain** button from the Edit Menu toolbar

    Notice that the grids controlled by the BAR domain are selected.
33. Push the **Cancel** button

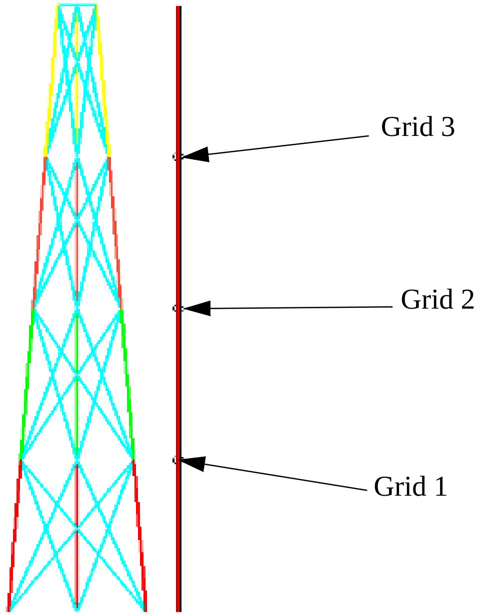
34. Right click on the Viewport window and select **Clear** → **All** to clear the selection

Creating Perturbations on Domains

35. From the **Design** category chooser, select **Shape Morphing Sets**
36. Push the **New Shape Set** button from the Edit menu toolbar to create a new shape morphing set.
37. Enter `Vertical1` for the **Name**
38. Make sure the **Domain Morphing Set** radio button is selected
39. Make sure the option `New Variable(-1.0, 1.0)`is selected for the **Design Variable**
40. Push **Next**>
41. Select all the domains from the Shape Domain list.
42. Push **Next**>

    If any of the grids are preselected, deselect all the grids first by pushing the **Select None** button.
43. Pick Grid 1 as shown in the picture below (this identifies where the perturbation will be applied).

    Verify that "1 grids selected" is shown under the grid selection panel. Otherwise, deselect all grids and repeat selection.



44. Enter `0.0`, `0.0`, `1.0` for **X**, **Y**, **Z** respectively

45. Enter `50.0` as the **Magnitude** of the perturbation
46. Push **Add Perturbation**

    Verify that “1 perturbations on 1 grids” appears at the bottom of the panel.

47. Push **Finish**
48. Push the **New Shape Set** button from the Edit menu toolbar to create a new shape morphing set.
49. Enter `Vertical2` for the **Name**
50. Make sure the **Domain Morphing Set** radio button is selected
51. Make sure the option `New Variable(-1.0, 1.0)`is selected for the **Design Variable**
52. Push **Next**>
53. Select all the domains from the Shape Domain list.
54. Push **Next**>

    If any of the grids are preselected, deselect all the grids first by pushing the **Select None** button.

55. Pick Grid 2 as shown in the picture above

    Verify that “1 grids selected” is shown under the grid selection panel. Otherwise, deselect all grids and repeat selection

56. Enter `0.0`, `0.0`, `1.0` for **X**, **Y**, **Z** respectively
57. Enter `50.0` as the **Magnitude** of the perturbation
58. Push **Add Perturbation**

    Verify that “1 perturbations on 1 grids” appears at the bottom of the panel.

59. Push **Finish**
60. Push the **New Shape Set** button from the Edit menu toolbar to create a new shape morphing set.
61. Enter `Vertical3` for the **Name**
62. Make sure the **Domain Morphing Set** radio button is selected
63. Make sure the option `New Variable(-1.0, 1.0)`is selected for the **Design Variable**
64. Push **Next**>
65. Select all the domains from the Shape Domain list.
66. Push **Next**>

    If any of the grids are preselected, deselect all the grids first by pushing the **Select None** button.

67. Pick Grid 3 as shown in the picture above

    Verify that "1 grids selected" is shown under the grid selection panel. Otherwise, deselect all grids and repeat selection

68. Enter `0.0`, `0.0`, `1.0` for **X**, **Y**, **Z** respectively

69. Enter `50.0` as the **Magnitude** of the perturbation

70. Push **Add Perturbation**

    Verify that "1 perturbations on 1 grids" appears at the bottom of the panel.

71. Push **Finish**

Checking Domain Morphing Perturbations

It is always a good idea to check if the combination of shape domain and perturbations will change the structural shape as you intended before you start the optimization.

72. From the **Post** tab, select the **Deform Mesh/Color Mesh** button

73. Select the **Oscillate** radio button

74. Under the **Deform Mesh**, select a **Shape Morphing Set Preview**

    Push **Static**, **Oscillate**, and **Ramp** buttons to test your preference. After the candidate shape change is confirmed as you intended, you will optimize the structure.

75. Push the **Up** button

Creating Perturbations on Grids

76. From the Viewport window, select the **Top** (X-Y plane) icon for the top view of the model

77. From the **Design** category chooser, select **Shape Morphing Sets**

78. Push the **New Shape Set** button from the Edit menu toolbar to create a new shape morphing set.

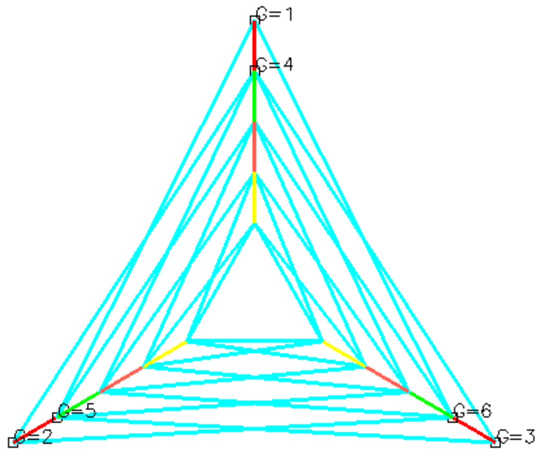
79. Enter `Horizontal1` for the **Name**

80. Select the **Raw Morphing Set** radio button

81. Make sure the option `New Variable(-1.0, 1.0)`is selected for the **Design Variable**

82. Push **Next**>

    If any of the grids are preselected, deselect all the grids first by pushing the **Select None** button.



83. Enter `4` in the **Select by Grid ID** field and push the **Add** button

    Verify that “1 grids selected” is shown under the grid selection panel. Otherwise, deselect all grids and repeat selection.

84. Push the **By 2 Grids...** button

85. Enter `4, 1` in the **Select by Grid ID** field and press the `Enter` button on the keyboard

86. Push **Next>**

87. Delete the existing value for **Z**

    The perturbation that needs to be applied should be in the X-Y plane along grid 4. As the grids 4 and 1 are not in the same X-Y plane, deleting the Z would make the perturbation to he oriented in the specific direction in the X-Y plane

88. Enter `25.0` as the **Magnitude** of the perturbation

89. Push **Add Perturbation**

    Verify that “1 perturbations on 1 grids” appears at the bottom of the panel.

90. Push the **Select None** button

91. Enter `6` in the **Select by Grid ID** field and push the **Add** button

    Verify that “1 grids selected” is shown under the grid selection panel. Otherwise, deselect all grids and repeat selection.

92. Push the **By 2 Grids...** button

93. Enter `6, 3` in the **Select by Grid ID** field and press the `Enter` button on the keyboard

94. Push **Next>**

95. Delete the existing value for **Z**

96. Enter `25.0` as the **Magnitude** of the perturbation
97. Push **Add Perturbation**

    Verify that “2 perturbations on 2 grids” appears at the bottom of the panel.
98. Push the **Select None** button
99. Enter `5` in the **Select by Grid ID** field and push the **Add** button

    Verify that “1 grids selected” is shown under the grid selection panel. Otherwise, deselect all grids and repeat selection.
100. Push the **By 2 Grids...** button
101. Enter `5, 2` in the **Select by Grid ID** field and press the `Enter` button on the keyboard
102. Push **Next>**
103. Delete the existing value for **Z**
104. Enter `25.0` as the **Magnitude** of the perturbation
105. Push **Add Perturbation**

     Verify that “3 perturbations on 3 grids” appears at the bottom of the panel.
106. Push the **Select None** button
107. Push **Finish**
108. Repeat steps 73 to 102 to apply two additional raw morphing sets using the grids from the table below.

Raw Morphing Set Name	Perturbated grids ID	Direction of the perturbation grid ID	Magnitude of the perturbation
Horizontal2	7	7,4	20.0
	8	8,5	20.0
	9	9,6	20.0
Horizontal3	10	10,7	15.0
	11	11,8	15.0
	12	12,9	15.0

Checking Raw Morphing Perturbations

109. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
110. Select the **Oscillate** radio button

111. Under the **Deform Mesh**, select a **Shape Morphing Set Preview**

 Push **Static**, **Oscillate**, and **Ramp** buttons to test your preference. After the candidate shape change is confirmed as you intended, you will optimize the structure.

112. Push the **Up** button

Define the Design Objective

113. From the **Design** category chooser, select **Objectives**
114. Push the **New Objective** button from the Edit menu toolbar
115. Enter `Mass` for the Name
116. Make sure the **Mass** radio button is selected
117. Push **Finish**

Define a Synthetic Response for Sum of Displacements

118. From the **Design** category chooser, select **Synthetic Responses**
119. Push the **New Synthetic Response** button from the Edit menu toolbar
120. Enter `Displacement_Sum` for the Name
121. Make sure the **User Function (DRESP2)** radio button is selected
122. Push **Next>**
123. Push the **+** button to add arguments of the equation
124. Make sure the **Fundamental Response...** radio button is selected
125. Push **Next>**
126. Select the **Displacement** radio button
127. Push **Next>**
128. Push the **Select None** button
129. Select one of the three grids on the top of the tower from the viewport
130. Select the **Translation Magnitude** radio button
131. Push **Next>**
132. Select the existing loadcase
133. Push **Next>**
134. Repeat steps 123 to 128 to add two more arguments for the displacements for the other two grids on the top of the tower

135. Enter `F=Arg1+Arg2+Arg3` for the equation
136. Push the **Finish** button

Define the Displacement Design Constraint

137. From the **Design** category chooser, select **Constraints**
138. Push the **New Constraint** button from the Edit menu toolbar
139. Select the **Synthetic Response** radio button
140. Enter `80.0` as **Upper Bound**
141. Push **Next>**
142. Select the existing synthetic response defined earlier
143. Push **Next>**
144. Select the existing loadcase
145. Push **Finish**

Set the Genesis Options

146. From the main menu bar, select **Genesis** → **Options...**
147. Select the **Output Control** tab
148. Select the **Analysis Output** checkbox and select **First & Last** for the option
149. Select the **Design Output** checkbox and select **First & Last** for the option
150. Select the **File Control** tab
151. Select the **Element Sizing File** checkbox and select **Create** for the option
152. Push the **Apply** button

Optimizing and Viewing Results

153. From the main menu bar, select **Genesis** → **Optimize**
154. When the optimization is successfully done, from the **Genesis Console Output** window select the **Import Post...** button
155. Select `SHDSG021_dsg.SHP` from the **New Post-Processing Files** window

    You can view individual frames with **Deform Mesh/Color Mesh** or you can make an animation with **Animation**. You can also animate shape changes and element results (e.g. stresses) simultaneously.

156. Select `SHDSG021_dsgxx.pch` punch result files

    Make sure to all the punch files are selected

157. Select `SHDSG021_dsgOPOSTxx.pch` punch result files
158. Push the **Import** button
159. When done, push the **Close** button in the **Genesis Console Output** window

Post-Processing the Results (Shape Changes)

160. Select the **Post** tab
161. Push the **Deform/Mesh Color Mesh** button
162. Select a Shape Change for the last design cycle
163. Push the **Filled Contours** radio button
164. Select a Shape Change for the last design cycle in the **Color Mesh** frame
165. Push the **Up** button

Post-Processing the Cross-sectional Area Results

166. Select the **Post** tab
167. Push the **Deform/Mesh Color Mesh** button
168. Push the **Filled Elements** radio button
169. In the **Color Mesh** frame, select `Cycle 0 PROD Area` to display area distribution for the initial design cycle
170. Select PROD Area for the final design cycle
171. Push the **Options..** button for **Color Mesh**
172. Select the **Hide Elements with No Value** Checkbox

    Notice that the BAR domains are not longer displayed in the Viewport

173. Push the **Close** button
174. From the listbox, change Value to Value Original and notice the fractional change in the areas
175. Select any element to display its value in the Design Studio Messages window
176. Push the **Up** button

Quit Design Studio

177. From the main menu bar, select **File** → **Quit**

178. Push the **Don't Save** button

Study the Results

179. Open the output file. Study the values of the objective and the constraint and complete the following table

Type	Initial Value	Final Value	change%
Objective			
Maximum Constraint Violations			

CHAPTER 7

Topography Optimization Examples

- **Simple Topography Optimization Setup**
- **Allowing Grid Movement in One Direction**
- **Design with Manufacturing Constraints - Extrusion, Symmetry, Cyclic, Axisymmetric Constraints**
- **Bead Fraction - Constraining Grid Movement**
- **Modify FE mesh based on Topography Optimization Result**
- **Design With and Without Bead Fraction Constraint**
- **Designing Shape of Solids with Topography**

7.1 Simple Topography Optimization Setup

Introduction

The purpose of this example is to learn how to create and solve a simple topography optimization problem. This example will review how to post-process the shape changes.

The following objective will be used:

Minimize Strain Energy

Subject to:

Volume $\leq$ 735.0

Example ID

`TGDSG001`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`TGDSG001.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with an static loadcase.
`TGDSG001_dsg.dat`	`Input data`	Created: Genesis input file including the topography optimization data
`TGDSG001_ref.dat`	`Input data`	Provided: File ready to be optimized. Almost identical to `TGDSG001_dsg.dat`
`TGDSG001_dsg.SHP`	`Shape Change file`	Created: Output file containing the shape changes at each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TGDSG001.dat`

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**

4. Study the existing torsional loading

Create Topography Optimization Data

5. From the **Design** category chooser, select **Topography**
6. Select PSHELL 4
7. Push the **Modify Topography Design** button from the Edit Menu toolbar
8. For Maximum Height, enter `0.8`
9. Push the **Finish** button

   Verify that there is a hammer icon next to the PSHELL label.
   Here, the hammer icon indicates that the grids associated with the PSHELL are being designed

Create the Objective

10. From the **Design** category chooser, select **Objectives**
11. Push the **New Objective** button from the Edit Menu toolbar
12. Enter `Strain Energy` for the name
13. Select **Strain Energy** for the **Response type**
14. Push **Next>**
15. Select the existing loadcase `TORSION LOAD`
16. Push the **Finish** button

Create the Constraint

17. From the **Design** category chooser, select **Constraints**
18. Push the **New Constraint** button from the Edit Menu toolbar
19. Enter `Volume` for the name
20. Select **Volume** for the **Response type**
21. Enter `735.0` for the Upper Bound
22. Push the **Finish** button

Request the Shape File to be Output

23. From the main menu bar, select **Genesis** → **Options...**
24. Select the **File Control** tab

25. For **Shape Change File**, choose **Create** option
26. Push the **Apply** button

Optimize the Structure Using Genesis

27. From the main menu bar, select **Genesis** → **Optimize**
28. Study the **Design History**, when done push the **Close** button
29. Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

30. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
31. Select the `TGDSG001_dsg.SHP` file.
32. Push the **Open** button

Postprocessing the Results

33. Select the **Post** tab
34. Push the **Deform Mesh/Color Mesh** button
35. Select a shape change result for any design cycle
36. For the **Color Mesh**, select the **Filled Contours** radio button
37. Select a shape change result for any design cycle for the color plot in the Viewport
38. Select **Normal to Surface** in the listbox near the **Options..** button
39. Study the shape in the final design cycle

    Notice that the grids are moved in both the positive as well as the negative z-direction to create the beads. This is because by default, the design variables controlling the grids are allowed to move in both directions.

40. Push the **Up** button

Quit Design Studio

41. From the main menu bar, select **File** → **Quit**
42. Push the **Don’t Save** button

7.2 Allowing Grid Movement in One Direction

Introduction

The purpose of this example is to learn how to create and solve a simple topography optimization problem. Also the topography data is set such that the grids are moved in one direction along the normal instead of both directions. This example will review how to post-process and to change the default maximum number of cycles for the optimization.

The following objective will be used:

Minimize Strain Energy

Subject to:

Volume $\leq$ 735.0

Example ID

`TGDSG002`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`TGDSG002.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with an static loadcase.
`TGDSG002_dsg.dat`	`Input data`	Created: Genesis input file including the topography optimization data
`TGDSG002_ref.dat`	`Input data`	Provided: File ready to be optimized. Same as `TGDSG002_dsg.dat`
`TGDSG002_dsg.SHP`	`Shape Change file`	Created: Output file containing the shape changes at each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TGDSG002.dat`

Review the Objective

3. From the **Design** category chooser, select **Objectives**
4. Select the existing objective
5. Push the **Modify Objective** button from the Edit Menu toolbar
6. Study the objective definition by going through the trail
7. When done, push the **Cancel** button

Review the Constraint

8. From the **Design** category chooser, select **Constraints**
9. Select the existing constraint
10. Push the **Modify Constraint** button from the Edit Menu toolbar
11. Study the constraint definition by going through the trail
12. When done, push the **Cancel** button

Check the Element Norms

13. Select the **Analysis** tab
14. From the category chooser, select **Elements**
15. From the Viewport, select all the displayed elements
16. Push the **Generate Orientation Vectors** button

    Verify that the orientation vectors of the elements point upward in the positive z-direction.
17. Right click on the Viewport window and select **Clear** → **All** to clear the selection

Create Topography Optimization Data

18. From the **Design** category chooser, select **Topography**
19. Select PSHELL 4
20. Push the **Modify Topography Design** button from the Edit Menu toolbar
21. For **Maximum Height**, enter `0.8`
22. Enter `0.0` for the Lower Bound

    Doing this would bound the design variable that controls the grid movement to only take positive values. As the element normals are in the positive z-direction, the grids are allowed to only move in that direction.

23. Push the **Advanced** button
24. For **Initial Value**, enter `0.1`
25. Push **Next>**
26. Push the **Finish** button

    Verify that there is a hammer icon next to the PSHELL label.
    Here, the hammer icon indicates that the grids associated with the PSHELL are being designed

Request the Shape File to be Output

27. From the main menu bar, select **Genesis** → **Options...**
28. Select the **File Control** tab
29. For **Shape Change File**, choose **Create** option
30. Push the **Apply** button

Increase Maximum Number of Design Cycles

31. From the main menu bar, select **Genesis** → **Options...**
32. Select the **Design Control** tab
33. For **Maximum Design Cycles**, enter `30`
34. Push the **Apply** button

Optimize the Structure Using Genesis

35. From the main menu bar, select **Genesis** → **Optimize**
36. Study the **Design History**, when done push the **Close** button
37. Study the **Genesis Console Output** window

Import the Shape Changes Post Processing File

38. From the **Genesis Console Output** window, select **Import Post...** button
39. Select the `TGDSG002_dsg.SHP` file.
40. Push the **Import** button
41. From the **Genesis Console Output** window, select **Close** button

Postprocessing the Results

42. Select the **Post** tab
43. Push the **Deform Mesh/Color Mesh** button
44. For the **Color Mesh**, select the **Filled Contours** radio button
45. Select a shape change result for any design cycle for the color plot in the Viewport
46. Select **U3 Component** in the listbox near the **Options..** button

    Notice that there is no change in the viewport image as in this case, the XYZ Magnitude is the same as the U3 Component because the grids only move in the positive z-direction.

47. Push the **Up** button

Quit Design Studio

48. From the main menu bar, select **File** → **Quit**
49. Push the **Don’t Save** button

7.3 Design with Manufacturing Constraints - Extrusion, Symmetry, Cyclic, Axisymmetric Constraints

Introduction

The purpose of this example is to learn how to create and solve a simple topography optimization problem and also enforce different types of manufacturing constraints on the bead formation.

The analysis model contains 6 individual plate structures that are fixed along one of its two short sides and has to carry a bending load on its opposite sides. A different set of manufacturing constraints is enforced on each of the plates. They are designed to be as stiff as possible. By default, the edges are not designed in topography. The user has the option to select whether to design the edges or not. In one of the plates, the edge of the plate is also designed. Only the four corner grids are not designed in this plate.

The following objective will be used:

Minimize Strain Energy

Subject to:

Volume of each plate $\leq$ 175.0

Example ID

`TGDSG003`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`TGDSG003.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with an static loadcase.
`TGDSG003_dsg.dat`	`Input data`	Created: Genesis input file including the topography optimization data
`TGDSG003_ref.dat`	`Input data`	Provided: File ready to be optimized. Same as `TGDSG003_dsg.dat`
`TGDSG003_dsg.SHP`	`Shape Change file`	Created: Output file containing the shape changes at each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TGDSG003.dat`

Review the Objective

3. From the **Design** category chooser, select **Objectives**
4. Select the existing objective
5. Push the **Modify Objective** button from the Edit Menu toolbar
6. Study the objective definition by going through the trail
7. When done, push the **Cancel** button

Review the Six Volume Constraints

8. From the **Design** category chooser, select **Constraints**
9. Select one of the existing constraints
10. Push the **Modify Constraint** button from the Edit Menu toolbar
11. Study the constraint definition by going through the trail
12. When done, push the **Cancel** button

    Notice that the volume of each of the properties is constraints rather than the whole model.

Create Topography Data - No Constraints

13. From the **Design** category chooser, select **Topography**
14. Select PSHELL 9
15. Push the **Modify Topography Design** button from the Edit Menu toolbar
16. For **Shape type**, select **Line**
17. For **Maximum Height**, enter `0.8`

18. Push the **Advanced** button
19. For **Initial Value**, enter `0.1`
20. Push **Next>**
21. Push the **Finish** button

    Verify that there is a hammer icon next to the PSHELL label.
    Here, the hammer icon indicates that the grids associated with the PSHELL are being designed

Create Topography Data - Triple Symmetry + Design Edges

22. From the **Design** category chooser, select **Topography**
23. Select PSHELL 4
24. Push the **Modify Topography Design** button from the Edit Menu toolbar
25. For **Shape type**, select **Cone**
26. For **Maximum Height**, enter `0.8`
27. For **Minimum Width**, enter `0.1`
28. For **Edge Grid Design**, select **Design Edges**
29. Push the **Change** button to define the **Symmetry Coord. Sys.**
30. Select `PSHELL_4` from the list of coordinate systems
31. Push **Next>**
32. For **Symmetry 1**, select **MYZ: Mirror about YZ plane**
33. For **Symmetry 2**, select **MZX: Mirror about XZ plane**
34. For **Symmetry 3**, select **MXY: Mirror about XY plane**
35. Push the **Advanced** button
36. For **Initial Value**, enter `0.1`
37. Push **Next>**
38. Push the **Finish** button

Create Topography Data - Extrusion + Symmetry Constraints

39. From the **Design** category chooser, select **Topography**
40. Select PSHELL 5
41. Push the **Modify Topography Design** button from the Edit Menu toolbar
42. For **Shape type**, select **Cone**

43. For **Maximum Height**, enter `0.8`
44. For **Minimum Width**, enter `0.1`
45. For **Edge Grid Design**, select **Design Edges**
46. Push the **Change** button to define the **Symmetry Coord. Sys.**
47. Select `PSHELL_5` from the list of coordinate systems
48. Push **Next>**
49. For **Symmetry 1**, select **EX: Extrude along X axis**
50. For **Symmetry 2**, select **MZX: Mirror about XZ plane**
51. Push the **Finish** button

Create Topography Data - Axisymmetry Constraint

52. From the **Design** category chooser, select **Topography**
53. Select PSHELL 7
54. Push the **Modify Topography Design** button from the Edit Menu toolbar
55. For **Shape type**, select **Cone**
56. For **Maximum Height**, enter `0.8`
57. For **Minimum Width**, enter `1.0`
58. For **Spread**, enter `1.2`
59. For **Edge Grid Design**, select the **Default** option
60. Push the **Change** button to define the **Symmetry Coord. Sys.**
61. Select `PSHELL_7` from the list of coordinate systems
62. Push **Next>**
63. For **Symmetry 1**, select **CZ: Cyclic about Z axis**
64. For **No of Cyclic Sections**, enter `0`
65. Push the **Finish** button

Create Topography Data - Cyclic Symmetry Constraint

66. From the **Design** category chooser, select **Topography**
67. Select PSHELL 6
68. Push the **Modify Topography Design** button from the Edit Menu toolbar
69. For **Shape type**, select **Cone**

70. For **Maximum Height**, enter `0.8`
71. For **Minimum Width**, enter `1.0`
72. For **Spread**, enter `1.2`
73. For **Edge Grid Design**, select the **Default** option
74. Push the **Change** button to define the **Symmetry Coord. Sys.**
75. Select `PSHELL_6` from the list of coordinate systems
76. Push **Next>**
77. For **Symmetry 1**, select **CZ: Cyclic about Z axis**
78. For **No of Cyclic Sections**, enter `4`
79. Push the **Finish** button

Create Topography Data - Cyclic Symmetry + Design Edges

80. From the **Design** category chooser, select **Topography**
81. Select PSHELL 8
82. Push the **Modify Topography Design** button from the Edit Menu toolbar
83. For **Shape type**, select **Cone**
84. For **Maximum Height**, enter `0.8`
85. For **Minimum Width**, enter `1.0`
86. For **Spread**, enter `1.2`
87. For **Edge Grid Design**, select **Design Edges**
88. Push the **Change** button to define the **Symmetry Coord. Sys.**
89. Select `PSHELL_8` from the list of coordinate systems
90. Push **Next>**
91. For **Symmetry 1**, select **CZ: Cyclic about Z axis**
92. For **No of Cyclic Sections**, enter `4`
93. Push **Next>**
94. Select the 4 corner grids of the plate from the Viewport

    While doing topography, the user has the option to select grids that are not designed. The four grids selected here are not designed but the remaining grids along the edges are designed.

95. Push the **Finish** button

Optimize the Structure Using Genesis

96. From the main menu bar, select **Genesis** → **Optimize**
97. Study the **Design History**, when done push the **Close** button
98. Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

99. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
100. Select the `TGDSG003_dsg.SHP` file.
101. Push the **Open** button

Postprocessing the Results

102. Select the **Post** tab
103. Push the **Deform Mesh/Color Mesh** button
104. For the **Color Mesh**, select the **Filled Contours** radio button
105. Select a shape change result for any design cycle for the color plot in the Viewport
106. Select **U3 Component** in the listbox near the **Options..** button

    Notice that there is no change in the viewport image as in this case, the XYZ Magnitude is the same as the U3 Component because the grids only move in the positive z-direction.

107. Push the **Up** button

Quit Design Studio

108. From the main menu bar, select **File** → **Quit**
109. Push the **Don’t Save** button

7.4 Bead Fraction - Constraining Grid Movement

Introduction

The purpose of this example is to learn how to perform topography optimization with bead fraction constraints. This example will review how to post-process the shape changes.

Bead Fraction (BEADFR) is used to constrain the movement of the grids to a defined fraction.

The following objective will be used:

Minimize Strain Energy

Example ID

`TGDSG004`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`TGDSG004.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with an static loadcase.
`TGDSG004_dsg.dat`	`Input data`	Created: Genesis input file including the topography optimization data
`TGDSG004_ref.dat`	`Input data`	Provided: File ready to be optimized. Nearly identical as `TGDSG004_dsg.dat`
`TGDSG004_dsg.SHP`	`DSG file`	Created: Output file containing the shape changes at each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TGDSG004.dat`

Review the Torsional loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase

5. Notice boundary conditions and loading in the Viewport
6. From the main menu, select **Edit** → **Deselect All** to clear the loadcase selection

Check the Element Norms

7. From the **Analysis** category chooser, select **Elements**
8. From the Viewport, select all the displayed elements
9. Push the **Generate Orientation Vectors** button

   Verify that the orientation vectors of the elements point upward in the positive z-direction.
10. Right click on the Viewport window and select **Clear** → **All** to clear the selection

Set up Topography Optimization Data

11. From the **Design** category chooser, select **Topography**
12. Select PSHELL 4
13. Push the **Modify Topography Design** button from the Edit Menu toolbar
14. For **Maximum Height**, enter `0.8`
15. For **Lower Bound**, enter `0.0`

    Doing this would bound the design variable that controls the grid movement to only take positive values. As the element normals are in the positive z-direction, the grids are allowed to only move in that direction.
16. Push the **Change** button for **Symmetry Coordinate Sys.**
17. Select the coordinate system with label `Coord_1`
18. Push **Next>**
19. From the **Symmetry 1**: pull down menu, select **MZX: Mirror about XZ plane**
20. Enter `0.25` for the **Bead Fraction**
21. Push the **Advanced** button
22. Enter `0.3` for the **Initial Value**
23. Push **Next>**
24. Push the **Finish** button

    Verify that there is a hammer icon next to the PSHELL label.
    Here, the hammer icon indicates that the grids associated with the PSHELL are also being designed

Create the Objective

25. From the **Design** category chooser, select **Objectives**
26. Push the **New Objective** button from the Edit Menu toolbar
27. Select **Strain Energy** for the **Response type**
28. Push **Next>**
29. Select the existing loadcase `TORSION LOAD`
30. Push the **Finish** button

Request the Shape File to be Output

31. From the main menu bar, select **Genesis** → **Options...**
32. Select the **File Control** tab
33. For **Shape Change File**, choose **Create** option
34. Push the **Apply** button

Save the Design Studio file

35. From the main menu bar, select **File** → **Save As...**
36. Enter `TGDSG004` as the Filename and push **Save (**as a Design Studio File)

Optimize the Structure Using Genesis

37. From the main menu bar, select **Genesis** → **Optimize**
38. Study the **Design History**, when done push the **Close** button
39. Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

40. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
41. Select the `TGDSG004_dsg.SHP` file.
42. Push the **Open** button

Postprocessing the Results

43. Select the **Post** tab
44. Push the **Deform Mesh/Color Mesh** button
45. Select a shape change result for any design cycle

46. To animate, push the **Oscillate** radio button
47. Study the shape in the final design cycle
48. Push the **Up** button

Quit Design Studio

49. From the main menu bar, select **File** → **Quit**
50. Push the **Don’t Save** button

7.5 Modify FE mesh based on Topography Optimization Result

Introduction

The purpose of this example is to learn how to generate a updated finite element mesh with part of the beads produced in the topography optimization.

A topography optimization problem is provided. The optimized shape is comprised of three distinct beads. One along the diagonal of the plate and two along the edges. An FE mesh is created with only the bead along the diagonal of the plate.

Example ID

`TGDSG005`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`TGDSG005.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with an static loadcase and topography optimization data.
`TGDSG005_dsg.SHP`	`DSG file`	Created: Output file containing the shape changes at each design cycle
`TGDSG005_new.dat`	`Input data`	Created: Updated Genesis input file of the FE mesh with only the diagonal bead
`TGDSG005_ref.dat`	`Input data`	Provided: Reference File. Same as `TGDSG005_new.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TGDSG005.dat`

Optimize the Structure Using Genesis

3. From the main menu bar, select **Genesis** → **Optimize**
4. Study the **Design History**, when done push the **Close** button
5. Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

6. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
7. Select the `TGDSG005_dsg.SHP` file.
8. Push the **Open** button

Viewing the Shape Changes

9. Select the **Post** tab
10. Push the **Deform Mesh/Color Mesh** button
11. Select a shape change result for any design cycle
12. In **Color Mesh**, select the **Filled Contours** radio button
13. Select the shape change result for the final design cycle

    Do not push the **Up** button. This would allow for the result contour to be still present in the Viewport window.

Moving Grids based on Shape Results

14. Select the **Analysis** tab
15. From the category chooser, select **Grids**
16. From the Viewport, select the **Iso Front-Left-Top** view icon
17. Right click in the Viewport and select **Polygon Region** from the menu to select the grids
18. Create a polygon by clicking on the Viewport to enclose the bead along the diagonal
19. When the entire bead is enclosed, right click again and select **Close Polygon**
20. Push the **Modify Grids** button from the Edit Menu toolbar
21. Select **Apply Deform Result Dataset** radio button
22. Push **Next>**
23. Select the shape change result for the last design cycle

    User has the option of scaling the shape change by using the Scale Factor field.
24. Push the **Finish** button

    Notice the grids along the diagonal bead are modified to create the FE mesh that resembles the optimization result.

Export the Input File

25. From the main menu bar, select **File** → **Export** → **Input Data...**
26. Enter `TGDSG005_new` and push the **Save** button

    This input file can be used to perform further analysis/optimization.

Quit Design Studio

27. From the main menu bar, select **File** → **Quit**
28. Push the **Don’t Save** button

7.6 Design With and Without Bead Fraction Constraint

Introduction

The purpose of this example is to learn how to perform topography optimization. This example will review how to post-process shape changes.

This example goes through the steps for creating topography optimization data. Two different topography optimization problems are solved. In the first problem, all the grids are allowed to move while in the second problem only a percentage of the grids are allowed to move using Bead Fraction (BEADFR).

The following objective will be used:

Minimize Strain Energy

Example ID

`TGDSG006`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`TGDSG006.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with an static loadcase
`TGDSG006_dsg.dat`	`Input data`	Created: Genesis input file of the FE mesh along with the optimization data
`TGDSG006_dsg.SHP`	`DSG file`	Created: Output file containing the shape changes at each design cycle
`TGDSG006_ref1.dat`	`Input data`	Provided: Reference File for the case without Bead Fraction constraint
`TGDSG006_ref2.dat`	`Input data`	Provided: Reference File for the case with Bead Fraction constraint

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TGDSG006.dat`

Study the Model

3. From the **Display** tab, push **Show/Hide Groups**
4. Select one of the PSHELLs to show/hide the group
5. Study the three PSHELL groups in the model

   PSHELL 286 comprises of the external hood layer
   PSHELL 288 is the layer that is designed using Topography to determine the bead pattern that produces maximum stiffness for the hood.
   PSHELL 287 is a layer that connects PSHELL 286 and PSHELL 288
6. Push the **Show All** button
7. Push the **Up** button

Check the Loading on the Model

8. From the **Analysis** category chooser, select **Loadcases**
9. Select one of loadcase.
10. Study the loading and boundary conditions in the Viewport
11. Select the other loadcase and study its loading conditions in the Viewport
12. From the main menu, select **Edit** → **Deselect All** to clear the loadcase selection

Set up Topography Optimization Data

13. From the **Design** category chooser, select **Topography**
14. Select PSHELL 288
15. Push the **Modify Topography Design** button from the Edit Menu toolbar
16. For **Maximum Height**, enter `25.0`
17. For **Minimum Width**, enter `35.0`
18. For **Spread**, enter `10.0`
19. For **Upper Bound**, enter `0.0`

    The upper bound is constrained to be 0.0 to restrict the beads to grow only in one direction (i.e. negative Z-direction)
20. Push the **Finish** button

    Verify that there is a hammer icon next to the PSHELL label.
    Here, the hammer icon indicates that the grids associated with the PSHELL are being designed

Create the Objective

21. From the **Design** category chooser, select **Objectives**
22. Push the **New Objective** button from the Edit Menu toolbar
23. For **Name**, enter `Strain Energy`
24. Select the **Strain Energy** radio button
25. Push **Next>**
26. Select both the loadcases using the **Shift** key
27. Push **Finish**

    Verify that there are two responses in the objectives list one for each loadcase

Request the Shape Post Processing File

28. From the main menu bar, select **Genesis** → **Options...**
29. Select the **File Control** tab
30. For **Shape Change File**, choose **Create** option
31. Push the **Apply** button

Optimize the Structure Using Genesis

32. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History**, when done push the **Close** button
    Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

33. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
34. Select the `TGDSG006_dsg.SHP` file.
35. Push the **Open** button

Postprocessing the Results

36. From the **Display** tab, push **Show/Hide Groups**

    To view the shape changes of PSHELL 288 easily, hide PSHELL 286 and PSHELL 287
37. To hide the groups, select PSHELL 286 and PSHELL 287
38. Select the **Post** tab
39. Push the **Deform Mesh/Color Mesh** button

40. From the **Deform Mesh**, select the shape change to view the shape in the specific cycle
41. For the **Color Mesh**, select the **Filled Contours** radio button
42. Select a shape change result for any design cycle

    Notice that maximum stiffness is achieved when most of the grids are perturbed to the maximum height. Even though this design provides maximum stiffness this might not be practical. So we impose a bead fraction constraint to limit the movement of the grids.

43. Push the **Up** button

Setting the Bead Fraction Constraint

44. From the **Design** category chooser, select **Topography**
45. Select PSHELL 288
46. Push the **Modify Topography Design** button from the Edit Menu toolbar
47. For **Bead Fraction**, enter `0.35`
48. Push the **Advanced.**. button
49. For **Initial Value**, enter `-1.0`
50. For **Bead Fraction Bound**, select **Both** from the listbox

    Selecting **Both** would result in exactly 35% of the grids to move
    Selecting **Lower** would result in the average of all the grid perturbations to be atleast 35%
    Selecting **Upper** would result in the average of all the grid perturbations to be atmost 35%

51. Push **Next>**
52. Push **Finish**

Request the Updated Input File

53. From the main menu bar, select **Genesis** → **Options...**
54. Select the **File Control** tab
55. For **Updated Input File**, choose **Last Cycle** option
56. Push the **Apply** button

Optimize the Structure Using Genesis

57. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History**, when done push the **Close** button
    Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

58. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
59. Select the `TGDSG006_dsg.SHP` file.
60. Push the **Open** button

Postprocessing the Results

61. Select the **Post** tab
62. Push the **Deform Mesh/Color Mesh** button

    The data imported from the new SHP file are loaded in addition to the shape change data that imported earlier.
    The order of the data-sets is the order in which you import the files.

63. For the **Color Mesh**, select a shape change result for final design cycle

    Notice the difference in the final shape between the first and second optimizations. In the first run, most of the grids went to the maximum perturbation while in the second run, only a fraction of the grids moved.

64. Push the **Up** button

Import the UPDATE file

65. From the main menu, select **File** → **New**
66. Push the **Don't Save** button
67. From the main menu, select **File** → **Import** → **Input Data...**
68. Select the `TGDSG006_dsgUPDATExx.dat` file where `xx` is the last design cycle number.
69. Push the **Open** button
70. Study the FE mesh with the updated shape.

    Notice that this file has only the updated bulk data and lacks the executive and solution control data.

Close Design Studio

71. From the main menu, select **File** → **Quit**
72. Push the **Don't Save** button

7.7 Designing Shape of Solids with Topography

Introduction

The purpose of this example is to learn how topography optimization can be used to design the shape of a structure modeled with solid elements. A cantilever beam modeled with solid elements is used as an analysis model. First a shell skin is created and unwanted elements of the skin are deleted. The remaining skin elements are used in the topography optimization.

The trick to designing solids with topography optimization is to create a thin skin made of shell elements on the solid elements to be designed and use mesh smoothing to move the interior grids to avoid distortions. The added skin is thin so it does not affect the stiffness of the model. As the skin elements and the solids share the same grids, the shape of the solid changes with the change in the shape of the skin.

The following optimization problem is solved:

Minimize Mass

Subject to:

Grid Stresses $\leq$ 260.0

Fourth Fundamental Frequency $\geq$ 700.0

Length in y-direction @free end $\geq$ 10.0

Length in z-direction @free end $\geq$ 10.0

Example ID

`TGDSG007`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`TGDSG007.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with loadcases
`TGDSG007_dsg.dat`	`Input data`	Created: Genesis input file of the FE mesh along with the skin and optimization data

TGDSG007_dsg.SHP	DSG file	Created: Output file containing the shape changes at each design cycle
TGDSG007_dsgUPDATExx.dat	UPDATE file	Created: Output file containing the updated FE mesh for design cycle 'xx'
TGDSG007_ref.dat	Input data	Provided: Reference File for the case without Bead Fraction constraint

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `TGDSG007.dat`

Check the Loading on the Model

3. From the **Analysis** category chooser, select **Loadcases**
4. Select one of loadcases.
5. Study the loading and boundary conditions in the Viewport
6. Select the other loadcase and study its loading conditions in the Viewport

Create a New Property for the Skin Elements

7. From the **Display** tab, push **Manage Groups** button
8. Push the **New Group** button from the Edit Menu toolbar
9. Enter `Skin` for the name
10. For **Type**, select **PSHELL**
11. Select the green for the property color by clicking on the green color in the list
12. Push the **Finish** button
13. Push the **Up** button

Create Skin Elements

14. From the **Analysis** category chooser, select **Elements**
15. Push the **New Elements** button from the Edit Menu toolbar
16. Choose the **Surface Elements from Selected Solid Elements** radio button
17. Push **Next>**
18. Push the **Select All** button to select all the elements in the Viewport
19. Select the created PSHELL property from the list

20. Push the **Finish** button

    Notice that you cannot see the newly created shell elements.

View Created Skin Elements

21. From the **Display** tab, push **Show/Hide Groups** button
22. Select the PSOLID group to turn off the solid elements

    Note that you can now see the shell elements created in the previous step.
23. Select the RBE group to turn off the rigid elements

Delete Unwanted Skin Elements

24. Select the **Analysis** tab and select **Elements** from the category chooser
25. Push the **Select None** button to clear any element selection
26. From the Viewport, select the 12 elements that cover the two faces of the beam parallel to the YZ plane

    To select these 12 elements easily, select the YZ view icon in the viewport. Drag a box within the beam (not enclosing the beam) to select only the elements on the two faces.
    Make sure that “**12 elements selected**” is in the summary
    If more than 12 elements are selected, push the **Select None** button and try the above step again.
27. From the Edit menu toolbar, select **Delete Elements** button

Set up Topography Optimization Data

28. From the **Design** category chooser, select **Topography**
29. Select the PSHELL group of the skin elements
30. Push the **Modify Topography Design** button from the Edit Menu toolbar
31. For the **Shape Type**, select **Line**
32. For **Maximum Height**, enter `50.0`
33. For the **Edge Grid Design**, select **Design Edges**
34. Push **Next>**
35. Push **Select None** button to clear any grid selections.

    The non-designable grids are already defined as a grid set. Now we will go to the Analysis tab, choose the defined grid set to select the nondesignable grids. After the grids are selected, we will comeback to the Topography definition and complete the trail.

36. Go to the **Analysis** tab and select **Grid Sets** from the category chooser
37. Select the existing `NonDesigned Grids` grid set

    Note that grids are selected in the Viewport
38. Go to the **Design** tab

    Design Studio remembers where you left the Design tab. Notice that the summary says "**10 grids selected**" for the nondesignable grids
39. Push the **Finish** button

    Verify that there is a hammer icon next to the PSHELL label.
    Here, the hammer icon indicates that the grids associated with the PSHELL are being designed

View All Groups in the Model

40. From the **Display** tab, push **Show/Hide Groups** button
41. Push the **Show All** button to view all the groups in the model

Create the Objective

42. From the **Design** category chooser, select **Objectives**
43. Push the **New Objective** button from the Edit Menu toolbar
44. For **Name**, enter `Mass`
45. Select the **Mass** radio button as the response type
46. Push **Finish**

Create Design Constraint for Grid Stresses

47. From the **Design** category chooser, select **Constraints**
48. Push the **New Constraint** button from the Edit Menu toolbar
49. For **Name**, enter `GSTRESS`
50. Select the **More Response Types...** radio button as the response type
51. Enter `260.0` for the **Upper Bound**
52. Push **Next>**
53. Select the **Grid Stress** radio button as the response type
54. Push **Next>**
55. Push the **Select All** button to select all the grids in the model

56. Push **Next>**
57. Select all the existing loadcases using the **Shift** key
58. Push **Finish**

Create Design Constraint for Frequency

59. Push the **New Constraint** button from the Edit Menu toolbar
60. For **Name**, enter `Frequency`
61. Select the **Frequency Mode Number** radio button as the response type and enter `4` in the adjacent textbox for the mode number
62. Enter `700.0` for the **Lower Bound**
63. Push **Next>**
64. Select the existing frequency loadcase
65. Push **Finish**

Create Geometric (DRESPG) Design Constraints

66. Push the **New Constraint** button from the Edit Menu toolbar
67. For **Name**, enter `Length_Y`
68. Select the **More Response Types...** radio button as the response type
69. Enter `10.0` for the **Lower Bound**
70. Push **Next>**
71. Select the **Geometric** radio button as the response type
72. Push **Next>**
73. Select two corner grids on the free end along the y-direction for calculating the distance

    Alternatively, one can enter `130, 132` in the **Select by Grid ID** textbox and press enter
74. Select the **Length** radio button for the **Geometric responses** type
75. Push **Finish**
76. Push the **New Constraint** button from the Edit Menu toolbar
77. For **Name**, enter `Length_Z`
78. Select the **More Response Types...** radio button as the response type
79. Enter `10.0` for the **Lower Bound**

80. Push **Next>**
81. Select the **Geometric** radio button as the response type
82. Push **Next>**
83. Select two corner grids on the free end along the z-direction for calculating the distance

    Alternatively, one can enter `124, 132` in the **Select by Grid ID** textbox and press enter
84. Select the **Length** radio button for the **Geometric responses** type
85. Push **Finish**

Request the Shape and Updated Input File

86. From the main menu bar, select **Genesis** → **Options...**
87. Select the **File Control** tab
88. For **Shape Change File**, choose **Create** option
89. For **Updated Input File**, choose **Last Cycle** option
90. Push the **Apply** button

Optimize the Structure Using Genesis

91. From the main menu bar, select **Genesis** → **Optimize**

    Study the **Design History**, when done push the **Close** button
    Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

92. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
93. Select the `TGDSG007_dsg.SHP` file.
94. Push the **Open** button

Postprocessing the Results

95. Select the **Post** tab
96. Push the **Deform Mesh/Color Mesh** button
97. From the Deform Mesh, select the shape change to view the shape of the cycle
98. For the **Color Mesh**, select the **Filled Contours** radio button

99. Select a shape change result for any design cycle

    Notice the change in the shape of the solid beam due to change in the skin.

100. Push the **Up** button

Import the UPDATE file

101. From the main menu, select **File** → **New**
102. Push the **Don’t Save** button
103. From the main menu, select **File** → **Import** → **Input Data...**
104. Select the `TGDSG007_dsgUPDATExx.dat` file where `xx` is the last design cycle number.
105. Push the **Open** button
106. Study the FE mesh with the updated shape.

    Notice that this file has only the updated bulk data and lacks the executive and solution control data.

Close Design Studio

107. From the main menu, select **File** → **Quit**
108. Push the **Don’t Save** button

CHAPTER 8

Freeform Optimization Examples

- **On Raw Perturbations (DVGRID) - Mirror Symmetries**
- **On Raw Perturbations (DVGRID) - Manufacturing Constraints and Coarsening**
- **On Domain Perturbations (DVGRIDC) - Axisymmetry and Grid Fraction Constraints**
- **On Domain Perturbations (DVGRIDC) - Mirror Symmetry**
- **Design of a Connecting Rod - Using Domain Perturbations**
- **On Grid Perturbations Generated Using Domains**

8.1 On Raw Perturbations (DVGRID) - Mirror Symmetries

Introduction

The purpose of this example is to learn how to create and solve a simple freeform optimization problem. This example goes through the process of defining symmetries and grid fraction constraints on the freeform optimization problem. This example will review how to post-process the shape changes.

Grid Fraction is used to constrain the movement of the grids to a defined fraction.

The following objective will be used:

Minimize Strain Energy

Subject to:

Grid Fraction $\leq$ 0.45

Example ID

FFDSG001

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
FFDSG001.dat	Input data	Provided: This file is imported into Design Studio and contains the finite element mesh along with static loadcases.
FFDSG001_dsg.dat	Input data	Created: Genesis input file including the freeformoptimization data
FFDSG001_ref.dat	Input data	Provided: File ready to be optimized. Same as FFDSG001_dsg.dat
FFDSG001_dsg.SHP	Shape Change file	Created: Output file containing the shape changes at each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: FFDSG001.dat

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Study the three existing torsional loadcases

Create Shape Morphing Set with Freeform

5. From the **Design** category chooser, select **Shape Morphing Sets**
6. Push the **New Shape Set** button from the Edit Menu toolbar
7. Enter `Perturbations` for the **Name**
8. Select the **Raw Morphing Set** radio button
9. Make sure `New Variable [-1.0, 1.0]` is selected for the **Design Variable**
10. Select the **Yes** radio button for **Freeform**
11. Push **Next>**
12. Push **Select None** button

    Grid sets are created for the grids where the perturbations are applied. Now we will select the grid set to select the grids.

13. Select the **Analysis** tab
14. From the **Analysis** Category Chooser, select **Grid Sets**
15. Select `Grid Set Top` from the list
16. Select the **Design** tab

    Notice that 2325 grids are selected. Design Studio remembers selection items until they are deselected

17. Enter `0.0`, `0.0`, `-1.0` for **X**, **Y**, **Z** components of the direction
18. Enter `1.2` for the **Magnitude**
19. Push the **Add Perturbation** button

    Notice “2325 perturbations on 2325 grids” in the summary

20. Push **Select None** button

    Now perturbations will be applied on the bottom surface grids

21. Select the **Analysis** tab
22. From the **Analysis** Category Chooser, select **Grid Sets**
23. Select `Grid Set Bottom` from the list

24. Select the **Design** tab

    Notice that 2325 grids are selected. Design Studio remembers selection items until they are deselected

25. Enter `0.0`, `0.0`, `1.0` for **X**, **Y**, **Z** components of the direction
26. Enter `1.2` for the **Magnitude**
27. Push the **Add Perturbation** button

    Notice "4650 perturbations on 4650 grids" in the summary
    Rotate the model to notice that the pertubations are added to the top as well as the bottom grids.

28. Push **Next>**

    Symmetry conditions are now applied based on a coordinate system

29. Push the **Change** button for **Symmetry Coord. Sys**
30. Select `Center CoordSys` for the Coordinate System
31. Push **Next>**
32. For **Symmetry 1**, select **MYZ: Mirror about YZ plane** option
33. For **Symmetry 2**, select **MZX: Mirror about XZ plane** option
34. For **Symmetry 3**, select **MXY: Mirror about XY plane** option
35. Enter `0.45` for the **Grid Fraction** to define the grid fraction constraint
36. Push the **Finish** button

Preview the Perturbations

37. Select the **Post** tab
38. Push the **Deform Mesh/Color Mesh** button
39. Under the Deform Mesh, select a Shape Morphing Set Preview
40. Select the **Oscillate** radio button
41. Under the Color Mesh, select the **Filled Contours** radio button
42. Under the Color Mesh, select a Shape Morphing Set Preview

    Notice the values of the color bar. The maximum value is `1.2` which is the value of the perturbation

43. Push the **Up** button

Create the Objective

44. From the **Design** category chooser, select **Objectives**

45. Push the **New Objective** button from the Edit Menu toolbar
46. Enter `StrainEnergy` for the **Name**
47. Select **Strain Energy** for the **Response type**
48. Push **Next>**
49. Select all the existing loadcases by selecting **Edit** → **Select All** from the main menu
50. Push the **Finish** button

    Notice that three objectives are created. Each one defines the strain energy for each loadcase.

Set Maximum Design Cycles

51. From the main menu bar, select **Genesis** → **Options...**
52. Select the **Design Control** tab
53. For **Maximum Design Cycles**, enter `25`
54. Push the **Apply** button

Set Genesis Parameters

55. From the main menu bar, select **Genesis** → **Options...**
56. Select the **Design Control** tab
57. Push the **Advanced...** button
58. Select the **Misc.** tab
59. For **Mesh Smoothing (MSMOOTH)**, choose **Yes** option

    If **MSMOOTH = Yes**, then mesh smoothing is performed on 2D planar surfaces and 3D elements. If **MSMOOTH =No**, then mesh smoothing will not be performed.
60. Push the **Close** button
61. Push the **Apply** button

Optimize the Structure Using Genesis

62. From the main menu bar, select **Genesis** → **Optimize**
63. Study the **Design History**, when done push the **Close** button
64. Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

65. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
66. Select the `FFDSG001_dsg.SHP` file.
67. Push the **Open** button

Postprocessing the Results

68. Select the **Post** tab
69. Push the **Deform Mesh/Color Mesh** button
70. Select a shape change result for any design cycle
71. For the **Color Mesh**, select the **Filled Contours** radio button
72. Select a shape change result for any design cycle for the color plot in the Viewport
73. Study the shape in the final design cycle

    Notice that the final shape is symmetric in all three planes of the coordinate system in the center of the model.

74. Push the **Up** button

Quit Design Studio

75. From the main menu bar, select **File** → **Quit**
76. Push the **Don't Save** button

Study the Output File

77. In a text editor load the Genesis data file: `FFDSG001_dsg.out`
78. Study briefly the file
79. Using the output file, complete the following table:

	Reference Answer	**Obtained Answer**
Number of Design Variables	608	
Initial Objective	3.000	
Optimal Objective	0.645	

Note that due to triple symmetry, each design variable controls upto to 8 surface grids. In this problem there were 4650 designable surface grids.

8.2 On Raw Perturbations (DVGRID) - Manufacturing Constraints and Coarsening

Introduction

The purpose of this example is to learn how to create and solve a simple freeform optimization problem. This example goes through the process of defining axisymmetry and extrusion manufacturing constraints along grid fraction constraints on the freeform optimization problem. This example will review how to post-process the shape changes.

The following objective will be used:

Minimize Strain Energy

Subject to:

Grid Fraction $\leq$ 0.45

Example ID

`FFDSG002`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`FFDSG002.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with static loadcases.
`FFDSG002_dsg.dat`	`Input data`	Created: Genesis input file including the freeform optimization data
`FFDSG002_ref.dat`	`Input data`	Provided: File ready to be optimized. Same as `FFDSG002_dsg.dat`
`FFDSG002_dsg.SHP`	`Shape Change file`	Created: Output file containing the shape changes at each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `FFDSG002.dat`

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Study the six existing torsional loadcases **-** 3 on each structure

Create the Objective

5. From the **Design** category chooser, select **Objectives**
6. Push the **New Objective** button from the Edit Menu toolbar
7. Enter `StrainEnergy` for the **Name**
8. Select **Strain Energy** for the **Response type**
9. Push **Next>**
10. Select all the existing loadcases by selecting **Edit** → **Select All** from the main menu
11. Push the **Finish** button

    Notice that six objectives are created. Each one defines the strain energy for each loadcase.

Create Freeform Shape Morphing Set with Axisymmetry

12. From the **Design** category chooser, select **Shape Morphing Sets**
13. Push the **New Shape Set** button from the Edit Menu toolbar
14. Enter `Perturbation 1` for the **Name**
15. Select the **Raw Morphing Set** radio button
16. Make sure `New Variable [-1.0, 1.0]` is selected for the **Design Variable**
17. Select the **Yes** radio button for **Freeform**
18. Enter `1.2` for the **Maximum Perturbation**
19. Push **Next>**
20. Push **Select None** button

    Grid sets are created for the grids where the perturbations are applied. Now we will select the grid set to select the grids.
21. Select the **Analysis** tab
22. From the **Analysis** Category Chooser, select **Grid Sets**
23. Select `1-Grid Set Top` from the list
24. Select the **Design** tab

    Notice that 2325 grids are selected. Design Studio remembers selection items until they are un-selected

25. Enter `0.0`, `0.0`, `-1.0` for **X**, **Y**, **Z** components of the direction
26. Enter `0.5` for the **Magnitude**
27. Push the **Add Perturbation** button

    Notice “2325 perturbations on 2325 grids” in the summary

28. Push **Select None** button

    Now perturbations will be applied on the bottom surface grids

29. Select the **Analysis** tab
30. From the **Analysis** Category Chooser, select **Grid Sets**
31. Select `1-Grid Set Bottom` from the list
32. Select the **Design** tab

    Notice that 2325 grids are selected. Design Studio remembers selection items until they are un-selected

33. Enter `0.0`, `0.0`, `1.0` for **X**, **Y**, **Z** components of the direction
34. Enter `0.5` for the **Magnitude**
35. Push the **Add Perturbation** button

    Notice “4650 perturbations on 4650 grids” in the summary

36. Push **Next>**
37. For the **Coarsening Method**, select **Diameter** option
38. Enter `2.0` for the **Real Coarse Parameter**

    Symmetry conditions are now applied based on a coordinate system

39. Push the **Change** button for **Symmetry Coord. Sys**
40. Select `1-Center CoordSys` for the Coordinate System
41. Push **Next>**
42. For **Symmetry 1**, select **CZ: Cyclic about Z axis** option
43. For **Symmetry 2**, select **MXY: Mirror about XY plane** option
44. Enter `0` for the **No. of Cyclic Sections** to define the grid fraction constraint

    Entering a value of “0” for the number of cyclic sections implies that infinite cyclic sections are created thereby resulting in axisymmetry

45. Enter `0.45` for the **Grid Fraction**

    Grid Fraction is used to constrain the movement of the grids to a defined fraction.

46. Push the **Advanced...** button

47. Select **Both** option for the **Grid Fraction Bound**

    Upper option creates a upper bound constraint on grid fraction
    Lower option creates a lower bound constraint on grid fraction
    Both option creates both upper and lower bound constraints on grid fraction

48. Enter `0.05` for the **Grid Fraction Tolerance**

    Grid Fraction Tolerance is used to add a tolerance on the grid fraction constraint.

49. Push **Next>**
50. Push the **Finish** button

Create Freeform Shape Morphing Set with Extrusion

51. From the **Design** category chooser, select **Shape Morphing Sets**
52. Push the **New Shape Set** button from the Edit Menu toolbar
53. Enter `Perturbation 2` for the **Name**
54. Select the **Raw Morphing Set** radio button
55. Make sure `New Variable [-1.0, 1.0]` is selected for the **Design Variable**
56. Select the **Yes** radio button for **Freeform**
57. Enter `1.2` for the **Maximum Perturbation**
58. Push **Next>**
59. Push **Select None** button

    Grid sets are created for the grids where the perturbations are applied. Now we will select the grid set to select the grids.

60. Select the **Analysis** tab
61. From the **Analysis** Category Chooser, select **Grid Sets**
62. Select `2-Grid Set Top` from the list
63. Select the **Design** tab

    Notice that 2325 grids are selected. Design Studio remembers selection items until they are un-selected

64. Enter `0.0`, `0.0`, `-1.0` for **X**, **Y**, **Z** components of the direction
65. Enter `0.5` for the **Magnitude**
66. Push the **Add Perturbation** button

    Notice “2325 perturbations on 2325 grids” in the summary

67. Push **Select None** button

    Now perturbations will be applied on the bottom surface grids

68. Select the **Analysis** tab
69. From the **Analysis** Category Chooser, select **Grid Sets**
70. Select `2-Grid Set Bottom` from the list
71. Select the **Design** tab

    Notice that 2325 grids are selected. Design Studio remembers selection items until they are un-selected

72. Enter `0.0`, `0.0`, `1.0` for **X**, **Y**, **Z** components of the direction
73. Enter `0.5` for the **Magnitude**
74. Push the **Add Perturbation** button

    Notice "4650 perturbations on 4650 grids" in the summary

75. Push **Next>**
76. For the **Coarsening Method**, select **Diameter** option
77. Enter `2.0` for the **Real Coarse Parameter**

    Symmetry conditions are now applied based on a coordinate system

78. Push the **Change** button for **Symmetry Coord. Sys**
79. Select `2-Center CoordSys` for the Coordinate System
80. Push **Next>**
81. For **Symmetry 1**, select **EX: Extrude along X axis** option
82. For **Symmetry 2**, select **MZX: Mirror about XZ plane** option
83. For **Symmetry 3**, select **MXY: Mirror about XY plane** option
84. Enter `0.45` for the **Grid Fraction** to define the grid fraction constraint
85. Push the **Advanced...** button
86. Select **Both** option from the **Grid Fraction Bound**

    Upper option creates a upper bound constraint on grid fraction
    Lower option creates a lower bound constraint on grid fraction
    Both option creates both upper and lower bound constraints on grid fraction

87. Enter `0.05` for the **Grid Fraction Tolerance**
88. Push **Next>**
89. Push the **Finish** button

Preview the Shape Changes

90. Select the **Post** tab

91. Push the **Deform Mesh/Color Mesh** button
92. Under the Deform Mesh, select a Shape Morphing Set Preview
93. Select the **Oscillate** radio button
94. Under the Color Mesh, select the **Filled Contours** radio button
95. Under the Color Mesh, select a Shape Morphing Set Preview

    Notice the values of the color bar. The maximum value is 0.5 which is the value of the perturbation

96. Push the **Up** button

Set Maximum Design Cycles

97. From the main menu bar, select **Genesis** → **Options...**
98. Select the **Design Control** tab
99. For **Maximum Design Cycles**, enter `25`
100. Push the **Apply** button

Set Genesis Parameters

101. From the main menu bar, select **Genesis** → **Options...**
102. Select the **Design Control** tab
103. Push the **Advanced...** button
104. Select the **Misc.** tab
105. For **Mesh Smoothing (MSMOOTH)**, choose **Yes** option
106. For **Shape Control (SHAPECN)**, choose **Prevent DISTOR Errors** option
107. Push the **Close** button
108. Select the **Analysis Control** tab
109. Push the **Advanced...** button
110. For **SHAPECK**, enter `0`
111. Push the **Close** button
112. Push the **Apply** button

Optimize the Structure Using Genesis

113. From the main menu bar, select **Genesis** → **Optimize**

114. Study the **Design History**, when done push the **Close** button
115. Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

116. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
117. Select the `FFDSG002_dsg.SHP` file.
118. Push the **Open** button

Postprocessing the Results

119. Select the **Post** tab
120. Push the **Deform Mesh/Color Mesh** button
121. Select a shape change result for any design cycle
122. For the **Color Mesh**, select the **Filled Contours** radio button
123. Select a shape change result for any design cycle for the color plot in the Viewport
124. Study the shape in the final design cycle

    Notice that the maximum value final shape change.It is at 1.2 even though the perturbation is only 0.5. This is because value for **Maximum Perturbation** is used to scale the perturbations.

125. Push the **Up** button

Quit Design Studio

126. From the main menu bar, select **File** → **Quit**
127. Push the **Don’t Save** button

8.3 On Domain Perturbations (DVGRIDC) - Axisymmetry and Grid Fraction Constraints

Introduction

The purpose of this example is to learn how to create and solve a simple freeform optimization problem using domain perturbations. This example goes through the process of defining an axisymmetric constraint along with grid fraction constraints. This example will review how to post-process the shape changes.

The following objective will be used:

Minimize Strain Energy

Subject to:

Grid Fraction $\leq$ 0.6

Example ID

`FFDSG003`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`FFDSG003.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with static loadcases.
`FFDSG003_dsg.dat`	`Input data`	Created: Genesis input file including the freeform optimization data
`FFDSG003_ref.dat`	`Input data`	Provided: File ready to be optimized. Same as `FFDSG003_dsg.dat`
`FFDSG003_dsg.SHP`	`Shape Change file`	Created: Output file containing the shape changes at each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `FFDSG003.dat`

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Study the existing torsional loadcase

Create the Objective

5. From the **Design** category chooser, select **Objectives**
6. Push the **New Objective** button from the Edit Menu toolbar
7. Enter `StrainEnergy` for the **Name**
8. Select **Strain Energy** for the **Response type**
9. Push **Next>**
10. Select the existing loadcase
11. Push the **Finish** button

Create the Design Variable

12. From the **Design** category chooser, select **Design Variables**
13. Push the **New Design Variable** button from the Edit Menu toolbar
14. Enter `Shape1` for the **Name**
15. Make sure the **Independent Design Variable** radio button is selected
16. Push **Next>**
17. Enter `0.0` for the **Initial Value**
18. Enter `0.0` for the **Lower Bound**

    Creating the design variable with the lower bound at 0.0 would allow the shape to change only in the positive direction of the perturbation
19. Enter `1.0` for the **Upper Bound**
20. Push the **Finish** button

Review the Shape Domains

21. From the **Design** category chooser, select **Shape Domains**
22. Study the existing shape domains

Create Freeform Domain Morphing Set with Axisymmetry

23. From the **Design** category chooser, select **Shape Morphing Sets**
24. Push the **New Shape Set** button from the Edit Menu toolbar
25. Enter `Perturbations` for the **Name**
26. Make sure the **Domain Morphing Set** radio button is selected
27. Select `Shape1` for the **Design Variable**
28. Select the **Yes** radio button for **Freeform**
29. Enter `1.0` for the **Initial Randomness**

    Initial Randomness controls the value of the random values used for the initial design. An initial randomness of 1.0 means that the bounds on the random values is 10% of the bounds of the design variable. A value of 0.0 means that no random values are used for the initial design.

30. Push **Next>**
31. Select the existing domains by selecting **Edit** → **Select All** from the main menu
32. Push **Next>**

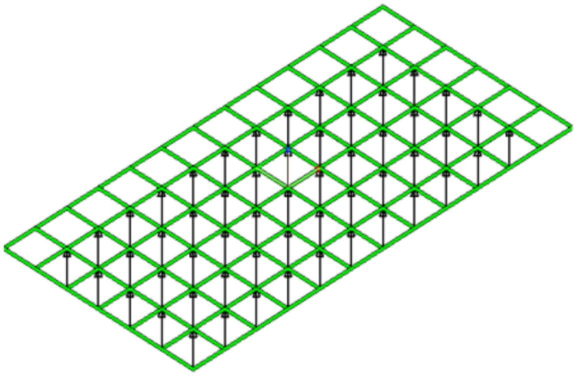
    Only the Grids that are corners of the domains need to be selected. To do so, the PSHELL group needs to be hidden.

33. Select the **Display** tab
34. Select the **Show/Hide Groups** button
35. Select PSHELL 22 from the list to hide the group
36. Select the **Design** tab
37. Push the **Select None** button
38. Select all the interior grids of the domains.

    Notice that 55 grids are selected

39. Enter `0.0`, `0.0`, `1.0` for **X**, **Y**, **Z** components of the direction
40. Enter `2.0` for the **Magnitude**
41. Push the **Add Perturbation** button

    Notice “55 perturbations on 55 grids” in the summary. The perturbations are shown in the figure below.



42. Push **Next>**

    Symmetry conditions are now applied based on a coordinate system

43. Push the **Change** button for **Symmetry Coord. Sys**
44. Select `Coord. Sys. 2` for the Coordinate System
45. Push **Next>**
46. For **Symmetry 1**, select **CZ: Cyclic about Z axis** option
47. Enter `0` for the **No. of Cyclic Sections**

    Entering a value of “0” for the number of cyclic sections implies that infinite cyclic sections are created thereby resulting in axisymmetry

48. Enter `0.6` for the **Grid Fraction** to define the grid fraction constraint
49. Push the **Finish** button

Preview the Shape Changes

50. Select the **Display** tab
51. Select the **Show/Hide Groups** button
52. Select PSHELL 22 from the list to show the group
53. Select the **Post** tab
54. Push the **Deform Mesh/Color Mesh** button
55. Under the Deform Mesh, select a Shape Morphing Set Preview
56. Select the **Oscillate** radio button
57. Under the Color Mesh, select the **Filled Contours** radio button
58. Under the Color Mesh, select a Shape Morphing Set Preview
59. Push the **Up** button

Optimize the Structure Using Genesis

60. From the main menu bar, select **Genesis** → **Optimize**
61. Study the **Design History**, when done push the **Close** button
62. Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

63. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
64. Select the `FFDSG003_dsg.SHP` file.
65. Push the **Open** button

Postprocessing the Results

66. Select the **Post** tab
67. Push the **Deform Mesh/Color Mesh** button
68. Select a shape change result for any design cycle
69. For the **Color Mesh**, select the **Filled Contours** radio button
70. Select a shape change result for any design cycle for the color plot in the Viewport
71. Study the shape in the final design cycle
72. Push the **Up** button

Quit Design Studio

73. From the main menu bar, select **File** → **Quit**
74. Push the **Don’t Save** button

Study the Output File

75. In a text editor load the Genesis data file: `FFDSG003_dsg.out`
76. Study briefly the file

77. Using the output file, complete the following table:

	Reference Answer	**Obtained Answer**
Number of Design Variables	15	
Initial Objective	146.2	
Optimal Objective	82.0	

Note that due to axi-symmetry, only 15 design variables were needed for solving the problem.

8.4 On Domain Perturbations (DVGRIDC) - Mirror Symmetry

Introduction

The purpose of this example is to learn how to create and solve a freeform optimization problem using domain perturbations. This example goes through the process of defining mirror symmetries along with grid fraction constraints. This example will review how to post-process the shape changes.

The following objective will be used:

Minimize Strain Energy

Subject to:

Grid Fraction $\leq$ 0.25

Example ID

`FFDSG004`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`FFDSG004.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with static loadcases.
`FFDSG004_dsg.dat`	`Input data`	Created: Genesis input file including the freeform optimization data
`FFDSG004_ref.dat`	`Input data`	Provided: File ready to be optimized. Same as `FFDSG004_dsg.dat`
`FFDSG004_dsg.SHP`	`Shape Change file`	Created: Output file containing the shape changes at each design cycle

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `FFDSG004.dat`

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Study the two existing bending loadcases

Create the Objective

5. From the **Design** category chooser, select **Objectives**
6. Push the **New Objective** button from the Edit Menu toolbar
7. Enter `StrainEnergy` for the **Name**
8. Select **Strain Energy** for the **Response type**
9. Push **Next>**
10. Select the two existing loadcases
11. Push the **Finish** button

Create Freeform Domain Morphing Set with Axisymmetry

12. From the **Design** category chooser, select **Shape Morphing Sets**
13. Push the **New Shape Set** button from the Edit Menu toolbar
14. Enter `Perturbations` for the **Name**
15. Make sure the **Domain Morphing Set** radio button is selected
16. Make sure `New Variable [-1.0, 1.0]` is selected for the **Design Variable**
17. Select the **Yes** radio button for **Freeform**
18. Enter `1.0` for the **Maximum Perturbation**
19. Enter `1.0` for the **Initial Randomness**
20. Push **Next>**
21. Select the existing domains by selecting **Edit** → **Select All** from the main menu
22. Push **Next>**

    Only the Grids that are corners of the domains need to be selected. To do so, the PSHELL group needs to be hidden.

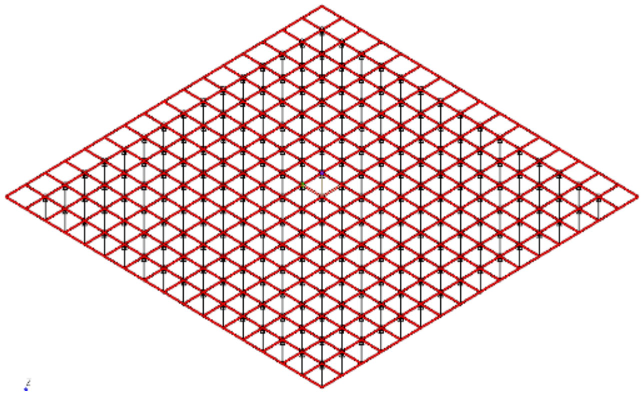
23. Select the **Display** tab
24. Select the **Show/Hide Groups** button
25. Select PSHELL 16 from the list to hide the group
26. Select the **Design** tab
27. Push the **Select None** button

28. Select all the interior grids of the domains.

    Notice that 225 grids are selected

29. Enter `0.0`, `0.0`, `1.0` for **X**, **Y**, **Z** components of the direction
30. Enter `1.0` for the **Magnitude**
31. Push the **Add Perturbation** button

    Notice “225 perturbations on 225 grids” in the summary. The perturbations are shown in the figure below.



32. Push **Next>**

    Symmetry conditions are now applied based on a coordinate system

33. Push the **Change** button for **Symmetry Coord. Sys**
34. Select `Coord. Sys. 2` for the Coordinate System
35. Push **Next>**
36. For **Symmetry 1**, select **MYZ: Mirror about YZ plane** option
37. For **Symmetry 2**, select **MZX: Mirror about XZ plane** option
38. Enter `0.25` for the **Grid Fraction** to define the grid fraction constraint
39. Push the **Finish** button

Preview the Shape Changes

40. Select the **Post** tab
41. Push the **Deform Mesh/Color Mesh** button
42. Under the Deform Mesh, select a Shape Morphing Set Preview
43. Select the **Oscillate** radio button
44. Under the Color Mesh, select the **Filled Contours** radio button
45. Under the Color Mesh, select a Shape Morphing Set Preview

46. Push the **Up** button

Optimize the Structure Using Genesis

47. From the main menu bar, select **Genesis** → **Optimize**
48. Study the **Design History**, when done push the **Close** button
49. Study the **Genesis Console Output**, when done push the **Close** button

Import the Shape Changes Post Processing File

50. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
51. Select the `FFDSG004_dsg.SHP` file.
52. Push the **Open** button

Postprocessing the Results

53. Select the **Post** tab
54. Push the **Deform Mesh/Color Mesh** button
55. Select a shape change result for any design cycle
56. For the **Color Mesh**, select the **Filled Contours** radio button
57. Select a shape change result for any design cycle for the color plot in the Viewport
58. Study the shape in the final design cycle
59. Push the **Up** button

Quit Design Studio

60. From the main menu bar, select **File** → **Quit**
61. Push the **Don’t Save** button

8.5 Design of a Connecting Rod - Using Domain Perturbations

Introduction

The purpose of this example is to learn how to use shape optimization to design the shape of a connecting rod. This example demonstrates the ease of creating domains and applying perturbations on the domains. One set of perturbations are setup to perform freeform optimization. A genesis generated perturbation file (DVG) is used to show the difference between the defined perturbation data and the perturbation data created for the freeform optimization problem.

The following objective will be used:

Minimize Mass

Subject to:

vonMises Stress $\leq$ 15000.0

Maximum Displacement $\leq$ 0.2

Example ID

FFDSG005

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
FFDSG005.dat	Input data	Provided: This file is imported into Design Studio and contains the finite element mesh along with static loadcases.
FFDSG005_dsg.dat	Input data	Created: Genesis input file including the freeform optimization data
FFDSG005_ref.dat	Input data	Provided: File ready to be optimized. Same as FFDSG005_dsg.dat
FFDSG005_dsg.DVG	Perturbation file	Created: Output file containing the perturbations created by genesis
FFDSG005_dsg.SHP	Shape Change file	Created: Output file containing the shape changes at each design cycle
FFDSG005_dsgxx.pch	Punch output file	Created: Output file containing the analysis results for design cycle xx

Start Design Studio

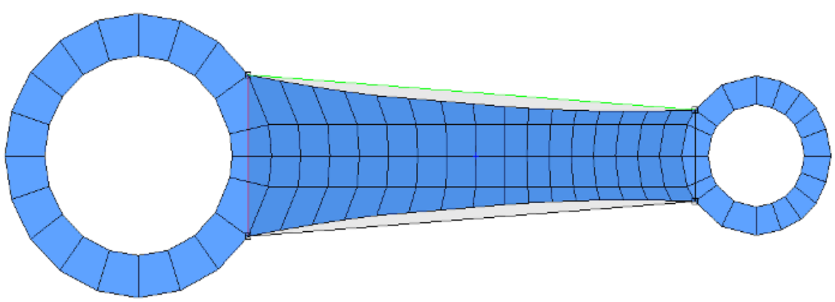
1. Start Design Studio
2. Import the Genesis data file: `FFDSG005.dat`

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Study the existing loadcases

Create Multiple 2-D (QUAD) Domains to Modify Y-Dimension

5. From the **Design** category chooser, select **Shape Domains**
6. Push the **New Domain** button from the Edit menu toolbar
7. Enter `Y-Dimension` for the **Name**
8. Make sure the **New Domains Quick Setup** radio button is selected
9. Push **Next>**
10. Select the `Create New Domain Group` option
11. Push **Next>**
12. Select the **Point by point** radio button
13. Select the **Quads** icon
14. Select the **Pick existing grids** radio button
15. From the Viewport window, change the view to be the XY plane (Top view)
16. Select the four existing grids from the Viewport as shown in the figure below



17. Select the `Quad` created
18. Push the **Subdivide** button

19. Enter `2` for the Elements in 1st dimension

    The 1st dimension is represented by the red color line for the edge of the quad domain. Enter 2 if the red colored line is along the y-axis. If the red color line is in the x-direction, then enter 10 for the Elements in 1st dimension.

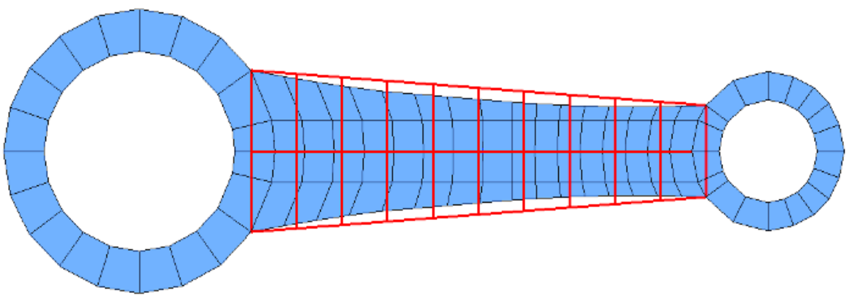
20. Enter `10` for the Elements in 2nd dimension

    The 2nd dimension is represented by the green color line for the edge of the quad domain. Enter 10 if the green colored line is along the x-axis. If the green color line is in the y-direction, then enter 2 for the Elements in 1st dimension.

21. Push **Next>**

22. Push the **Finish** button.

    In this step, 20 QUAD domains were created by creating a big domain and dividing it. The created domains should look like the picture below.



Review the Shape Domains

23. From the **Design** category chooser, select **Shape Domains**

24. Select one of the created QUAD domain

25. Push the **Modify Domain** button from the Edit Menu toolbar

    Notice that the grids controlled by the QUAD domain are selected.

26. Push the **Cancel** button

27. Right click on the Viewport window and select **Clear** → **All** to clear the selection of the grids

Create Freeform Domain Morphing Set for Y-Dimension

28. From the **Design** category chooser, select **Shape Morphing Sets**

29. Push the **New Shape Set** button from the Edit Menu toolbar

30. Enter `Y-Dimension` for the **Name**

31. Make sure the **Domain Morphing Set** radio button is selected

32. Make sure `New Variable [-1.0, 1.0]` is selected for the **Design Variable**
33. Select the **Yes** radio button for **Freeform**
34. Enter `10.0` for the **Maximum Perturbation**
35. Push **Next>**
36. Select the existing domains by selecting **Edit** → **Select All** from the main menu
37. Push **Next>**
38. Push the **Select None** button
39. Select the 9 grids of the top domains as shown in the figure below.

    Notice that 9 grids are selected

40. Enter `0.0`, `-1.0`, `0.0` for **X**, **Y**, **Z** components of the direction
41. Enter `1.0` for the **Magnitude**
42. Push the **Add Perturbation** button

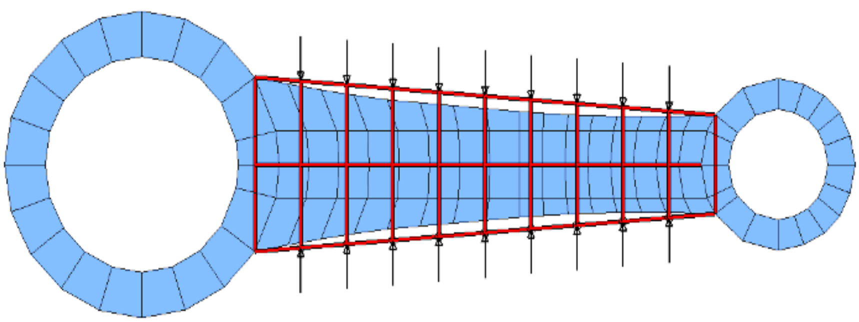
    Notice "9 perturbations on 9 grids" in the summary

43. Push the **Select None** button
44. Select the 9 grids of the bottom domains.

    Notice that 9 grids are selected

45. Enter `0.0`, `1.0`, `0.0` for **X**, **Y**, **Z** components of the direction
46. Enter `1.0` for the **Magnitude**
47. Push the **Add Perturbation** button

    Notice "18 perturbations on 18 grids" in the summary. The perturbations are shown in the figure below.



48. Push the **Finish** button

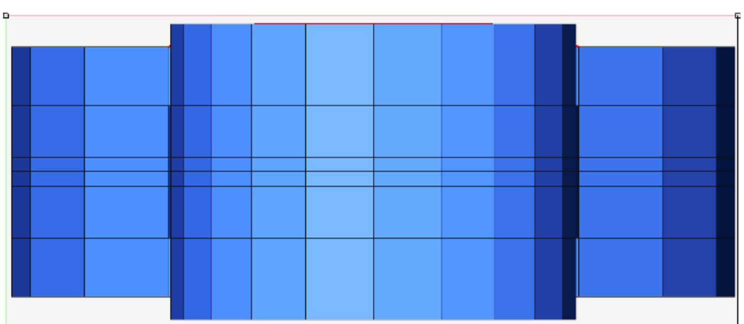
Preview the Shape Morphing Set

49. Select the **Post** tab

50. Push the **Deform Mesh/Color Mesh** button
51. Under the Deform Mesh, select the existing Shape Morphing Set Preview
52. Select the **Oscillate** radio button
53. Under the Color Mesh, select the **Filled Contours** radio button
54. Under the Color Mesh, select the existing Shape Morphing Set Preview
55. Push the **Up** button

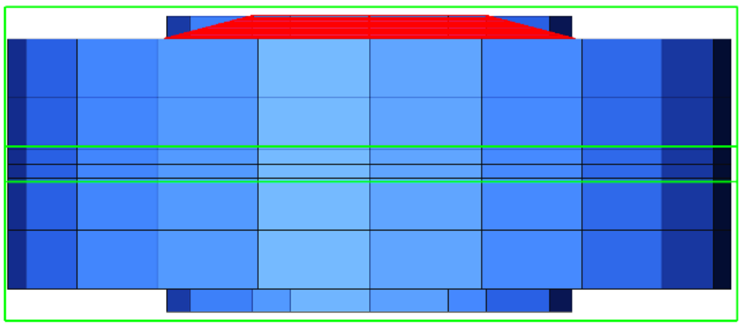
Create Multiple 2-D (QUAD) Domains to Modify Z-Dimension

56. From the **Design** category chooser, select **Shape Domains**
57. Push the **New Domain** button from the Edit menu toolbar
58. Enter `Z-Dimension` for the **Name**
59. Make sure the **New Domains Quick Setup** radio button is selected
60. Push **Next**>
61. Select the `Create New Domain Group` option
62. Push **Next>**
63. Select the **Drag out size** radio button
64. Select the **Quads** icon
65. Select the **Pick points on workplane** radio button
66. Select the **YZ plane of screen (SCR)** icon
67. Push the **Relocate origin** button
68. Enter `-32.0`, `0.0`, `0.0` for **X**, **Y**, **Z** and press the Enter button on the keyboard
69. From the Viewport window, change the view to be the YZ plane (Back view)
70. Drag a rectangular domain on the Viewport that encloses the entire structure as shown in the figure below



71. Select the `Quad` created
72. Push the **Subdivide** button
73. Enter `1` for the Elements in 1st dimension
74. Enter `3` for the Elements in 2nd dimension
75. Enter `0.25` for the Bias
76. Push **Next>**
77. Push the **Finish** button.

    The created domains should look like the picture below.



Review the Shape Domains

78. From the **Design** category chooser, select **Shape Domains**
79. Select one of the created QUAD domain
80. Push the **Modify Domain** button from the Edit Menu toolbar

    Notice that the grids controlled by the QUAD domain are selected.

81. Push the **Cancel** button
82. Right click on the Viewport window and select **Clear** → **All** to clear the selection of the grids

Create Domain Morphing Set for Z-Dimension

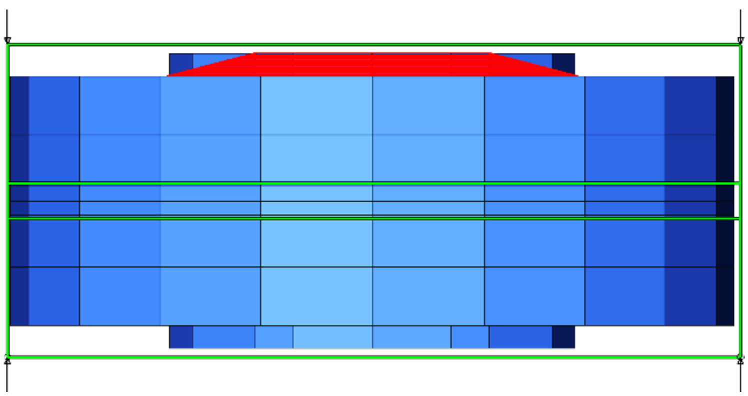
83. From the **Design** category chooser, select **Shape Morphing Sets**
84. Push the **New Shape Set** button from the Edit Menu toolbar
85. Enter `Z-Dimension` for the **Name**
86. Make sure the **Domain Morphing Set** radio button is selected
87. Make sure `New Variable [-1.0, 1.0]` is selected for the **Design Variable**
88. Make sure **No** radio button is selected for **Freeform**

89. Push **Next>**
90. Select the three `Z-Dimension` domains
91. Push **Next>**
92. Push the **Select None** button
93. Select the 2 grids on the top as shown in the figure below
94. Enter `0.0`, `0.0`, `-1.0` for **X**, **Y**, **Z** components of the direction
95. Enter `10.0` for the **Magnitude**
96. Push the **Add Perturbation** button

    Notice "2 perturbations on 2 grids" in the summary
97. Push the **Select None** button
98. Select the 2 grids on the bottom as shown in the figure below
99. Enter `0.0`, `0.0`, `1.0` for **X**, **Y**, **Z** components of the direction
100. Enter `10.0` for the **Magnitude**
101. Push the **Add Perturbation** button

    Notice "4 perturbations on 4 grids" in the summary.
102. Push the **Finish** button

    The perturbations are shown in the figure below.



Preview the Second Shape Morphing Set

103. Select the **Post** tab
104. Push the **Deform Mesh/Color Mesh** button
105. Under the Deform Mesh, select the second Shape Morphing Set Preview
106. Select the **Oscillate** radio button

107. Under the Color Mesh, select the **Filled Contours** radio button
108. Under the Color Mesh, select the second Shape Morphing Set Preview
109. Push the **Up** button

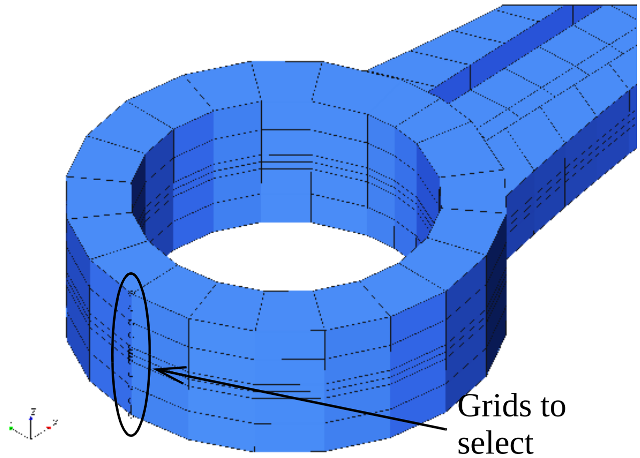
Create the Objective

110. From the **Design** category chooser, select **Objectives**
111. Push the **New Objective** button from the Edit Menu toolbar
112. Enter `Mass` for the **Name**
113. Make sure **Mass** is selected for the **Response type**
114. Push the **Finish** button

Create the Design Constraints

115. From the **Design** category chooser, select **Constraints**
116. Push the **New Constraint** button from the Edit Menu toolbar
117. Enter `Stress` for the **Name**
118. Make sure **Stress** radio button is selected for the **Response type**
119. Enter `15000.0` for the **Upper Bound**
120. Push **Next>**
121. Select PSOLID 1 from the list
122. Push **Next>**
123. Make sure von Mises is selected for the option for PSOLID Stress
124. Push **Next>**
125. Select the existing loadcase
126. Push the **Finish** button
127. Push the **New Constraint** button from the Edit Menu toolbar
128. Enter `Displacement` for the **Name**
129. Select the **Displacement** radio button for the **Response type**
130. Enter `0.2` for the **Upper Bound**
131. Push **Next>**

132. Select the 7 grids along the left end of the structure as shown in the figure below



133. Select the **Translational Magnitude** radio button for the component

134. Push **Next>**

135. Select the existing loadcase

136. Push the **Finish** button

Request for Genesis Perturbation File

137. From the main menu bar, select **Genesis** → **Options...**

138. Select the **File Control** tab

139. For **Grid Perturbation File**, select the **Create** option

140. Push the **Apply** button

Run Genesis in CHECK mode to create Perturbation (DVG) File

141. From the main menu bar, select **Genesis** → **Check Data**

142. Study the **Genesis Console Output** window

Import the Perturbation File

143. From the **Genesis Console Output** window, select the **Import Post...** button

144. Select the `FFDSG005_dsg.DVG` file.

145. Push the **Import** button

146. When done, push the **Close** button on the **Genesis Console Output** window

Compare the Splitted Perturbations vs. Original Perturbations

147. Select the **Post** tab
148. Push the **Deform Mesh/Color Mesh** button
149. Under the Deform Mesh, select a Shape Morphing Set Preview
150. Select the **Oscillate** radio button
151. Under the Color Mesh, select the **Filled Contours** radio button
152. Under the Color Mesh, select the `DVAR 1 Shape Morphing Set Preview`
153. Under the Color Mesh, select the `DVAR 1 Perturbation`

    Notice that in the `Shape Morphing Set Preview`, DVAR 1 controls all the perturbations along the length of the connecting rod as defined. But in the `Perturbation`, DVAR 1 controls the perturbation of only one corner of a domain. As this perturbation is defined as freeform, each of the perturbations applied for the Y-Dimension is being controlled by a different design variable. That is the reason for the additional perturbations in the DVG file. Also notice the difference in the magnitude of the perturbation. The Shape Morphing Set Preview shows the magnitude of the perturbation applied while the DVG perturbation shows a magnitude that is scaled by the **Maximum Perturbation** value defined while creating the Morphing Set.

154. Under the Color Mesh, select the `DVAR 2 Shape Morphing Set Preview`
155. Under the Color Mesh, select the `DVAR 2 Perturbation`

    As the shape morphing set associated with DVAR 2 is not defined as freeform, the perturbations look the same in the `Shape Morphing Set Preview` as well as the `Perturbation` from the DVG file

156. Under the Color Mesh, study the other created Perturbations

    Notice that 17 additional design variables were created and associated with the perturbations on the Y-Dimension shape morphing set

157. Push the **Up** button

Optimize the Structure Using Genesis

158. From the main menu bar, select **Genesis** → **Optimize**
159. Study the **Design History**, when done push the **Close** button
160. Study the **Genesis Console Output** window

Import the Shape Changes Post Processing File

161. From the **Genesis Console Output** window, select the **Import Post...** button

162. Select the `FFDSG005_dsg.SHP` file along with the punch files created for the analysis output
163. Push the **Import** button
164. When done, push the **Close** button on the **Genesis Console Output** window

Postprocessing the Shape Change Results

165. Select the **Post** tab
166. Push the **Deform Mesh/Color Mesh** button
167. Select a shape change result for any design cycle
168. For the **Color Mesh**, select the **Filled Contours** radio button
169. Select a shape change result for any design cycle for the color plot in the Viewport
170. Study the shape in the final design cycle
171. Push the **Up** button

View the Analysis Results

172. Push the **Deform Mesh/Color Mesh** button
173. For the **Color Mesh**, select the **Filled Contours** radio button
174. Select a Displacement result for the first design cycle for the color plot in the Viewport
175. Select a Displacement result for the last design cycle

    Study the difference and notice the magnitude of the displacement in the final design cycle. Is the displacement constraint active?

176. For the **Color Mesh**, select the **Filled Elements** radio button
177. Select a Stress result for the first design cycle
178. Select a Stress result for the last design cycle

    Study the difference and notice the magnitude of the Stresses in the final design cycle. Is the Stress constraint active?

179. Push the **Up** button

Quit Design Studio

180. From the main menu bar, select **File** → **Quit**
181. Push the **Don’t Save** button

8.6 On Grid Perturbations Generated Using Domains

Introduction

The purpose of this example is to learn how to create and solve a freeform optimization problem using the grid perturbations generated by domains. This example goes through the process of defining extrusion constraints along with grid fraction constraints. This example will review how to post-process the shape changes.

The following objective will be used:

Minimize Strain Energy

Subject to:

Grid Fraction $\leq$ 0.2

Example ID

`FFDSG006`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
`FFDSG006.dat`	`Input data`	Provided: This file is imported into Design Studio and contains the finite element mesh along with static loadcases.
`FFDSG006_dsg.dat`	`Input data`	Created: Genesis input file including the freeform optimization data
`FFDSG006_ref.dat`	`Input data`	Provided: File ready to be optimized. Same as `FFDSG006_dsg.dat`
`FFDSG006_dsg.SHP`	`Shape Change file`	Created: Output file containing the shape changes at each design cycle

Start Design Studio

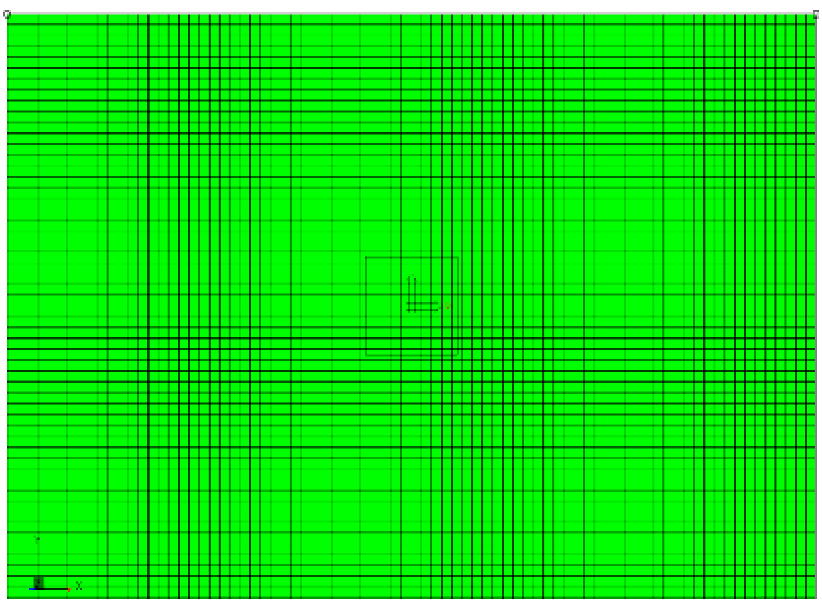
1. Start Design Studio
2. Import the Genesis data file: `FFDSG006.dat`

Review the Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Study the existing loadcases

Create 1-D (BAR) Domain

5. From the **Design** category chooser, select **Shape Domains**
6. Push the **New Domain** button from the Edit menu toolbar
7. Enter `Bar` for the **Name**
8. Make sure the **New Domains Quick Setup** radio button is selected
9. Push **Next**>
10. Select the `Create New Domain Group` option
11. Push **Next>**
12. Select the **Point by point** radio button
13. Select the **Lines** icon
14. Select the **Pick existing grids** radio button
15. From the Viewport window, change the view to be the XY plane (Top view)
16. Select the two corner grids of the top of the structure from the Viewport as shown in the figure below



17. Push the **Finish** button.

Create Freeform Domain Morphing Set

18. From the **Design** category chooser, select **Shape Morphing Sets**
19. Push the **New Shape Set** button from the Edit Menu toolbar
20. Enter `Ribs-X` for the **Name**
21. Make sure the **Domain Morphing Set** radio button is selected
22. Make sure `New Variable [-1.0, 1.0]` is selected for the **Design Variable**
23. Select the **Yes** radio button for **Freeform**
24. For the **Freeform Level**, select the **Split All Generated Perts.** option

    Using this option, each grid perturbation generated by the domain is controlled by a different design variable

25. Enter `5.0` for the **Maximum Perturbation**
26. Enter `1.0` for the **Initial Randomness**
27. Push **Next>**
28. Select the existing BAR domain
29. Push **Next>**
30. Push the **Select None** button
31. Select the grid close to the center of the BAR domain (Grid ID `477`)
32. Enter `0.0`, `0.0`, `1.0` for **X**, **Y**, **Z** components of the direction
33. Enter `1.0` for the **Magnitude**
34. Push the **Add Perturbation** button

    Notice “1 perturbations on 1 grids” in the summary

35. Push **Next>**
36. For the **Coarsening Method**, select **Diameter** option
37. Enter `2.0` for the **Real Coarse Parameter**
38. Push the **Change** button for **Symmetry Coord. Sys**
39. Select `Coord_1` for the Coordinate System

    Symmetry conditions will be applied based on the above selected coordinate system

40. Push **Next>**
41. For **Symmetry 1**, select **EX: Extrude along X axis** option
42. For **Symmetry 2**, select **MZX: Mirror about XZ plane** option
43. Enter `0.2` for the **Grid Fraction** to define the grid fraction constraint

44. Push the **Finish** button

Modify the Design Variable

45. From the **Design** category chooser, select **Design Variables**
46. Select the existing design variable (`Shape1`) which is created while defining the morphing set
47. Push the **Modify Design Variable** button from the Edit Menu toolbar
48. Push **Next>**
49. Modify the value of the **Lower Bound** to be `0.0`
50. Push the **Finish** button

Preview the Shape Changes

51. Select the **Post** tab
52. Push the **Deform Mesh/Color Mesh** button
53. Under the Deform Mesh, select a Shape Morphing Set Preview
54. Select the **Oscillate** radio button
55. Under the Color Mesh, select the **Filled Contours** radio button
56. Under the Color Mesh, select a Shape Morphing Set Preview

    Notice the variation of the structure along the x-axis. Since the perturbation is on a grid in the middle of the BAR domain the variation is quadratic.

57. Push the **Up** button

Create the Objective

58. From the **Design** category chooser, select **Objectives**
59. Push the **New Objective** button from the Edit Menu toolbar
60. Enter `StrainEnergy` for the **Name**
61. Select **Strain Energy** for the **Response type**
62. Push **Next>**
63. Select the existing loadcase
64. Push the **Finish** button

Optimize the Structure Using Genesis

65. From the main menu bar, select **Genesis** → **Optimize**
66. Study the **Design History**, when done push the **Close** button
67. Study the **Genesis Console Output** window

Import the Shape Changes Post Processing File

68. From the **Genesis Console Output** window, select the **Import Post...** button
69. Select the `FFDSG006_dsg.SHP` file
70. Push the **Import** button
71. When done, push the **Close** button on the **Genesis Console Output** window

Postprocessing the Shape Change Results

72. Select the **Post** tab
73. Push the **Deform Mesh/Color Mesh** button
74. Select a shape change result for any design cycle
75. For the **Color Mesh**, select the **Filled Contours** radio button
76. Select a shape change result for any design cycle for the color plot in the Viewport
77. Study the shape in the final design cycle

    Notice the variation of the height of each rib. The variation is quadratic along the BAR domain

78. Push the **Up** button

Quit Design Studio

79. From the main menu bar, select **File** → **Quit**
80. Push the **Don't Save** button

CHAPTER 9

Composite Optimization Examples

9.1 Layer Thickness Optimization of a Cantilever Plate

Introduction

The purpose of this example is to introduce the basic steps to create and solve a simple composite optimization problem. This example will show how to create design variables for layer thicknesses and how to link them to the composite properties. This example will also review the basic convention of the PCOMP data on layer numbering. Finally, this example will show how to visualize layer thickness, layer angle and failure indices.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

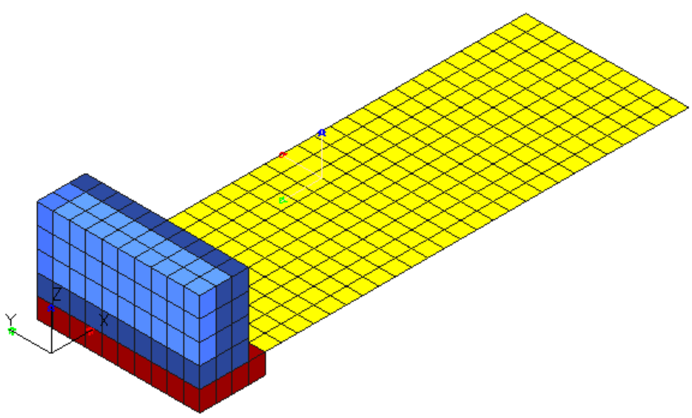
Subject to:

Failure Index$\leq$ 0.5 (all elements)

Designable region:

Six thicknesses of the composite plate

The following picture shows the structure:



Example ID

`CMDSG001`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
CMDSG001.dat	Input data	Provided: Contains the finite element mesh along with applied load and boundary conditions.
CMDSG001_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in CMDSG001.dat
CMDSG001_dsg.out	Output file	Generated by a Genesis run within Design Studio. This is a Genesis output file.
CMDSG001_dsgxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the analysis results for design cycle “xx”.
CMDSG001_dsgOPOSTxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the element thickness results for design cycle “xx”.
CMDSG001_dsgSSOLxx.dat	Input data	Generated by a Genesis run within Design Studio. This is the input data of a solid model obtained using the thicknesses of the shells for design cycle “xx”. The file is for visualization purposes and not intended for analysis.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG001.dat`

Study the Analysis Problem

3. From the main menu bar, select **Genesis** → **Model Summary**

   Verify that the model has the following characteristics:
   Number of grids: 572
   Number of CQUAD4 elements: 270

4. Push the **Close** button
5. Select the **Analysis** tab
6. From the category chooser, select **Loadcases**
7. Select the existing loadcase

Clearing the Selection

8. From the main menu, select **Edit**→ **Deselect All**

Check the Element Norms

9. Select the **Display** tab

10. Push the **Show/Hide Groups** button
11. Push the **Hide All** button
12. Select PCOMP 10, to display the composite elements
13. Select the **Analysis** tab
14. From the category chooser, select **Elements**
15. Push the **Select All** button
16. Push the **Generate Orientation Vectors** button

    Verify that the orientation vectors of the elements in the plate point upward.

Clearing the Selection

17. Right-click the Viewport, select **Clear**→ **All**

Study the Group Properties of the PCOMP

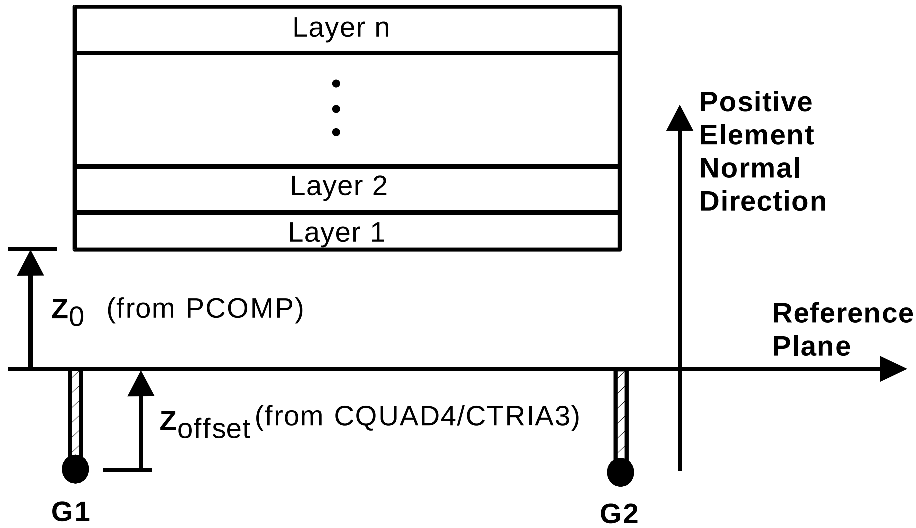
18. From the **Analysis** category chooser, select **Group Properties**
19. Select PCOMP 10
20. Push the **Modify Group Property** button from the Edit menu toolbar

    Verify that Z0=0.0 and that:

Layer	Material	Thickness	Angle
1	103	0.3	45.0
2	103	0.3	0.0
3	102	0.2	0.0
4	102	0.2	30.0
5	102	0.2	60.0
6	102	0.2	90.0
7	103	0.3	45.0

21. Push the **Cancel** button

The following picture shows the definition of PCOMP.



In the above figure you can see the convention for numbering the composite layers. Layer 1 is at the bottom of the composite. Layer n is at the top of the composite. The orientation vector (norm) of the elements defines what is top and what is bottom.
In this particular example, Z0=0.0 and Z0ffset=0.0. So layer 1 grows from the element grids upward.

Create Six Design Variables

22. Select the **Design** tab
23. From the category chooser, select **Design Variables**
24. Push the **New Design Variable** button from the Edit Menu toolbar
25. For **Name,** enter T2
26. Accept default type of design variables, **Independent Design Variables**
27. Push the **Next>** button
28. Using the following table, enter the appropriate values for **Initial value**, **Lower bound** and **Upper bound**:

Name	Initial Value	Lower Bound	Upper Bound
T2	0.3	0.005	3.0

29. Push the **Finish** button

30. Repeat the steps from 24 to 29 to create five additional design variables as follows:

Name	Initial Value	Lower Bound	Upper Bound
T3	0.2	0.005	3.0
T4	0.2	0.005	3.0
T5	0.2	0.005	3.0
T6	0.2	0.005	3.0
T7	0.3	0.005	3.0

Create the Sizing Region

31. From the **Design** category chooser, select **Sizing**
32. Select the PCOMP 10
33. Push the **Modify Sizing Design** button from the Edit Menu toolbar
34. For the **Thickness** of **Layer 2,** select design variable `T2`
35. For the **Thickness** of **Layer 3,** select design variable `T3`
36. For the **Thickness** of **Layer 4,** select design variable `T4`
37. For the **Thickness** of **Layer 5,** select design variable `T5`
38. For the **Thickness** of **Layer 6,** select design variable `T6`
39. For the **Thickness** of **Layer 7,** select design variable `T7`
40. Push the **Finish** button

    Verify that the hammer icon is next to PCOMP 10.
    The hammer icon indicates that the PCOMP group is being size-designed.

Defining the Design Objective

41. From the category chooser, select **Objectives**
42. Push the **New Objective** button from the Edit Menu toolbar
43. For **Name,** enter `Mass`
44. Accept the default, **Mass** response type
45. Accept the **Min** Objective Definition switch
46. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Defining the Design Constraints

47. From the category chooser, select **Constraints**
48. Push the **New Constraint** button from the Edit Menu toolbar
49. For **Name,** enter `Findex`
50. Select the **Failure Index** response
51. Select **Selected Groups** option of Failure Index
52. Enter `0.5` for the **Upper Bound**
53. Push the **Next>** button
54. Select PCOMP 10
55. Push the **Next>** button
56. Select the static loadcase
57. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Clearing the Selection

58. Right-click the Viewport, select **Clear→ All**

Request Element Stresses and Element Forces

59. Select the **Analysis** tab
60. From the category chooser, select **Loadcases**
61. Select the existing loadcase
62. Push the **Modify Loadcase** button from the Edit menu toolbar
63. Push the **Next>** button
64. Push the **Next>** button
65. Push the **Next>** button
66. For **Element Stress**, make sure that **Post** is selected

    The layer thicknesses as well as the layer angles are printed along with the element stresses.
67. For **Element Force**, make sure that **Post** is selected

    The layer failure indices are printed along with the element forces.
68. Push the **Finish** button

Request the SSOL File

The SSOL file is a Genesis input file where the composite elements will be converted into solid elements for the purpose of better visualizing the thicknesses distribution. This file is not intended to be used for analysis or optimization.

69. From the main menu bar, select **Genesis** → **Options...**
70. Select the **File Control** tab
71. For **Shell-to-Solid File**, pick **Create(Fixed Norms)**

    The name of the SSOL file will be:
    `CMDSG001_dsgSSOLxx.dat` (where xx corresponds to the last design cycle number).
72. Push the **Apply** button

Request the OPOST (Sizing data) Post-Processing File

The OPOST file is a Genesis post-processing file that contains the optimization results, in this case the thickness of the composite elements. This file only contains the total thickness of the composite, it does not contain the individual layer thicknesses.

73. From the main menu bar, select **Genesis** → **Options**
74. Select the **File Control** tab
75. For **Element Sizing File**, choose **Create**
76. Push the **Apply** button

Optimize the Structure Using Genesis

77. From the main menu bar, select **Genesis** → **Optimize**
78. Study the **Design History** charts; when done, push the **Close** button
79. Study the **Genesis Console Output** window

Import the Analysis Post-Processing Files

80. From the **Genesis Console Output** window, select **Import Post...** button
81. Using the **Shift** key, select all the files post-processing files
82. Push the **Import** button
83. From the **Genesis Console Output** window, select **Close** button

Post-Processing some Analysis Results

84. Select the **Post** tab
85. Push the **Deform/Mesh Color Mesh** button
86. Select a Composite Failure Index Result for any design cycle
87. Select a Composite Failure Index Result for the last design cycle
88. Will the design fail?
89. Select the Composite Stress Result for the last design cycle
90. From the category chooser in the **Color Mesh** option, change **Von Mises** to **Layer Thickness**

    Now you visualize the layer thickness 1.

91. Select one element in the viewport

    The value of the thickness of the element is printed in the Message window

92. Select one element in the viewport
93. In the box near the category chooser, change 1 to 2

    Now you visualize the layer thickness 2.

94. Select one element in the viewport
95. What is the thickness of Layer 2?
96. From the category chooser (drop-down listbox by the side of the **Options** button) in the **Color Mesh** option, change **Layer Thickness** to **Layer Angle**
97. Select one element in the viewport
98. What is the angle of Layer 2?
99. Change **Layer Angle** to **Total Thickness**
100. What is the total thickness of the composite in the last design cycle?
101. What is the total thickness of the composite in the first design cycle?

Post-Processing the Results (Total Thickness on Composites)

102. Select a Thickness Result for the first design cycle
103. Select a Thickness Result for the last design cycle
104. Push the **Up** button

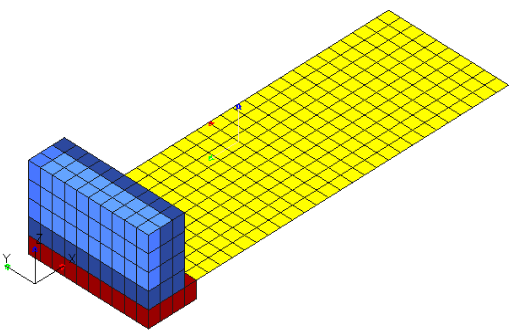
Import the Solid File

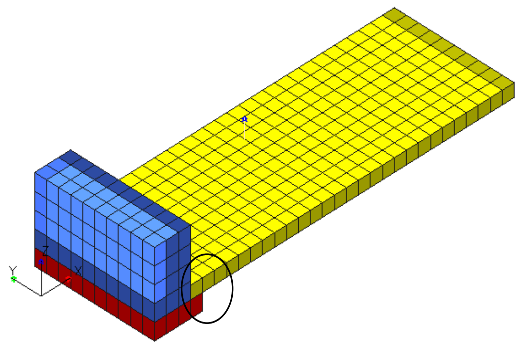
105. From the main menu bar, select **File** → **Import**→ **Input data...**
106. Import the Genesis data file: `CMDSG001_dsgSSOLxx.dat` (`XX` corresponds to the last design cycle)

Study the Composite Thickness

107. Select the Display tab
108. Push the **Show All** button
109. Hide and show PSOLID 54 few times to view the thickness

Verify that the composite grew from the bottom upward.





Quit Design Studio

110. From the main menu bar, select **File** → **Quit**
111. Push the **Don't Save** button

Study the Output File

112. In a text editor load the Genesis data file: `CMDSG001_dsg.out`
113. Study briefly the file

114. Using the output file, complete the following table:

Design Variable Name	Original Design Variable Value Reference Solution	Final Design Variable Value Reference Solution	Final Design Variable Value
T2	0.3	0.503	
T3	0.2	0.005	
T4	0.2	0.010	
T5	0.2	0.005	
T6	0.2	0.152	
T7	0.3	0.005	

115. In a text editor load the Genesis data file: `CMDSG001_dsgUPDATExx.dat` (xx corresponds to the last design cycle), and study briefly the file

Using the PCOMP 10 data entry, complete the following table:

PCOMP Layer Number	Optimal Layer Thickness Reference Solution	Optimal Layer Thickness
1 (not designed)	0.3	
2	0.503	
3	0.005	
4	0.010	
5	0.005	
6	0.152	
7	0.005	

Note the Format of PCOMP is:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z_0	NSM		F.T.	TREF	GE	LAM	MEM
+	MID1	T1	θ_1	SOUT1	MID2	T2	θ_2	SOUT2	
+	MID3	T3	θ_3	SOUT3	-etc.-				

9.2 Composite Optimization of a Cantilever Plate Considering the Location of the Reference Plane

Introduction

The purpose of this example is to learn how to design the location of the reference plane of composite elements. Most of the design data for this problem is provided. The key data you will be creating is the one needed for designing the reference plane. This example also shows how to visualize a solid representation of the composite.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

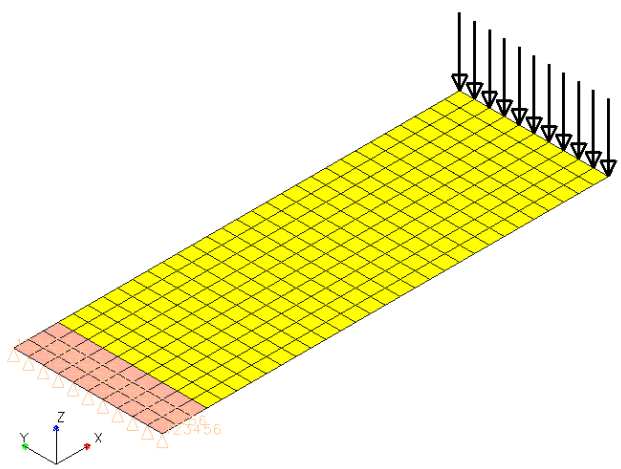
Failure Index$\leq$ 0.5 (all elements)

Designable region:

Six thicknesses of the composite plate

Location of the reference plane

The following picture shows the structure. There are two regions in the model, one is designable and the other is not. The non-designable area is the one near the fixed end:



Example ID

CMDSG002

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`CMDSG002.dat`	`Input data`	Provided: Contains the finite element mesh along with applied load and boundary conditions.
`CMDSG002_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains the same data as in `CMDSG002.dat`
`CMDSG002_dsgSSOL00.dat`	`Input data`	Generated by a Genesis run within Design Studio. This is the input data of a solid model obtained using the thicknesses of the shells for design cycle “`00`”. The file is for visualization purposes and not intended for analysis.
`CMDSG002_dsgSSOLxx.dat`	`Input data`	Generated by a Genesis run within Design Studio. This is the input data of a solid model obtained using the thicknesses of the shells for design cycle “`xx`”. The file is for visualization purposes and not intended for analysis.
`CMDSG002Sizing_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains the same data as in `CMDSG002.dat` including data generated in this example to design the reference plane
`CMDSG002Sizing_dsgSSOLxx.dat`	`Input data`	Generated by a Genesis run within Design Studio. This is the input data of a solid model obtained using the thicknesses of the shells for design cycle “`xx`”. The file is for visualization purposes and not intended for analysis.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG002.dat`

Check the Element Norms

3. Select the **Analysis** tab
4. From the category chooser, select **Elements**
5. From the Viewport, select all the displayed elements
6. Push the **Generate Orientation Vectors** button

   Verify that the orientation vectors of the elements in the composite plate point upward.

Clearing the Selection

7. Right-click the Viewport, select **Clear→ All**

Request the SSOL File

The SSOL file is a Genesis input file where the shell elements are converted into solid elements for the purpose of better visualizing the thicknesses distribution. This file is not intended to be used for analysis or optimization.

8. From the main menu bar, select **Genesis** → **Options...**
9. Select the **File Control** tab
10. For **Shell-to-Solid File**, select **Create(Fixed Norms)**

    The name of the SSOL file will be: `CMDSG002_dsgSSOL00.dat`.
11. Push the **Apply** button

Analyze the Structure Using Genesis

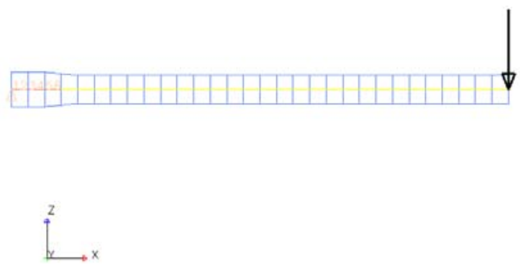
12. From the main menu bar, select **Genesis** → **Single Analysis**

Import the Solid File

13. Import the Genesis data file: `CMDSG002_dsgSSOL00.dat`

Change View

14. Push the **Left view** button to change view to the X-Z Plane



15. Study the Left View

    Verify that the top and bottom of the composite elements are located opposite each other and that Z1=-Z2 for each element (Z1 is the distance from reference plane to the bottom of the plane, Z2 is the distance from the reference plane to the top).

Re-Import the Input Data

16. From the main menu bar, select **File** → **New**
17. Push the **Don't Save** button
18. Import the Genesis data file: `CMDSG002.dat`, again

Optimize the Structure Using Genesis

19. From the main menu bar, select **Genesis** → **Optimize**
20. Study the **Design History** charts; when done, push the **Close** button
21. Study the **Genesis Console Output**; when done, push the **Close** button

Study the OPT File

22. In a text editor load the Genesis data file: `CMDSG002_dsg.OPT`
23. Study briefly the file
24. Considering that the Format of PCOMP is:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z_0	NSM		F.T.	TREF	GE	LAM	MEM
+	MID1	T1	θ_1	SOUT1	MID2	T2	θ_2	SOUT2	
+	MID3	T3	θ_3	SOUT3	-etc.-				

Fill out the following table:

Design Cycle	Z0 (PCOMP 10) Reference Solution	Z0 (PCOMP 10)
0	-0.85000	
1	-0.85000	
2	-0.85000	
3	-0.85000	

Import the Solid File

25. Import the Genesis data file: `CMDSG002_dsgSSOLxx.dat` (where `xx` corresponds to the last design cycle number)

Change View

26. Push the **Left view** button to change view to the X-Z Plane
27. Study the Left View

Verify that the top and the bottom of the designable composite elements are no longer located opposite each other. The reason for that is that Z0 (distance from Reference plane to bottom of the plate) is not being designed. Because of that, Z0= -0.85 (the initial value), for all designable elements.

Re-Import the Input Data

28. From the main menu bar, select **File** → **New**
29. Push the **Don’t Save** button
30. Import the Genesis data file: `CMDSG002.dat`, again

Create an Equation Design Variable

31. Select the **Design** tab
32. From the **Design** category chooser, select **Design Variables**
33. Push the **New Design Variable** button from the Edit Menu toolbar to create a new design variable
34. For **Name,** enter `Z0`
35. Select **Equation Design Variable**
36. Push the **Next>** button
37. Replace “F =” by: `F=-0.5*(0.3+Arg1+Arg2+Arg3+Arg4+Arg5+Arg6)`
38. Select (highlight) the Design Variable `1 T2` from the **Master Variable** list

39. Push the + button to add additional variables
40. Select the Design Variable 2 T3 from the **Master Variable** list
41. Push the + button to add additional variables
42. Select the Design Variable 3 T4 from the **Master Variable** list
43. Push the + button to add additional variables
44. Select the Design Variable 4 T5 from the **Master Variable** list
45. Push the + button to add additional variables
46. Select the Design Variable 5 T6 from the **Master Variable** list
47. Push the + button to add additional variables
48. Select the Design Variable 6 T7 from the **Master Variable** list
49. In the **Equation Parameter** list, replace `Arg1` by `T2,` then push the keyboard key Return or Enter

    Notice that Arg1 is automatically changed to T2 in the equation.
50. In the **Equation Parameter** list, replace `Arg2` by `T3,` then push the keyboard key Return or Enter
51. In the **Equation Parameter** list, replace `Arg3` by `T4,` then push the keyboard key Return or Enter
52. In the **Equation Parameter** list, replace `Arg4` by `T5,` then push the keyboard key Return or Enter
53. In the **Equation Parameter** list, replace `Arg5` by `T6,` then push the keyboard key Return or Enter
54. In the **Equation Parameter** list, replace `Arg6` by `T7,` then push the keyboard key Return or Enter
55. Verify that the equation is now: `F=-0.5*(0.3+T2+T3+T4+T5+T6+T7)`
56. Push the **Finish** button

    Notice that there is an asterisk in front of the equation design variable, which indicates that this design variable is not being used yet. The "**E**" indicates this is an **E**quation design variable.

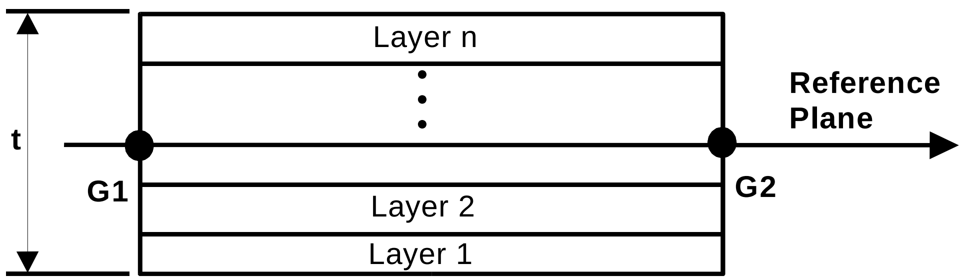
Assign the Design Variable for Designing the Location of the Reference Plane, Z0

57. Stay with the **Design** tab
58. From the **Design** category chooser, select **Sizing**

59. Select PCOMP 10
60. Push the **Modify Sizing Design** button from the Edit Menu toolbar
61. For **Z0,** select design variable `Z0`
62. Push the **Finish** button

Note

The following figure shows the desired location for the reference plane:



Typical case: Reference Plane = Mid-plane

$Z_0 = -t/2$ $\quad Z_{offset} = 0$

Request the SSOL File

63. From the main menu bar, select **Genesis** → **Options...**
64. Select the **File Control** tab
65. For **Shell-to-Solid File**, select **Create(Fixed Norms)**

    The name of the SSOL file will be:
    `CMDSG002Sizing_dsgSSOLxx.dat` (xx corresponds to the last cycle number)

66. Push the **Apply** button

Save the Design Studio Database File

67. From the main menu bar, select **File** → **Save As...**
68. Enter `CMDSG002Sizing` as the Filename and push **Save (**as a Design Studio File**)**

Optimize the Structure Using Genesis

69. From the main menu bar, select **Genesis** → **Optimize**

70. Study the **Genesis Console Output**; when done, push the **Close** button

Study the OPT File

71. In a text editor load the Genesis data file: `CMDSG002Sizing_dsg.OPT`
72. Study briefly the file
73. Considering that the Format of PCOMP is:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z_0	NSM		F.T.	TREF	GE	LAM	MEM
+	MID1	T1	θ_1	SOUT1	MID2	T2	θ_2	SOUT2	
+	MID3	T3	θ_3	SOUT3	-etc.-				

Verify that Z0, corresponding to PCOMP 10, changes as the design variables change; fill out the following table:

Design Cycle	Z0 (PCOMP 10) Reference Solution	Z0 (PCOMP 10)
0	-0.850	
1	-0.578	
2	-0.626	
3	-0.595	
4	-0.581	
5	-0.582	
Last Design Cycle	-0.582	

Import the Solid File

74. Import the Genesis data file: `CMDSG002Sizing_dsgSSOLxx.dat` (where xx corresponds to the last design cycle number)

75. Push the **Left view** button to change view to the X-Z Plane

76. Study the Left View

    Verify that the top and bottom of the composite elements are now located opposite each other and that Z1=-Z2 for each element.

Quit Design Studio

77. From the main menu bar, select **File** → **Quit**

78. Push the **Don't Save** button

9.3 Layer Angle Optimization of a Cantilever Plate

Introduction

The purpose of this example is to learn how to design layer angles of composite elements. Most of the design data for this problem is provided. The key data, that you will be creating, is the one needed for designing some selected angles. This example will also review the convention of the PCOMP data on angle measurements. Finally, this example will show how to visualize the values of layer angles. The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

Failure Index$\leq$ 0.5 (all elements)

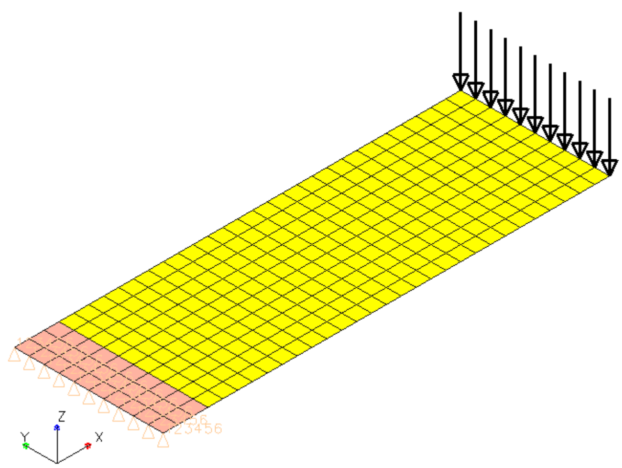
Designable region:

Six thicknesses of the composite plate (data provided)

Z0 (Location of Reference Plane (data provided)

Six Angles of the composite plate

The following picture shows the structure. There are two regions in the model, one is designable and the other is not. The non-designable area is the one near the fixed end.



Example ID

CMDSG003

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
CMDSG003.dat	Input data	Provided: Contains the finite element mesh along with applied load and boundary conditions.
CMDSG003_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in CMDSG003.dat
CMDSG003_dsg.out	Output file	Generated by a Genesis run within Design Studio. This is a Genesis output file.
CMDSG003_dsgxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the analysis results for design cycle “xx”.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG003.dat`

Check the Element Norms

3. Select the **Analysis** tab
4. From the category chooser, select **Elements**
5. From the Viewport, select all the displayed elements
6. Push the **Generate Orientation Vectors** button

   Verify that the orientation vectors of the elements in the composite plate point upward.

Clearing the Selection

7. Right-click the Viewport, select **Clear**→ **All**

Study the Local Coordinate System

8. Select the **Analysis** tab
9. From the category chooser, select **Coordinate Systems**
10. Select the local coordinate system, `Angle_Reference`

    Verify that the X-axis (represented by a red arrow) of the local coordinate system is parallel to the Y-axis of the basic coordinate system.

Clearing the Selection

11. From the main menu, select **Edit → Deselect All**

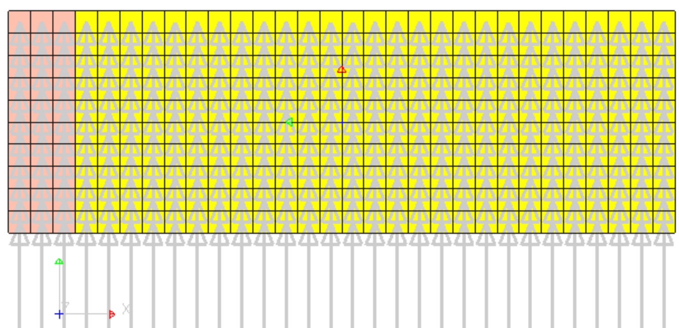
Change View

12. Push the **Top view** button to change view to the X-Y Plane

Study the Theta Vectors

13. From the category chooser, select **Elements**
14. Push the **Select all** button
15. Push the **Generate Theta Vectors** button

Verify that the theta vectors are parallel to the X-axis of the local coordinate system.



Considering that the format for the QUAD4 elements is:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA	ZOFFS	

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a PSHELL or PCOMP property data (Integer > 0 or blank, default is EID).
4-7	G1-G4	GRID identification numbers of connection point (Integers > 0, all unique).

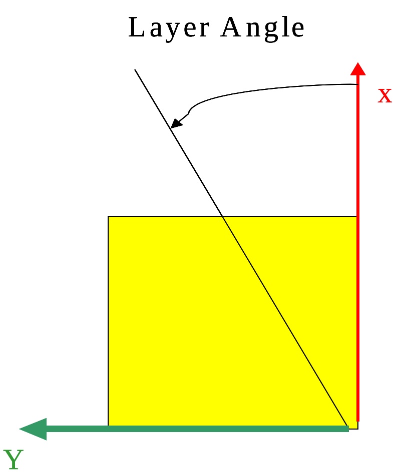
8	THETA	Material property orientation specification (Real or blank; or Integer ≥ 0). If Real, specifies the material property orientation angle in degrees. The sketch below gives the sign convention for THETA. If Integer, the orientation of the material x-axis is along the projection onto the plane of the element of the x-axis of the coordinate system specified by the integer value.
9	ZOFFS	Offset from the surface of grid points to the element reference plane (Real or blank. Default=0.0).

and that in this example, the material orientation (Theta) is provided via the local coordinate system 1, as is shown for one of the elements from the input data:

1	2	3	4	5	6	7	8	9	10
CQUAD4	902679	10	46857	47910	47936	46931	1		

you will know that the layer angles will be measured as shown in the next figure:

.



Clearing the Selection

16. Right-click the Viewport, select **Clear→ All**

Create Five Design Variables

17. Select the **Design** tab
18. From the category chooser, select **Design Variables**
19. Push the **New Design Variable** button from the Edit Menu toolbar
20. For **Name,** enter `Angle2-3`
21. Accept default type of design variables, **Independent Design Variables**

22. Push the **Next>** button
23. Using the following table, enter the appropriate values for **Initial value**, **Lower bound** and **Upper bound**

Name	Initial Value	Lower Bound	Upper Bound
Angle2-3	0.0	-90.0	90.0

24. Push the **Finish** button
25. Repeat the previous steps to create 4 additional design variables as follows:

Name	Initial Value	Lower Bound	Upper Bound
Angle4	30.0	-90.0	90.0
Angle5	60.0	-90.0	90.0
Angle6	90.0	0.0	180.0
Angle7	45.0	-90.0	90.0

Verify that left to the “**I**” of the new design variable there are asterisks “*****”. The asterisks mean that the corresponding design variables have not been referenced yet.

Add the Design Variables to Design the Angles of the Existing Sizing Region

26. Stay with the **Design** tab
27. From the **Design** category chooser, select **Sizing**
28. Select the PCOMP 10
29. Push the **Modify Sizing Design** button from the Edit Menu toolbar
30. For the **Angle** of **Layer 2,** select design variable `Angle2-3`
31. For the **Angle** of **Layer 3,** select design variable `Angle2-3`
32. For the **Angle** of **Layer 4,** select design variable `Angle4`
33. For the **Angle** of **Layer 5,** select design variable `Angle5`
34. For the **Angle** of **Layer 6,** select design variable `Angle6`
35. For the **Angle** of **Layer 7,** select design variable `Angle7`
36. Push the **Finish** button

Verify that all Design Variables are Referenced

37. Stay with the **Design** tab
38. From the **Design** category chooser, select **Design Variables**

    Verify that left to the “**I**” or “**E**”of each Design Variable there are no asterisks “*****”.

Save the Design Studio Database File

39. From the main menu bar, select **File** → **Save As...**
40. Enter `CMDSG003` as the Filename and push **Save (**as a Design Studio File**)**

Optimize the Structure Using Genesis

41. From the main menu bar, select **Genesis** → **Optimize**
42. Study the **Design History** charts; when done, push the **Close** button
43. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Analysis Post-Processing Files

44. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
45. Select the `CMDSG003_dsg00.pch` file and put a checkmark in the **Import Similar Results for All Design Cycles** checkbox

    Putting a checkmark in the checkbox will cause Design Studio to load many result files (one for each design cycle) in one step.

46. Push the **Open** button

Post-Processing some Analysis Results

47. Select the **Post** tab
48. Push the **Deform/Mesh Color Mesh** button
49. Select the Composite Stress Result for the last design cycle

    Layer results, including layer angles, are stored together with stress results.

50. From the category chooser in the **Color Mesh** option, change **Von Mises** to **Layer Angle**

    Now you visualize the layer angle1.

51. In the box near the category chooser, change 1 to 2
52. In the viewport, select one element associated with the designable region

    The value of the layer angle is printed in the Design Studio Messages window.
    Make sure the **Filled Elements** radio button is selected.

53. Repeat the above steps to find the values of all layers

Study the Output File

54. In a text editor load the Genesis data file: `CMDSG003_dsg.out`
55. Study briefly the file
56. Using the output file, complete the following table:

Design Variable Name	Original Angle Design Variable Value Reference Solution	Final Angle Design Variable Value Reference Solution	Final Angle Design Variable Value
Angle2-3	0.0	-3.738	
Angle4	30.0	23.91	
Angle5	60.0	16.40	
Angle6	90.0	88.49	
Angle7	45.0	1.58	

Your results might be slightly different from the reference solution provided due to machine precision which can lead to a different local optimum.

Study the OPT File

57. In a text editor load the Genesis data file: `CMDSG003_dsg.OPT`
58. Study briefly the file
59. Considering that the Format of PCOMP is:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z_0	NSM		F.T.	TREF	GE	LAM	MEM
+	MID1	T1	θ_1	SOUT1	MID2	T2	θ_2	SOUT2	
+	MID3	T3	θ_3	SOUT3	-etc.-				

Fill out the angle values, of PCOMP 10, using the following table:

Layer	Angle (PCOMP 10) Initial Design Reference Solution	Angle (PCOMP 10) Last Design Reference Solution	Angle (PCOMP 10) Initial Design	Angle (PCOMP 10) Last Design
1	45.0	45.0		

2	0.0	-3.738		
3	0.0	-3.738		
4	30.0	23.91		
5	60.0	16.40		
6	90.0	88.49		
7	45.0	1.58		

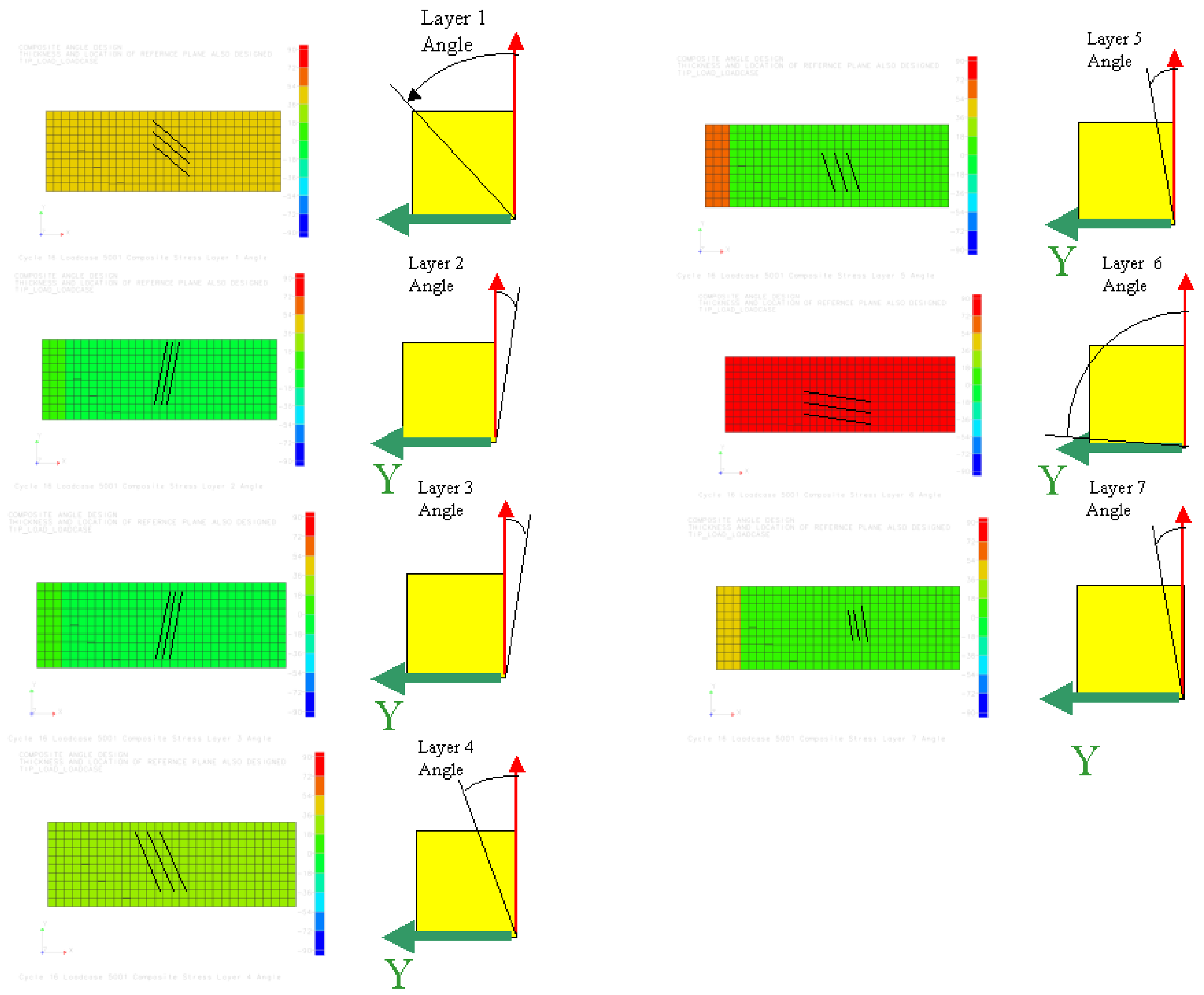
Quit Design Studio

60. From the main menu bar, select **File** → **Quit**
61. Push the **Don't Save** button

Angle Results

The following table and figures, show the angles of each layer

Layer	Angle (PCOMP 10) Last Design
1	45.0
2	-3.738
3	-3.738
4	23.91
5	16.40
6	88.49
7	1.58


Layer 1
Angle
Layer 2
Angle
Y
Layer 3
Angle
Y
Layer 4
Angle
Y
Layer 5
Angle
Y
Layer 6
Angle
Y
Layer 7
Angle
Y

9.4 Shape Optimization of Composite Cantilever Plate using Shape Domains and Morphing Sets

Introduction

The purpose of this example is to learn how to perform shape optimization with composite elements. Most of the design data (objective function and constraints) for this problem is provided. The key data, that you will be creating, is the one needed for the shape optimization. The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

Failure Index$\leq$ 0.5 (all elements)

Designable region:

Six thicknesses of the composite plate (data provided)

Z0 (Location of Reference Plane, data provided)

Six Angles of the composite plate (data provided)

Two shape design variables

Example ID

`CMDSG004`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`CMDSG004.dat`	`Input data`	Provided: Contains the finite element mesh along with applied load and boundary conditions and some design data.
`CMDSG004_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `CMDSG004.dat`
`CMDSG004_dsg.out`	`Output file`	Generated by a Genesis run within Design Studio. This is a Genesis output file.
`CMDSG004_dsgxx.pch`	`Punch Output data`	Generated by a Genesis run within Design Studio. This is a punch output file containing the analysis results for design cycle “`xx`”.

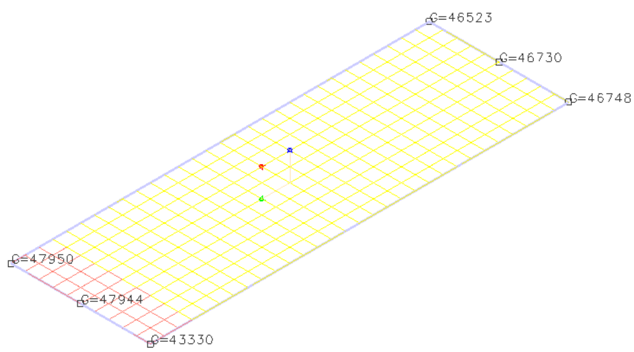
`CMDSG004_dsg.SHP`	`Shape Change data`	Generated using Genesis within Design Studio. This file contains the shape changes during the optimization

Start Design Studio

1. Start Design Studio
2. Open the Design Studio database file: `CMDSG004.dat`

Creating a Shape Domain

3. Select **Design** tab
4. From the **Design** category chooser, select **Shape Domains**
5. Push the **New Domain** button from the Edit Menu toolbar to create a new Shape Domain.
6. Enter Name `rectangle`
7. Check the **Define A New Original Domain** radio button
8. Push **Next>**
9. Select `Domain 57` as Domain group
10. Push **Next**>
11. Select **Quad** as Type
12. Push **Next**>
13. Pick the four corner grids from the viewport in either the clockwise or counter-clockwise direction



You can use the **Select by Grid ID** option in the following order: 43330,47950, 46523 and 46748

14. Push **Next**>

15. Push the **Select Interior Grids** button

    Verify that there are 341 grids selected.

16. Push the **Finish** button

Creating Width Perturbations with Shape Morphing Sets

17. From the **Design** category chooser, select **Shape Morphing Sets**
18. Push the **New Shape Set** button from the Edit Menu toolbar to create a new shape morphing set
19. Enter Name `width`
20. Select **Domain Morphing Set** as Type

    You need to define a new design variable for this shape morphing set. You will create the design variable and then come back to where you are now.
    Do not push **Finish** or **Cancel** at this point.

21. From the **Design** category chooser, select **Design Variables**
22. Push the **New Design Variable** button from the Edit Menu toolbar
23. Enter `width` as the Name and select **Independent Design Variable**
24. Push **Next**>
25. Enter `0.0` for **Initial Value**, `-1.0` for **Lower Bound**, and `1.0` for **Upper Bound**
26. Push the **Finish** button
27. From the **Design** category chooser, select **Shape Morphing Sets**

    This takes you back to where you left off.

28. Select `width` in the **Design Variable** menu
29. Push **Next**>
30. Select `rectangle` from the Shape Domain list
31. Push **Next**>
32. Push the **Select None** button
33. Use the **Select by Grid ID** textbox to select grids `46523` and `47950` in order

    Verify that there is “2 grids selected”. The perturbation will be applied on this 2 grids

34. Push the **By 2 Grids...** button
35. Use the Select by Grid ID textbox to select grids `46748` and `46523` in order

    This defines the direction of the perturbation from 46748 to 46523

36. Push **Next**>

37. Enter: `8.0` as the Magnitude of the perturbation
38. Push **Add Perturbation**

    Verify that there is “2 perturbations on 2 grids”.
39. Push the **Finish** button

Creating Curvature Perturbations with Shape Morphing Sets

40. From the **Design** category chooser, select **Shape Morphing Sets**
41. Push the **New Shape Set** button from the Edit Menu toolbar to create a new shape morphing set
42. Enter Name `curvature`
43. Select **Domain Morphing Set** as Type

    You will create a design variable and then come back.
    Do not push **Finish** or **Cancel** at this point.
44. From the **Design** category chooser, select **Design Variables**
45. Push the **New Design Variable** button from the Edit Menu toolbar to create a new design variable
46. Enter `curvature` as the **Name** and select **Independent Design Variable**
47. Push **Next**>
48. Enter `0.1` for **Initial Value**, `-1.0` for **Lower Bound**, and `1.0` for **Upper Bound**.

    Use a small number (0.1) for Initial Value to avoid possible zero gradient.
49. Push the **Finish** button
50. From the **Design** category chooser, select **Shape Morphing Sets**

    This takes you back to where you left off.
51. Select `curvature` in the **Design Variable** menu
52. Push **Next**>
53. Select `rectangle` from the Shape Domain list
54. Push **Next**>
55. Push the **Select None** button
56. Pick grids 46730 and 47944 in order

    Verify that there is “2 grids selected”. This identifies where the perturbation will be applied
57. Enter `0.0` for **X**, `0.0` for **Y**, `1.0` for **Z**
58. Enter: `5.0` as the **Magnitude** of the perturbation

59. Push **Add Perturbation**

    Verify that there is “2 perturbations on 2 grids”

60. Push the **Finish** button

Save the Design Studio Database File

61. From the main menu bar, select **File** → **Save As...**
62. Enter `CMDSG004` as the Filename and push **Save (**as a Design Studio File**)**

Checking Perturbations

It is always good ideas to check if the combination of shape domain and perturbations will change the structural shape as you intended before you start the optimization.

63. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
64. Under the **Deform Mesh**, select a Shape Morphing Set Preview
65. Select the **Oscillate** radio button
66. Push the **Up** button

    One can also run Genesis in check mode and load the DVG file which contains candidate shape changes, then visualize them in **Deform Mesh/Color Mesh** in Design Studio.

Optimizing and Viewing Results

67. From the main menu bar, select **Genesis** → **Optimize**
68. When the optimization is done, select the **Import Post...** button from the **Genesis Console Output** window
69. Select the `CMDSG004_dsg.SHP` file

    You can view individual frames with **Deform Mesh/Color Mesh** or you can make an animation with **Animation**. You can also animate shape changes and element results (e.g. stresses) simultaneously.

70. Select the `CMDSG004_dsgxx.pch` punch result files

    Make sure to select all the files using the **Shift** key

71. Push the **Import** button
72. When importing is done, select the **Close** button from the **Genesis Console Output** window
73. Push **Animation** in the **Post** tab
74. Select **Shape Change** from the **Deform Results Type** menu

75. Select **Composite Failure Index** from the **Color Results Type** menu
76. Select **Filled Contours**
77. Push **Next>**
78. Select all the Cycles for Shape Change from the list
79. Push **Next>**
80. Select all the Cycles for Loadcase 5001 Failure Index from the list

    Also test with different loadcases later.
81. Push **Next>**
82. Push the **Finish** button
83. Push the **Deform/Mesh Color Mesh** button
84. Select the Composite Failure Index Result for the last design cycle
85. Will the design fail?

Quit Design Studio

86. From the main menu bar, select **File** → **Quit**
87. Push the **Don't Save** button

Study the Results

88. Open the output file `CMDSG004_dsg.out`
89. Study the values of the objective and the constraint and complete the following table

Type	Initial Value	Final Value	change%
Objective			
Maximum Constraint Violations			

9.5 Topometry Optimization of a Composite Plate using Discrete Angles with/without Coarsening

Introduction

The purpose of this example is to introduce the basic steps to create and solve a simple topometry optimization problem on a composite material with seven layers. In this problem, the basic steps to create discrete design variables are reviewed. The problem is divided in two parts. In part 1 the coarsening option of topometry is not used, while in part 2 it is used.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

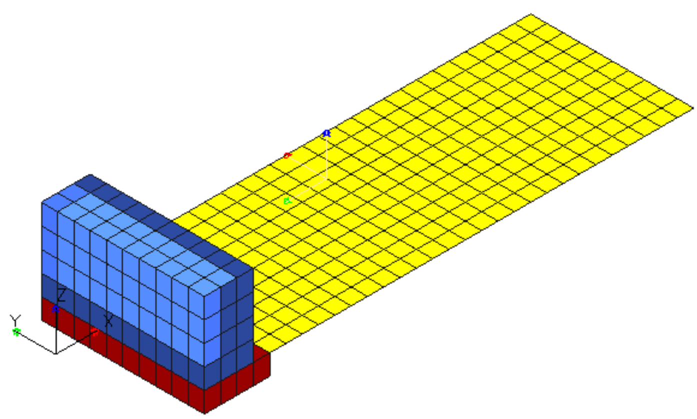
Subject to:

Failure Index $\leq$ 0.9 (all elements)

Designable region:

Element of PCOMP 10

The following picture shows the structure:



Example ID

CMDSG005

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
CMDSG005_1.dat	Input data	Provided: Contains the finite element mesh along with applied load and boundary conditions.
CMDSG005_1_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in CMDSG005_1.dat
CMDSG005_1_dsgSSOLxx.dat	Input data	Generated by a Genesis run within Design Studio. This is the input data of a solid model obtained using the thicknesses of the shells for design cycle "xx". The file is for visualization purposes and not intended for analysis.
CMDSG005_1_dsgOPOSTxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the element thickness results for design cycle "xx".
CMDSG005_1_dsgxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the analysis results for design cycle "xx".
CMDSG005_1_ref.dat	Input data	Provided: Reference file similar to the file CMDSG005_1_dsg.dat
CMDSG005_2.dat	Input data	Generated by Design Studio: File exported at the end of part 1. Used as starting file for part 2. Similar to CMDSG005_1_dsg.dat or CMDSG005_1_ref.dat
CMDSG005_2_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in CMDSG005_2.dat
CMDSG005_2_dsgxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the analysis results for design cycle "xx".
CMDSG005_2_ref.dat	Input data	Provided: Reference file similar to the file CMDSG005_2_dsg.dat
CMDSG005_2_dsgUPDATExx.dat	Input data	Generated by a Genesis run within Design Studio. This is the updated input data for design cycle "xx".
CMDSG005_2_dsgSSOLxx.dat	Input data	Generated by a Genesis run within Design Studio. This is the input data of a solid model obtained using the thicknesses of the shells for design cycle "xx". The file is for visualization purposes and not intended for analysis.
CMDSG005_2_dsgOPOSTxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the element thickness results for design cycle "xx".

9.5.1 Part 1

The main purpose of this part of the example is to learn how to create the necessary design data in order to perform topometry optimization.

When you finish this example, you should have created a file named: `CMDSG005_2.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG005_1.dat`

Create Seven Independent Design Variables for the Thickness

3. Select the **Design** tab
4. From the category chooser, select **Design Variable**
5. Push the **New Design Variable** button from the Edit Menu toolbar
6. For **Name,** enter `T1`
7. Accept default type of design variables, **Independent Design Variables**
8. Push the **Next>** button
9. Using the following table, enter the appropriate values for **Initial value**, **Lower bound** and **Upper bound**:

Name	Initial Value	Lower Bound	Upper Bound
T1	0.3	0.005	3.0

10. Push the **Finish** button
11. Repeat the steps from 5 to 10 to create six additional design variables as follows:

You can also use the shortcut **Ctrl+C - Crtl+V** to copy - paste the design variables. Be sure to use the correct values for the initial value, the lower bounds and the upper bound.

Name	Initial Value	Lower Bound	Upper Bound
T2	0.2	0.005	3.0
T3	0.2	0.005	3.0
T4	0.2	0.005	3.0
T5	0.2	0.005	3.0

T6	0.2	0.005	3.0
T7	0.3	0.005	3.0

Create Seven Discrete Independent Design Variables for the Angles

12. Select the **Design** tab
13. From the category chooser, select **Design Variables**
14. Push the **New Design Variable** button from the Edit Menu toolbar
15. For **Name,** enter `A1`
16. Select the **Discrete Design Variable** radio button
17. Push the **Next>** button
18. Using the following table, enter the appropriate values for **Initial value**, **Lower bound**, **Upper bound, First Discrete Value**, **Increment and Num. of additional values.** ,

Name	Initial Value	Lower Bound	Upper Bound	First Discrete Value	Increment	Num. of additional values
A1	45.0	-90.0	90.0	-90	5	36

To generate and add Values push **the Generate & Add Values** button.

19. Push the **-** button to delete the first Discrete Value
20. Push the **Finish** button
21. Repeat the steps 14 to 20 to create six additional design variables as follows:
You can also use the shortcut ctrl+C - crtl+V to copy - paste the design variables. Be sure to use the good values for the initial value, the lower bounds, the upper bound, the first discrete Value, the increment and the num. of additional values.

Name	Initial Value	Lower Bound	Upper Bound	First Discrete Value	Increment	Num. of additional values
A2	0.0	-90.0	90.0	-90	5	36
A3	0.0	-90.0	90.0	-90	5	36
A4	30.0	-90.0	90.0	-90	5	36
A5	60.0	-90.0	90.0	-90	5	36

A6	90.0	0.0	180.0	0	5	36
A7	45.0	-90.0	90.0	-90	5	36

Create the Sizing Region

22. From the **Design** category chooser, select **Sizing**
23. Select the PCOMP 10
24. Push the **Modify Sizing Design** button from the Edit Menu toolbar
25. For the **Thickness** of **Layer 1,** select design variable T1
26. For the **Angle** of **Layer 1,** select design variable A1
27. For the **Thickness** of **Layer 2,** select design variable T2
28. For the **Angle** of **Layer 2,** select design variable A2
29. For the **Thickness** of **Layer 3,** select design variable T3
30. For the **Angle** of **Layer 3,** select design variable A3
31. For the **Thickness** of **Layer 4,** select design variable T4
32. For the **Angle** of **Layer 4,** select design variable A4
33. For the **Thickness** of **Layer 5,** select design variable T5
34. For the **Angle** of **Layer 5,** select design variable A5
35. For the **Thickness** of **Layer 6,** select design variable T6
36. For the **Angle** of **Layer 6,** select design variable A6
37. For the **Thickness** of **Layer 7,** select design variable T7
38. For the **Angle** of **Layer 7,** select design variable A7
39. Push the **Finish** button

    Verify that the hammer icon is next to PCOMP 10.
    The hammer icon indicates that the PCOMP group is being size-designed.

Define a Topometry Optimization

40. From the **Design** category chooser, select **Topometry**
41. Select the PCOMP 10
42. Push the **Modify Topometry Design** button from the Edit Menu toolbar
43. Push the **Finish** button

Defining the Design Objective

44. From the category chooser, select **Objectives**
45. Push the **New Objective** button from the Edit Menu toolbar
46. For **Name,** enter `Mass`
47. Accept the default, **Mass** response type
48. Accept the **Min** Objective Definition switch
49. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Defining the Design Constraints

50. From the category chooser, select **Constraints**
51. Push the **New Constraint** button from the Edit Menu toolbar
52. For **Name,** enter `Findex`
53. Select the **Failure Index** response
54. Select **Selected Groups** option of Failure Index
55. Enter `0.9` for the **Upper Bound**
56. Push the **Next>** button
57. Select PCOMP 10
58. Push the **Next>** button
59. Select the static loadcase
60. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Clearing the Selection

61. Right-click the Viewport, select **Clear→ All**

Request Element Stresses and Element Forces

62. Select the **Analysis** tab
63. From the category chooser, select **Loadcases**
64. Select the existing loadcase

65. Push the **Modify Loadcase** button from the Edit menu toolbar
66. Push the **Next>** button
67. Push the **Next>** button
68. Push the **Next>** button
69. For **Element Stress**, make sure that **Post** is selected

    The layer thicknesses as well as the layer angles are printed along with the element stresses.

70. For **Element Force**, make sure that **Post** is selected

    The layer failure indices are printed along with the element forces.

71. Push the **Finish** button

Request the SSOL File

The SSOL file is a Genesis input file where the shell elements will be converted into solid elements for the purpose of better visualization of the thicknesses distribution. This file is not intended to be used for analysis or optimization.

72. From the main menu bar, select **Genesis** → **Options...**
73. Select the **File Control** tab
74. For **Shell-to-Solid File**, pick **Create(Fixed Norms)**

    The name of the SSOL file will be:
    `CMDSG005_1_dsgSSOLxx.dat` (where `xx` corresponds to the last design cycle number).

75. Push the **Apply** button

Request the OPOST (Sizing data) Post-Processing File

The OPOST file is a Genesis post-processing file that contains the optimization results, in this case the thickness of the composite elements. This file only contains the total thickness of the composite, it does not contain the individual layer thicknesses.

76. From the main menu bar, select **Genesis** → **Options**
77. Select the **File Control** tab
78. For **Element Sizing File**, choose **Create**
79. Push the **Apply** button

Increase the Maximum Number of Design Cycles

80. From the main menu bar, select **Genesis** → **Options...**

81. Select the **Design Control** tab
82. Enter `30` for the **Maximum Design Cycles**
83. Push the **Apply** button

Optimize the Structure Using Genesis

84. From the main menu bar, select **Genesis** → **Optimize**
85. Study the **Design History** charts; when done, push the **Close** button
86. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Analysis Post-Processing Files

87. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
88. Select the `CMDSG005_1_dsg00.pch` file and check the **Import Similar Results for All Design Cycles** check box
89. Push the **Open** button

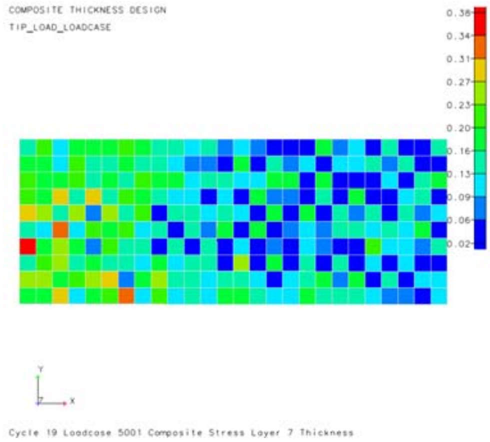
Post-Processing some Analysis Results

90. Select the **Post** tab
91. Push the **Deform Mesh/Color Mesh** button
92. Select the Composite Failure Index Result for the last design cycle
93. Select the Composite Stress Result for the last design cycle
94. From the category chooser in the **Color Mesh** option, change **Von Mises** to **Layer Thickness**

    Now you visualize the layer thickness 1.

95. In the box near the category chooser, change 1 to 7.

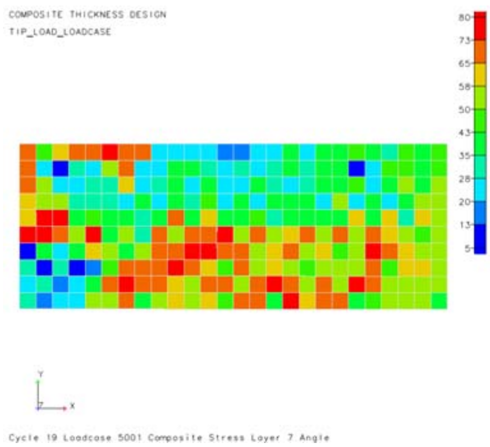
    Now you visualize the layer thickness 7.
    For example this is the layer thickness 7 for the last cycle:



    The results can change a little bit depending on the computer.

96. From the category chooser (drop-down listbox by the side of the **Options** button) in the **Color Mesh** option, change **Layer Thickness** to **Layer Angle**

    Now you visualize the layer angle 7.
    For example this is the layer angle 7 for the last cycle:



97. In the box near the category chooser, change 7 to 1

    Now you visualize the layer angle 1.

98. Push the **Up** button

Export the Input File

99. From the main menu bar, select **File** → **Export** → **Input Data...** (or Ctrl-E)

100. Enter `CMDSG005_2.dat` as Filename

101. Push the **Save** button

Study the SSOL File

102. From the main menu bar, select **File** → **New**

103. Push the **Don't Save** button

104. Import the Genesis data file: `CMDSG005_1_dsgSSOLxx.dat`

    xx corresponds to the last design cycle number

105. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**

106. For **Files of Type**, choose **All Files**

107. Select the `CMDSG005_1_dsgOPOSTxx.pch` file

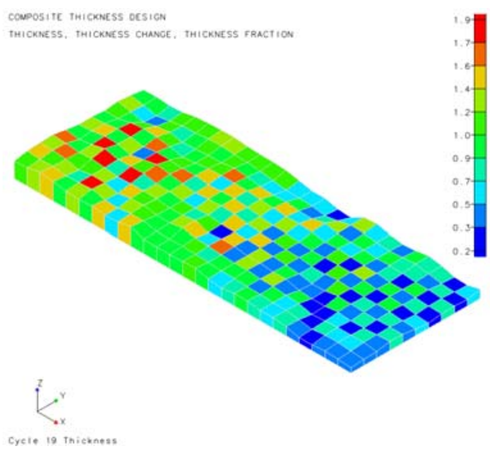
108. Push the **Open** button

109. Select the **Post** tab

110. Push the **Deform Mesh/Color Mesh** button

111. Select the Thickness Result for the last design cycle

112. Study the Viewport



Quit Design Studio

113. From the main menu bar, select **File** → **Quit**

114. Push the **Don't Save** button

Study the Output File

115. In a text editor load the Genesis data file: `CMDSG005_1_dsg.out`

116. Study briefly the file

117. Using the output file, complete the following table:

Type	Initial Value	Final Value Reference Solution (1)	Final Value (2)
Objective Function (Mass)	2.3036E-06	2.0162E-06	
Maximum Constraint Violation	0	0	

(1) Result from `CMDSG005_1_ref.dat`
(2) Result from your run, `CMDSG005_1_dsg.out`

118. Study the bottom of the output file:

There are three different kinds of constraints for this problem:

The first one is the maximum constraint violation which is the Findex in this case. First Genesis optimizes with continuous variables (it is why you can see the max constraint violation decrease quickly and increase again when Genesis starts to use discrete variables).
For example, if Findex=1.0, the maximum constraint violation will be: (1.0 - 0.5)/0.5=100%
0.9 is the constraint bound.

The second one is the maximum discrete violation absolute criteria. When Genesis uses the continuous variables the discrete criteria isn't considered. It is when Genesis starts to use the discrete variables that Genesis will try to reduce this maximum discrete violation.
For example, if the value is 52.21, the lower nearby value is 50 and the upper nearby value is 55, the maximum discrete violation absolute criteria is: (52.21-50)/50=4.42%
Genesis takes the closest number from the value so in this case it is 50.

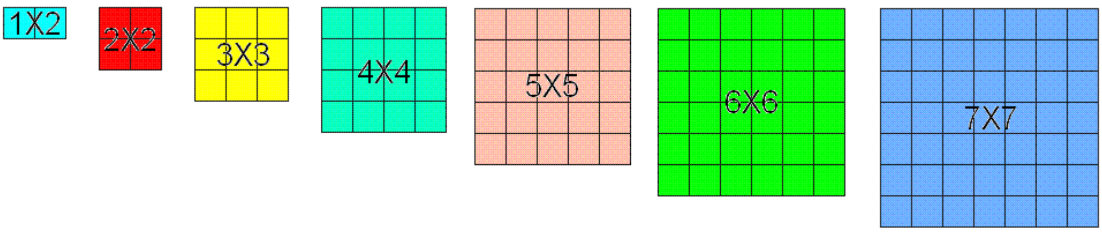
The third one is the maximum discrete violation bias criteria. The difference with the absolute criteria is that the bias measures the distance between the value and the closest number.
For example, if the value is 52.21, the lower nearby value is 50 and the upper nearby value is 55, the maximum discrete violation bias criteria is: (52.21-50)/(55-50)=44.2%
55-50 is the distance between the lowest and the upper closest number.

9.5.2 Part 2

The purpose of this part of the example is to create the necessary data to use the coarse option available for topometry optimization.

Coarse topometry, is a group by group sizing optimization capability, as opposed to element by element sizing optimization when no coarse option is used. The groups are automatically created by the program based on a user specified number that indicates how many elements should be in each group. The groups are formed using a technique that groups elements that touch each other. Coarse topometry, is a compromise between sizing and standard topometry optimization.

Common values of Max elements used are 2, 4, 9, 16, 25, 36, 49.In an ideal situation all groups would look similar to the following picture:.



In practice sometimes we do not get groups perfectly formed as shown above, but usually the groups are good enough to represent a common area. Any number can be used. In this problem, a value of 4 will be used.

The main purpose of using the coarse option is to reduce the time. In large problems this could be very important.

If you do not have the `CMDSG005_2.dat` file generated in part 1, copy the file `CMDSG005_1_ref.dat to CMDSG005_2.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG005_2.dat`

Set the Coarse Conditions in Topometry

3. From the **Design** category chooser, select **Topometry**
4. Select the PCOMP 10
5. Push the **Modify Topometry Design** button from the Edit Menu toolbar

6. For **Coarse Method:** select **Max elements per design variable**
7. For **Coarse Parameter**, enter 4 for the value
8. Push the **Finish** button

Optimize the Structure Using Genesis

9. From the main menu bar, select **Genesis** → **Optimize**
10. Study the **Design History** charts; when done, push the **Close** button
11. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Analysis Post-Processing Files

12. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
13. Select the `CMDSG005_2_dsg00.pch` file and check the **Import Similar Results for All Design Cycles** check box
14. Push the **Open** button

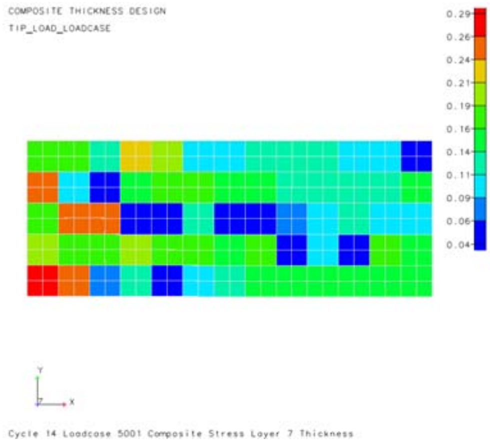
Post-Processing some Analysis Results

15. Select the **Post** tab
16. Push the **Deform Mesh/Color Mesh** button
17. Select the Composite Failure Index Result for the last design cycle
18. Select the Composite Stress Result for the last design cycle
19. From the category chooser in the **Color Mesh** option, change **Von Mises** to **Layer Thickness**

   Now you visualize the layer thickness 1.

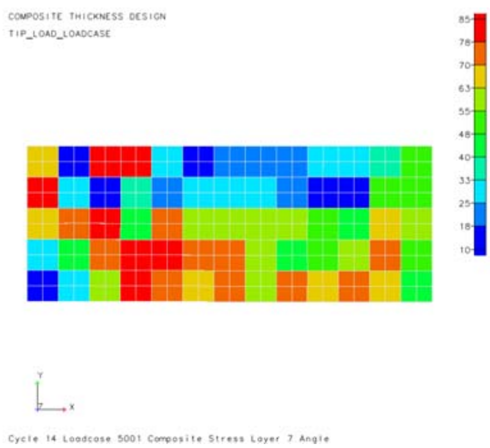
20. In the box near the category chooser, change 1 to 7.

    Now you visualize the layer thickness 7.
    For example this is the layer thickness 7 for the last cycle:



21. From the category chooser (drop-down listbox by the side of the **Options** button) in the **Color Mesh** option, change **Layer Thickness** to **Layer Angle**

    Now you visualize the layer angle 7.
    For example this is the layer angle 7 for the last cycle:



22. In the box near the category chooser, change 7 to 1

    Now you visualize the layer angle 1.

Quit Design Studio

23. From the main menu bar, select **File** → **Quit**

24. Push the **Don’t Save** button

Study the Initial Updated File

25. In a text editor load the Genesis data file: `CMDSG005_2_dsgUPDATE00.dat`
26. Study briefly the file
27. How many PCOMP entries exists in the file?

    Reference answer: 56
    Note: There are 260 designable elements.
    Note: 260/5=52 (close to 56).

Study the Final (Optimized) Updated File

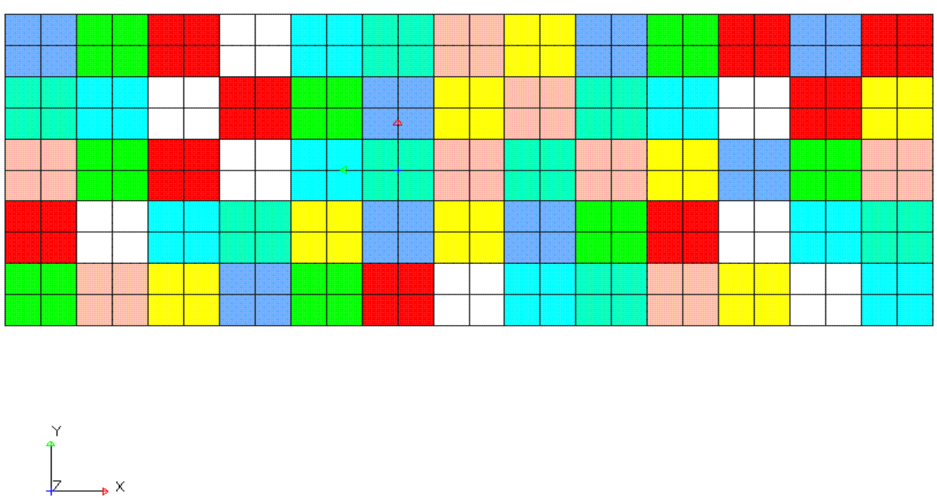
28. In a text editor load the Genesis data file: `CMDSG005_2_dsgUPDATExx.dat` (xx corresponds to the last design cycle)
29. Study briefly the file

Start Design Studio

30. Start Design Studio
31. Import the Genesis data file: `CMDSG005_2_dsgUPDATE00.dat`

Study the Update File in Design Studio

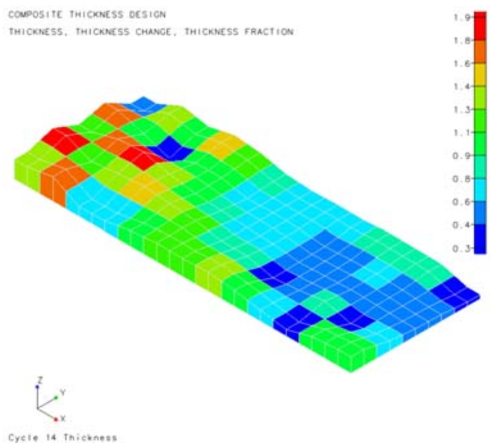
32. Study briefly the file in the Viewport



Study the SSOL File

33. From the main menu bar, select **File** → **New**
34. Push the **Don’t Save** button
35. Import the Genesis data file: `CMDSG005_2_dsgSSOLxx.dat`

    xx corresponds to the last design cycle number
36. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
37. For **Files of Type**, choose **All Files**
38. Select the `CMDSG005_2_dsgOPOSTxx.pch` file
39. Push the **Open** button
40. Select the **Post** tab
41. Push the **Deform Mesh/Color Mesh** button
42. Select the Thickness Result for the last design cycle
43. Study the Viewport



Quit Design Studio

44. From the main menu bar, select **File** → **Quit**
45. Push the **Don’t Save** button

Study the Output File

46. In a text editor load the Genesis data file: `CMDSG005_2_dsg.out`
47. Study briefly the file

48. Using the output file, complete the following table:

Type	Initial Value	Final Value Reference Solution (1)	Final Value (2)
Objective Function (Mass)	2.3036E-06	2.0227E-06	
Maximum Constraint Violation	0	0	

(1) Result from `CMDSG005_2_ref.dat`
(2) Result from your run, `CMDSG005_2_dsg.out`

Compare Part1 and Part2

49. Using the two output files, complete the following table:

Quantity	Part1 without Coarse Reference Solution (1)	Part2 with Coarse Reference Solution (2)	Part1 without Coarse (3)	Part2 with Coarse (4)
NUMBER OF INDEPENDENT DESIGN VARIABLES	3120	910		
NUMBER OF DISCRETE DESIGN VARIABLES	1560	455		
NUMBER OF TOPOMETRY DESIGNABLE ELEMENTS	260	260		
NUMBER OF TOPOMETRY DESIGNABLE GROUPS	260	65		
NUMBER OF TOPOMETRY ELEMENTS/GROUPS	1	4		
ELAPSED TIME (sec)	502	236		
% REDUCTION IN OBJECTIVE FUNCTION (MASS)	12.5	12		

(1) Result from `CMDSG005_1_ref.dat`
(2) Result from `CMDSG005_2_ref.dat`
(3) Result from your run, `CMDSG005_1_dsg.out`
(4) Result from your run, `CMDSG005_2_dsg.out`

50. Using the tables you filled before (part 1-step 124 and part 2-step 48), complete the following table:

Type	Part1-Part2 Initial Value	Part1 Final Value Reference Solution (1)	Part2 Final Value Reference Solution (2)	Part1 Final Value (3)	Part2 Final Value (3)
Objective Function (Mass)	2.3036E-06	2.0162E-06	2.0227E-06		
Maximum Constraint Violation	0	0	0		

(1) Result from `CMDSG005_1_ref.dat`
(2) Result from `CMDSG005_2_ref.dat`
(3) Result from your run, `CMDSG005_1_dsg.out`
(4) Result from your run, `CMDSG005_2_dsg.out`

9.6 Topometry Optimization of a Composite Plate using Linked Groups

Introduction

The purpose of this example is to introduce the basic steps to create and solve a simple composite optimization problem with four PCOMP entries. This example will show how to build a master PCOMP entry and three slave PCOMP entries. This example will also review the basic steps to define symmetry conditions for topometry optimization and show the difference between Negate and Repeat options for splitting the angles of the PCOMP.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

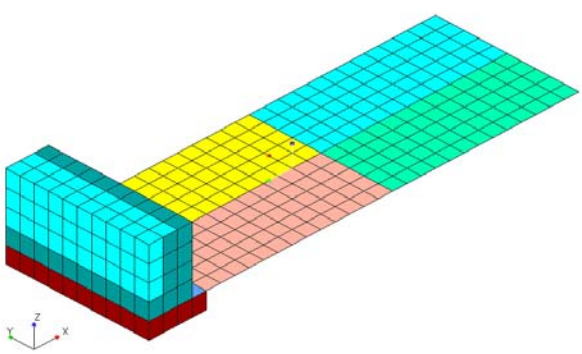
Subject to:

Failure Index $\leq$ 0.5 (all elements)

Designable region:

Elements of 4 PCOMP groups

The following picture shows the structure:



Example ID

CMDSG006

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
CMDSG006_1.dat	Input data	Provided: Contains the finite element mesh along with applied load and boundary conditions.
CMDSG006_1_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in CMDSG006_1.dat
CMDSG006_1_dsgSSOLxx.dat	Input data	Generated by a Genesis run within Design Studio. This is the input data of a solid model obtained using the thicknesses of the shells for design cycle “xx”. The file is for visualization purposes and not intended for analysis.
CMDSG006_1_dsgOPOSTxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the element thickness results for design cycle “xx”.
CMDSG006_1_dsgxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the analysis results for design cycle “xx”.
CMDSG006_1_ref.dat	Input data	Provided: Reference file similar to the file CMDSG006_1_dsg.dat
CMDSG006_2.dat	Input data	Generated by Design Studio: File exported at the end of part 1. Used as starting file for part 2. Similar to CMDSG006_1_dsg.dat or CMDSG006_1_ref.dat
CMDSG006_2_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in CMDSG006_2.dat
CMDSG006_2_dsgxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the analysis results for design cycle “xx”.
CMDSG006_2_ref.dat	Input data	Provided: Reference file similar to the file CMDSG006_2_dsg.dat
CMDSG006_2_dsgSSOLxx.dat	Input data	Generated by a Genesis run within Design Studio. This is the input data of a solid model obtained using the thicknesses of the shells for design cycle “xx”. The file is for visualization purposes and not intended for analysis.
CMDSG006_2_dsgOPOSTxx.pch	Punch Output data	Generated by a Genesis run within Design Studio. This is a punch output file containing the element thickness results for design cycle “xx”.

9.6.1 Part 1

The main purpose of this part of the example is to learn how to use Master and Slaves Option to perform Topometry Optimization with the Negate Option.

The master and Slaves Option is used to reduce data creation because you need to create only the parameters for the Master, and this option will enforce the Global symmetry.

When you finish this example, you should have created a file named: `CMDSG006_2.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG006_1.dat`

Create the Sizing Region for the master PCOMP

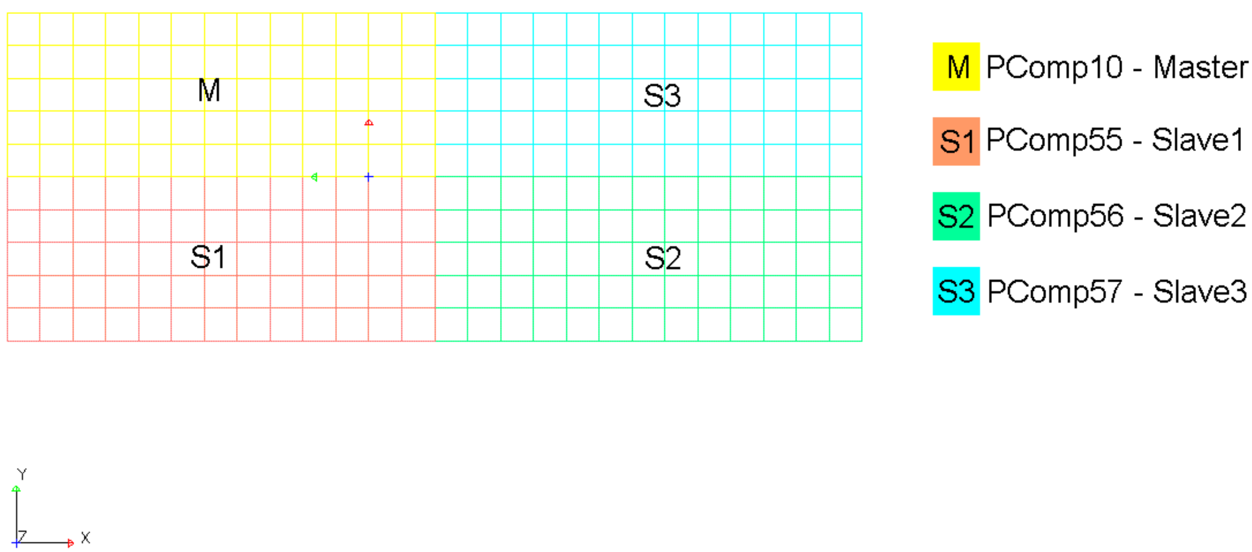
3. From the **Design** category chooser, select **Sizing**
4. Select the PCOMP 10
5. Push the **Modify Sizing Design** button from the Edit Menu toolbar
6. For the **Thickness** of **Layer 2,** select design variable `T2`
7. For the **Angle** of **Layer 2,** select design variable `A2`
8. For the **Thickness** of **Layer 3,** select design variable `T3`
9. For the **Angle** of **Layer 3,** select design variable `A3`
10. For the **Thickness** of **Layer 4,** select design variable `T4`
11. For the **Angle** of **Layer 4,** select design variable `A4`
12. For the **Thickness** of **Layer 5,** select design variable `T5`
13. For the **Angle** of **Layer 5,** select design variable `A5`
14. For the **Thickness** of **Layer 6,** select design variable `T6`
15. For the **Angle** of **Layer 6,** select design variable `A6`
16. For the **Thickness** of **Layer 7,** select design variable `T7`
17. For the **Angle** of **Layer 7,** select design variable `A7`
18. Push the **Finish** button

    Verify that the hammer icon is next to PCOMP 10.
    The hammer icon indicates that the PCOMP group is being size-designed.

Define a Topometry Optimization and Create the Sizing Region for the slaves PCOMP

19. From the **Design** category chooser, select **Topometry**
20. Select the PCOMP 10
21. Push the **Modify Topometry Design** button from the Edit Menu toolbar
22. Push the **Linked Groups...** button
23. Select the three Slaves PCOMP: PCOMP 55 (`Slave1`), PCOMP 56 (`Slave2`) and PCOMP 57 (`Slave3`)

    Hold the **Ctrl** key to select the threes slaves PCOMP
    Note: The element of PCOMP 55, PCOMP 56, and PCOMP 57 are going to be design using the property of PCOMP 10.



24. Push the **Next>** button
25. Push the **Change** button for the Symmetry Coord. Sys
26. Select the `Angle_Coordinate_System`
27. Push the **Next>** button
28. For **Symmetry 1**, choose **MYZ: Mirror about YZ plane**
29. Push the **Advanced...** button
30. For **PCOMP Mirrored Angles**, choose **Negate**
31. Push the **Next>** button

32. Push the **Finish** button

    Verify that the rings icons are next to PCOMP 55, PCOMP 56 and PCOMP 57.
    Verify that from the Design category if you select Sizing, the rings icons are next to PCOMP 55, PCOMP 56 and PCOMP 57.

Defining the Design Objective

33. From the category chooser, select **Objectives**
34. Push the **New Objective** button from the Edit Menu toolbar
35. For **Name,** enter `Mass`
36. Accept the default, **Mass** response type, and **Entire Model** in the pull down menu
37. Accept the **Min** Objective Definition switch
38. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Defining the Design Constraints

39. From the category chooser, select **Constraints**
40. Push the **New Constraint** button from the Edit Menu toolbar
41. For **Name,** enter `Findex`
42. Select the **Failure Index** response
43. Select **Selected Groups** option of Failure Index
44. Enter `0.5` for the **Upper Bound**
45. Push the **Next>** button
46. Select PCOMP 10, PCOMP 55, PCOMP 56 and PCOMP 57
47. Push the **Next>** button
48. Select the static loadcase
49. Push the **Finish** button

Clearing the Selection

50. Right-click the Viewport, select **Clear**→ **All**

Request the SSOL File

The SSOL file is a Genesis input file where the composite elements will be converted into solid elements for the purpose of better visualizing the thicknesses distribution. This file is not intended to be used for analysis or optimization.

51. From the main menu bar, select **Genesis** → **Options...**
52. Select the **File Control** tab
53. For **Shell-to-Solid File**, pick **Create(Fixed Norms)**

    The name of the SSOL file will be:
    `CMDSG006_1_dsgSSOLxx.dat` (where xx corresponds to the last design cycle number).
54. Push the **Apply** button

Request the OPOST (Sizing Data) Post-Processing File

The OPOST file is a Genesis post-processing file that contains the optimization results, in this case the thickness of the composite elements. This file only contains the total thickness of the composite, it does not contain the individual layer thicknesses.

55. From the main menu bar, select **Genesis** → **Options**
56. Select the **File Control** tab
57. For **Element Sizing File**, choose **Create**
58. Push the **Apply** button

Optimize the Structure Using Genesis

59. From the main menu bar, select **Genesis** → **Optimize**
60. Study the **Design History** charts; when done, push the **Close** button
61. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Analysis Post-Processing Files

62. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
63. Select the `CMDSG006_1_dsg00.pch` file and check the **Import Similar Results for All Design Cycles** check box
64. Push the **Open** button

Post-Processing some Analysis Results

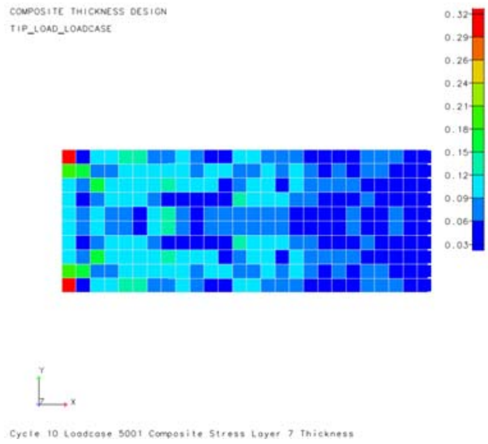
65. Select the **Post** tab

66. Push the **Deform Mesh/Color Mesh** button
67. Select the Composite Failure Index Result for the last design cycle
68. Select the Composite Stress Result for the last design cycle
69. From the category chooser in the **Color Mesh** option, change **Von Mises** to **Layer Thickness**

    Now you visualize the layer thickness 1.

70. In the box near the category chooser, change 1 to 7.

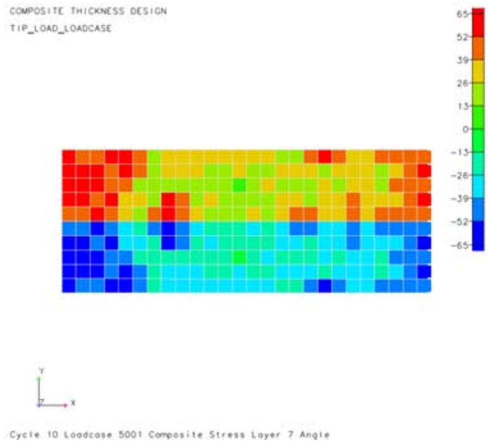
    Now you visualize the layer thickness 7.



The results can change a little bit depending on the computer.

71. From the category chooser (drop-down listbox by the side of the **Options** button) in the **Color Mesh** option, change **Layer Thickness** to **Layer Angle**

    Now you visualize the layer angle 7.



72. In the box near the category chooser, change 7 to 1

    Now you visualize the layer angle 1.

Export the Input file

73. From the main menu bar, select **File** → **Export** → **Input Data...** (or Ctrl-E)
74. Enter `CMDSG006_2.dat` as Filename
75. Push the **Save** button

Quit Design Studio

76. From the main menu bar, select **File** → **Quit**
77. Push the **Don't Save** button

Study the Output File

78. In a text editor load the Genesis data file: `CMDSG006_1_dsg.out`
79. Study briefly the file
80. Using the output file, complete the following table:

Type	Initial Value	Final Value Reference Solution (1)	Final Value (2)
Objective Function (Mass)	2.3452E-06	1.9752E-06	
Maximum Constraint Violation	0	0	

(1) Result from `CMDSG006_1_ref.dat`
(2) Result from your run, `CMDSG006_1_dsg.out`

9.6.2 Part 2

The main purpose of this part of the example is to learn how to use the Repeat Option for the PCOMP Mirrored Angles

If you do not have the `CMDSG006_2.dat` file generated in part 1, copy the file `CMDSG006_1_ref.dat to CMDSG006_2.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG006_2.dat`

Change the PCOMP Mirrored Angles

3. From the **Design** category chooser, select **Topometry**
4. Select the PCOMP 10
5. Push the **Modify Topometry Design** button from the Edit Menu toolbar
6. Push the **Advanced...** button
7. For **PCOMP Mirrored Angles** select **Repeat**

   Negate: Angles of symmetrical elements are forced to have the same magnitudes but opposite signs
   Repeat: Angles of symmetrical elements are forced to have the same magnitudes and same-signs

8. Push the **Next>** button
9. Push the **Finish** button

Request the SSOL File

The SSOL file is a Genesis input file where the composite elements will be converted into solid elements for the purpose of better visualizing the thicknesses distribution. This file is not intended to be used for analysis or optimization.

10. From the main menu bar, select **Genesis** → **Options...**
11. Select the **File Control** tab
12. For **Shell-to-Solid File**, pick **Create(Fixed Norms)**

    The name of the SSOL file will be:
    `CMDSG006_2_dsgSSOLxx.dat` (where xx corresponds to the last design cycle number).

13. Push the **Apply** button

Request the OPOST (Sizing data) Post-Processing File

The OPOST file is a Genesis post-processing file that contains the optimization results, in this case the thickness of the composite elements. This file only contains the total thickness of the composite, it does not contain the individual layer thicknesses.

14. From the main menu bar, select **Genesis** → **Options**
15. Select the **File Control** tab
16. For **Element Sizing File**, choose **Create**
17. Push the **Apply** button

Optimize the Structure Using Genesis

18. From the main menu bar, select **Genesis** → **Optimize**
19. Study the **Design History** charts; when done, push the **Close** button
20. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Analysis Post-Processing Files

21. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
22. Select the `CMDSG006_2_dsg00.pch` file and check the **Import Similar Results for All Design Cycles** check box
23. Push the **Open** button

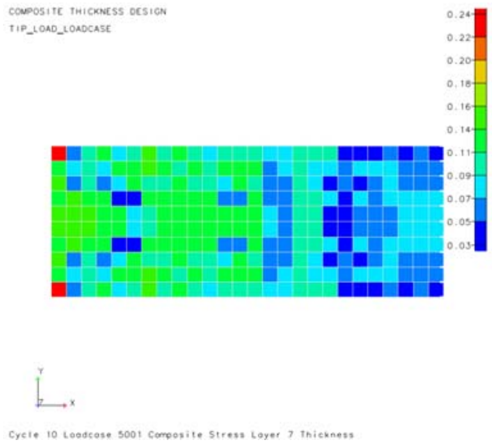
Post-Processing some Analysis Results

24. Select the **Post** tab
25. Push the **Deform Mesh/Color Mesh** button
26. Select the Composite Failure Index Result for the last design cycle
27. Select the Composite Stress Result for the last design cycle
28. From the category chooser in the **Color Mesh** option, change **Von Mises** to **Layer Thickness**

    Now you visualize the layer thickness 1.

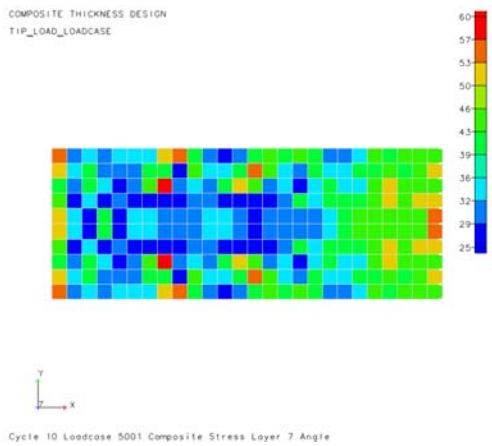
29. In the box near the category chooser, change 1 to 7.

    Now you visualize the layer thickness 7.



30. From the category chooser (drop-down listbox by the side of the **Options** button) in the **Color Mesh** option, change **Layer Thickness** to **Layer Angle**

    Now you visualize the layer angle 7.



31. In the box near the category chooser, change 7 to 1

    Now you visualize the layer angle 1.

Quit Design Studio

32. From the main menu bar, select **File** → **Quit**
33. Push the **Don't Save** button

Study the Output File

34. In a text editor load the Genesis data file: `CMDSG006_2_dsg.out`

35. Study briefly the file

36. Using the output file, complete the following table:

Type	Initial Value	Final Value Reference Solution (1)	Final Value (2)
Objective Function (Mass)	2.3452E-06	1.9748E-06	
Maximum Constraint Violation	0	0	

(1) Result from `CMDSG006_2_ref.dat`
(2) Result from your run, `CMDSG006_2_dsg.out`

Compare Part1 and Part2

37. Using the tables you filled before (part 1-step 84 and part 2-step 37), complete the following table:

Type	Part1-Part2 Initial Value	Part1 Final Value Reference Solution (1)	Part2 Final Value Reference Solution (2)	Part1 Final Value (3)	Part2 Final Value (4)
Objective Function (Mass)	2.3452E-06	1.9752E-06	1.9748E-06		
Maximum Constraint Violation	0	0	0		

(1) Result from `CMDSG006_1_ref.dat`
(2) Result from `CMDSG006_2_ref.dat`
(3) Result from your run, `CMDSG006_1_dsg.out`
(4) Result from your run, `CMDSG006_2_dsg.out`

Note: the results are similar between Repeat and Negate for the objective function. In fact this option is useful for the user to choose a type of structure.

9.7 Topometry Optimization of a Composite Plate without splitting the Angles

Introduction

The purpose of this example is to introduce the basic steps to create and solve a simple topometry optimization problem with four different composite properties (PCOMP). This example will show how to use the option: not split the design of orientation angles.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

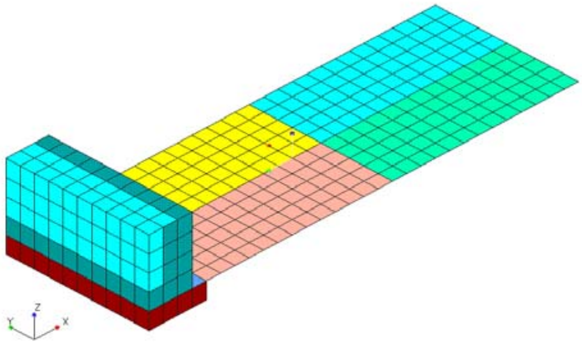
Subject to:

Failure Index $\leq$ 0.5 (all elements)

Designable region:

Six thicknesses of the composite plate
Six Angles of the composite plate

The following picture shows the structure:



Example ID

CMDSG007

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`CMDSG007_1.dat`	`Input data`	Provided: Contains the finite element mesh along with applied load and boundary conditions.
`CMDSG007_1_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `CMDSG007_1.dat`
`CMDSG007_1_dsgSSOLxx.dat`	`Input data`	Generated by a Genesis run within Design Studio. This is the input data of a solid model obtained using the thicknesses of the shells for design cycle "`xx`". The file is for visualization purposes and not intended for analysis.
`CMDSG007_1_dsgOPOSTxx.pch`	`Punch Output data`	Generated by a Genesis run within Design Studio. This is a punch output file containing the element thickness results for design cycle "`xx`".
`CMDSG007_1_dsgxx.pch`	`Punch Output data`	Generated by a Genesis run within Design Studio. This is a punch output file containing the analysis results for design cycle "`xx`".
`CMDSG007_1_ref.dat`	`Input data`	Provided: Reference file similar to the file `CMDSG007_1_dsg.dat`
`CMDSG007_2.dat`	`Input data`	Generated by Design Studio: File exported at the end of part 1. Used as starting file for part 2. Similar to `CMDSG007_1_dsg.dat` or `CMDSG007_1_ref.dat`
`CMDSG007_2_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `CMDSG007_2.dat`
`CMDSG007_2_dsgxx.pch`	`Punch Output data`	Generated by a Genesis run within Design Studio. This is a punch output file containing the analysis results for design cycle "`xx`".
`CMDSG007_2_ref.dat`	`Input data`	Provided: Reference file similar to the file `CMDSG007_2_dsg.dat`
`CMDSG007_2_dsgSSOLxx.dat`	`Input data`	Generated by a Genesis run within Design Studio. This is the input data of a solid model obtained using the thicknesses of the shells for design cycle "`xx`". The file is for visualization purposes and not intended for analysis.
`CMDSG007_2_dsgOPOSTxx.pch`	`Punch Output data`	Generated by a Genesis run within Design Studio. This is a punch output file containing the element thickness results for design cycle "`xx`".

9.7.1 Part 1

The main purpose of this part of the example is to optimize a structure with the option Normal for the topometry usage. This optimization is necessary to be able to compare the results with the Part 2 - Never Split.

When you finish this example, you should have created a file named: `CMDSG007_2.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG007_1.dat`

Defining the Design Objective

3. From the category chooser, select **Objectives**
4. Push the **New Objective** button from the Edit Menu toolbar
5. For **Name,** enter `Mass`
6. Accept the default, **Mass** response type, and **Entire Model** from the pull down menu
7. Accept the **Min** Objective Definition switch
8. Push the **Finish** button

   Verify that now there is one response in the objectives list.

Defining the Design Constraints

9. From the category chooser, select **Constraints**
10. Push the **New Constraint** button from the Edit Menu toolbar
11. For **Name,** enter `Findex`
12. Select the **Failure Index** response
13. Select **Selected Groups** option of Failure Index
14. Enter `0.5` for the **Upper Bound**
15. Push the **Next>** button
16. Select all the PCOMP groups 10, 55, 56, and 57
17. Push the **Next>** button
18. Select the static loadcase

19. Push the **Finish** button

    Verify that now there is one response in the constraint list.

Request the SSOL File

The SSOL file is a Genesis input file where the composite elements will be converted into solid elements for the purpose of better visualizing the thicknesses distribution. This file is not intended to be used for analysis or optimization.

20. From the main menu bar, select **Genesis** → **Options...**
21. Select the **File Control** tab
22. For **Shell-to-Solid File**, pick **Create(Fixed Norms)**

    The name of the SSOL file will be:
    `CMDSG007_1_dsgSSOLxx.dat` (where `xx` corresponds to the last design cycle number).
23. Push the **Apply** button

Request the OPOST (Sizing data) Post-Processing File

The OPOST file is a Genesis post-processing file that contains the optimization results, in this case the thickness of the composite elements. This file only contains the total thickness of the composite, it does not contain the individual layer thicknesses.

24. From the main menu bar, select **Genesis** → **Options**
25. Select the **File Control** tab
26. For **Element Sizing File**, choose **Create**
27. Push the **Apply** button

Optimize the Structure Using Genesis

28. From the main menu bar, select **Genesis** → **Optimize**
29. Study the **Design History** charts; when done, push the **Close** button
30. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Analysis Post-Processing Files

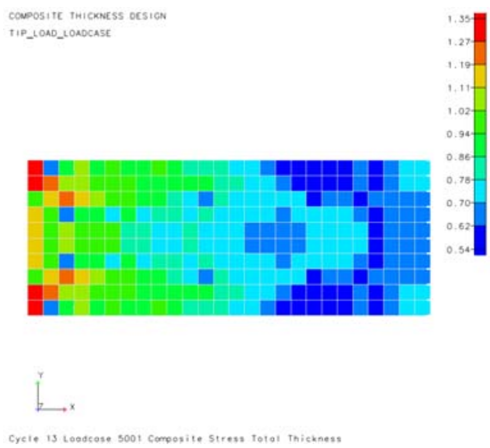
31. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
32. Select the `CMDSG007_1_dsg00.pch` file and check the **Import Similar Results for All Design Cycles** check box

33. Push the **Open** button

Post-Processing some Analysis Results

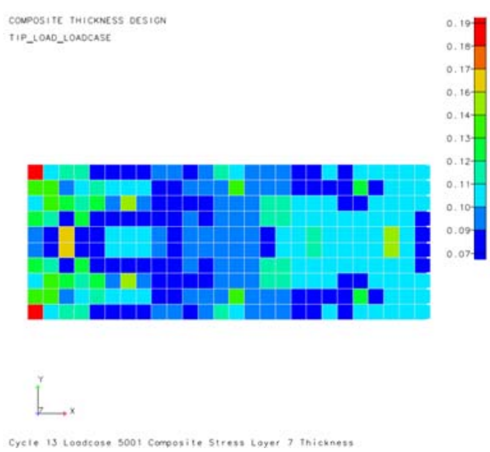
34. Select the **Post** tab
35. Push the **Deform/Mesh Color Mesh** button
36. Select the Composite Failure Index Result for the last design cycle
37. Select the Composite Stress Result for the last design cycle
38. From the category chooser in the **Color Mesh** option, change **Von Mises** to **Layer Thickness**

Now you visualize the layer thickness 1.



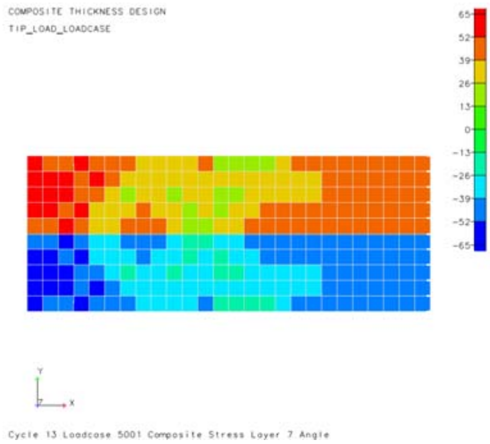
39. In the box near the category chooser, change 1 to 7.

Now you visualize the layer thickness 7.



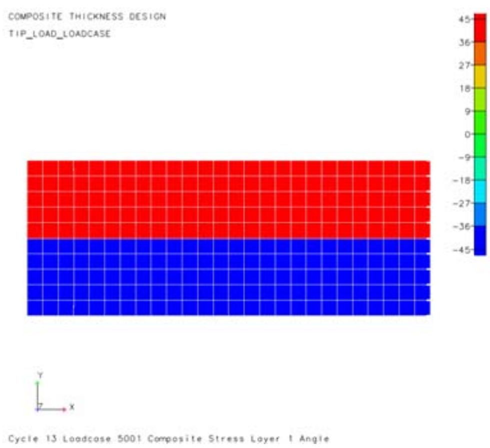
40. From the category chooser (drop-down listbox by the side of the **Options** button) in the **Color Mesh** option, change **Layer Thickness** to **Layer Angle**

    Now you visualize the layer angle 7.



41. In the box near the category chooser, change 7 to 1

    Now you visualize the layer angle 1.



Export the Input File

42. From the main menu bar, select **File** → **Export** → **Input Data...** (or Ctrl-E)
43. Enter `CMDSG007_2.dat` as Filename
44. Push the **Save** button

Quit Design Studio

45. From the main menu bar, select **File** → **Quit**
46. Push the **Don't Save** button

Study the Output File

47. In a text editor load the Genesis data file: `CMDSG007_1_dsg.out`
48. Study briefly the file
49. Using the output file, complete the following table:

Type	Initial Value	Final Value Reference Solution (1)	Final Value (2)
Objective Function (Mass)	2.3452E-06	1.9695E-06	
Maximum Constraint Violation	0	0	

(1) Result from `CMDSG007_1_ref.dat`
(2) Result from your run, `CMDSG007_1_dsg.out`

9.7.2 Part 2

The main purpose of this part of the example is to optimize a structure without splitting the design of orientation angles while doing topometry

If you do not have the `CMDSG007_2.dat` file generated in part 1, copy the file `CMDSG007_1_ref.dat to CMDSG007_2.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG007_2.dat`

Define the Option Never Split for the Angles

3. Select the **Design** tab
4. From the category chooser, select **Design Variables**
5. Select all the Discrete Design Variables: A2, A3, A4, A5, A6, A7
6. Push the **Modify Design Variable** button from the Edit menu toolbar
7. Push the **Next>** button
8. Push the **Advanced Parameters...** button
9. Check the **Topometry Usage** check box
10. For **Topometry Usage** select **Never Split** option
11. Push the **Next>** button
12. Push the **Finish** button

Optimize the Structure Using Genesis

13. From the main menu bar, select **Genesis** → **Optimize**
14. Study the **Design History** charts; when done, push the **Close** button
15. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Analysis Post-Processing Files

16. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
17. Select the `CMDSG007_1_dsg00.pch` file and check the **Import Similar Results for All Design Cycles** check box

18. Push the **Open** button

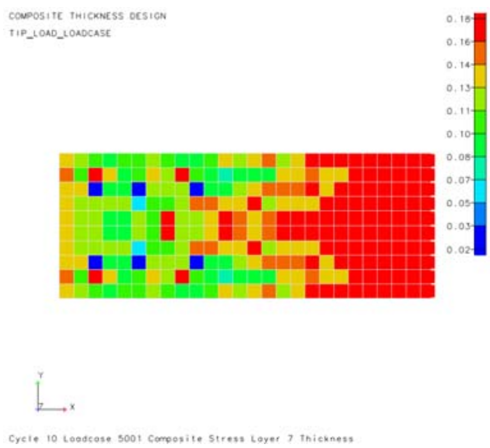
Post-Processing some Analysis Results

19. Select the **Post** tab
20. Push the **Deform/Mesh Color Mesh** button
21. Select the Composite Failure Index Result for the last design cycle
22. Select the Composite Stress Result for the last design cycle
23. From the category chooser in the **Color Mesh** option, change **Von Mises** to **Layer Thickness**

    Now you visualize the layer thickness 1.

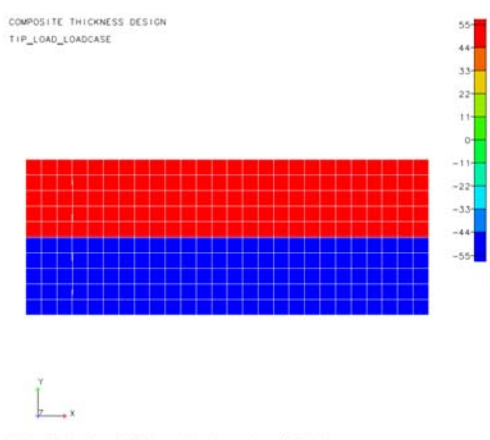
24. In the box near the category chooser, change 1 to 7

    Now you visualize the layer thickness 7.



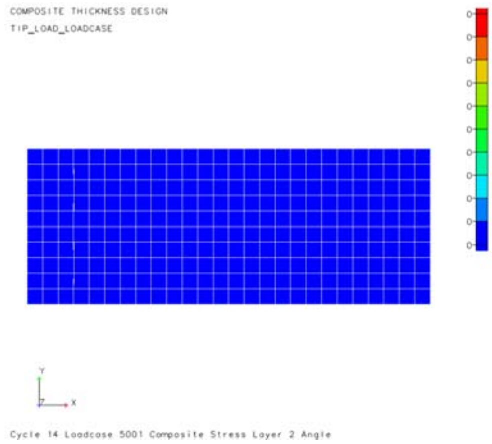
25. From the category chooser (drop-down listbox by the side of the **Options** button) in the **Color Mesh** option, change **Layer Thickness** to **Layer Angle**

    Now you visualize the layer angle 7.



26. In the box near the category chooser, change 7 to 2.

    Now you visualize the layer angle 2.



Quit Design Studio

27. From the main menu bar, select **File** → **Quit**
28. Push the **Don’t Save** button

Study the Output File

29. In a text editor load the Genesis data file: `CMDSG007_2_dsg.out`
30. Study briefly the file
31. Using the output file, complete the following table:

Type	Initial Value	Final Value Reference Solution (1)	Final Value (2)
Objective Function (Mass)	2.3452E-06	2.0353E-06	
Maximum Constraint Violation	0	0	

(1) Result from post processing: `CMDSG007_2_ref.out`
(2) Result from post processing with your run: `CMDSG007_2_dsg.out`

32. How much does the mass change?

    Reference answer: 13.2%

33. Use the post processing in Design studio to fill the following table:

Angle Layer	Original Angle Reference Solution (1)	Final Angle for elements in Master Reference Solution (1)	Final Angle for elements opposite to the Master with respect to the YZ symmetry plane Reference Solution (1)	Final Angle for elements in Master (2)	Final Angle for elements opposite to the Master with respect to the YZ symmetry plane (2)
2	0	0	0		
3	0	-5	5		
4	30	15	-15		
5	60	40	-40		
6	90	85	-85		
7	45	55	-55		

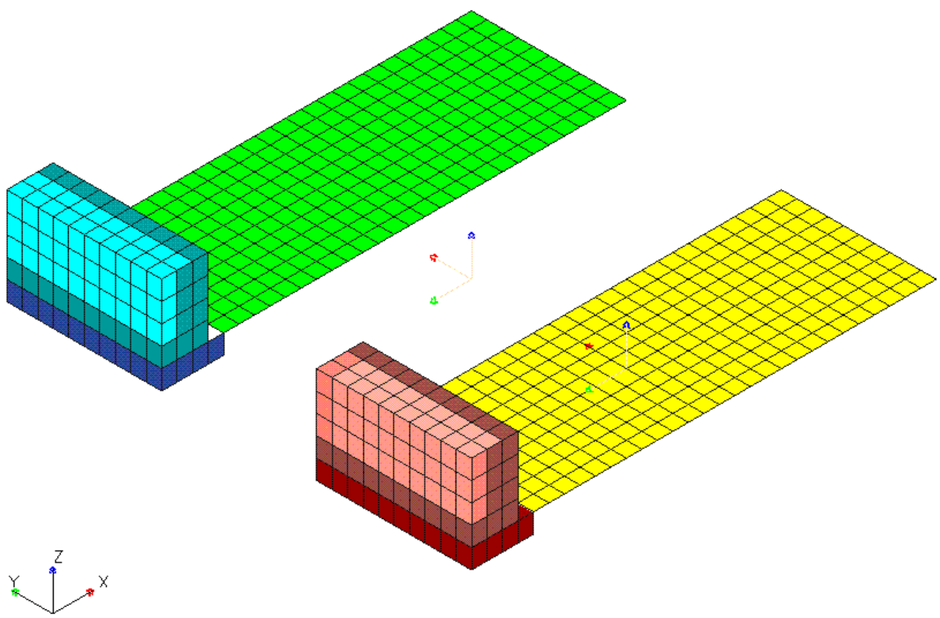
(1) Result from running: `CMDSG007_2_ref.dat`
(2) Result from post processing with your run: `CMDSG007_2_dsg.out`

9.8 Aligning the Element and Material Coordinate Systems of the Mesh of a Composite Plate

Introduction

The purpose of this example is to introduce the basic steps to fix the mesh of a composite plane. In this problem, the basic steps to create a coordinate system are reviewed. The problem is divided into two parts. The theta vectors are studied in part 1 and the orientation vectors in part 2.

The following picture shows the structure:



Example ID

CMDSG008

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`CMDSG008_1.dat`	`Input data`	Provided: Contains the finite element mesh along with applied load and boundary conditions for part 1
`CMDSG008_2.dat`	`Input data`	Provided: Contains the finite element mesh along with applied load and boundary conditions for part 2
`CMDSG008_3.dat`	`Input data`	Provided: Contains the finite element mesh along of a cylinder to study orientations

9.8.1 Part 1

The main purpose of this part of the example is to learn how to fix the mesh to align the theta vectors.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG008_1.dat`

Hide Groups

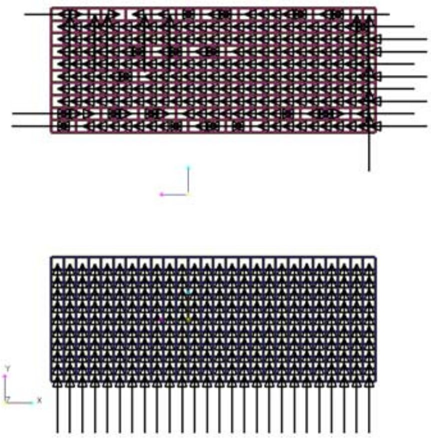
3. Select the **Display** tab
4. Push the **Show/Hide Groups** button
5. Push the **Hide All** button
6. Pick in the group list PCOMP 10 and PCOMP 55 to show

   You can also pick directly the group to hide in the Viewport. Use the **Show All**, **Hide All,** and **Invert All** buttons to easily select the group(s) to hide.
7. Push the **Up** button

Check the Theta Vectors

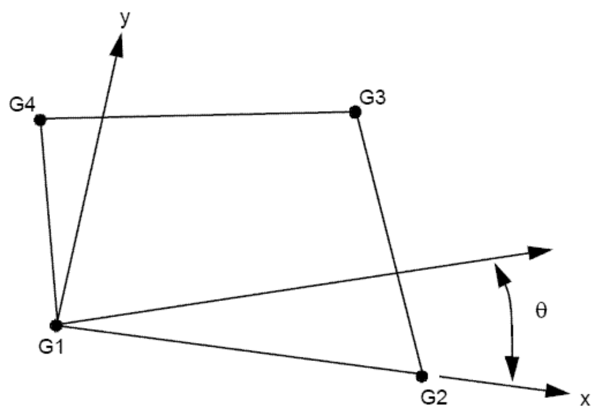
8. Select the **Analysis** tab
9. From the category chooser, select **Elements**
10. Push the **Select All** button
11. Push the **Generate Theta Vectors** button
12. In the Viewport, select the **Top** view icon

13. Study the differences between the two PCOMP



14. Which one needs to be fixed?

Reference answer: the PCOMP above in the viewport (PCOMP 55)
Note: The theta vector is useful to set up the orientation of each element. The angle THETA of each layer (or each element) is the angle between the x-axis of the coordinate system specified and the orientation of the layer (or element).
THETA is used for material property orientation specification. If Theta is a real number then it specifies the material property orientation angle in degrees. If Theta is an integer then the orientation of the material x-axis is along the projection on to the plane of the element of the x-axis of the coordinate system specified by the integer value.



Clearing the Selection

15. Right-click the Viewport, select **Clear**→ **All**

Change 2-D Elements’ Theta

16. From the **Analysis** category chooser, select **Elements**

17. Select the elements of the PCOMP 55
18. Push the **Modify Elements** button from the Edit menu toolbar
19. Select the **Change 2-D Element's Theta** radio button
20. Push **Next>**
21. Select the `Angle_Coordinate_System_Part1_1` coordinate system
22. Push the **Finish** button
23. Push the **Generate Theta Vectors** button

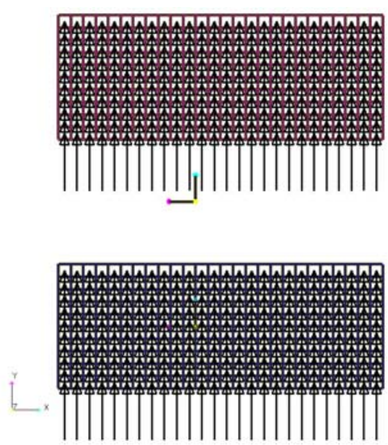
    Now you can see that all the theta vectors are parallel to the X-Axis (in red) of the coordinate system called **Angle_Coordinate_System_Part1**

Clearing the Selection

24. Right-click the Viewport, select **Clear**→ **All**

Compare the Two PCOMP's Theta Vectors

25. Select the **Analysis** tab
26. From the category chooser, select **Elements**
27. Push the **Select All** button
28. Push the **Generate Theta Vectors** button
29. Study the two PCOMP



30. Is there a difference for the theta vectors?

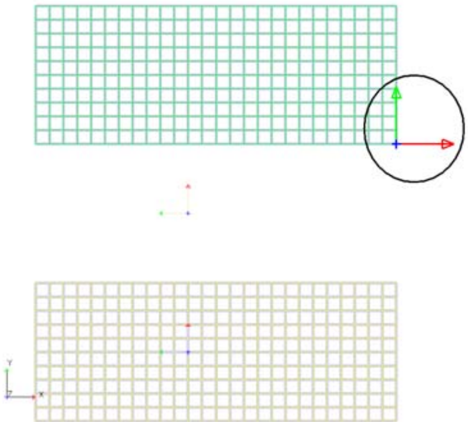
    Reference answer: No, all are parallel to the X-Axis (in red) of the coordinate system called **Angle_Coordinate_System_Part1_1**

Clearing the Selection

31. Right-click the Viewport, select **Clear→ All**

Create a New Coordinate System

32. From the **Analysis** category chooser, select **Coordinate Systems**
33. Push the **New Coordinate System** button from the Edit Menu toolbar
34. Enter `Angle_Coordinate_System_Part1_2` in the name field
35. Select the grid as shown in the following picture:



Alternatively, you can enter for **Select by grid ID** 48121

36. Push the **Finish** button

Change 2-D Elements' Theta

37. From the **Analysis** category chooser, select **Elements**
38. Select the elements of the PCOMP 55
39. Push the **Modify Elements** button from the Edit menu toolbar
40. Select the **Change 2-D Element's Theta** radio button
41. Push **Next>**
42. Select the `Angle_Coordinate_System_Part1_2` coordinate system
43. Push the **Finish** button
44. Push the **Generate Theta Vectors** button

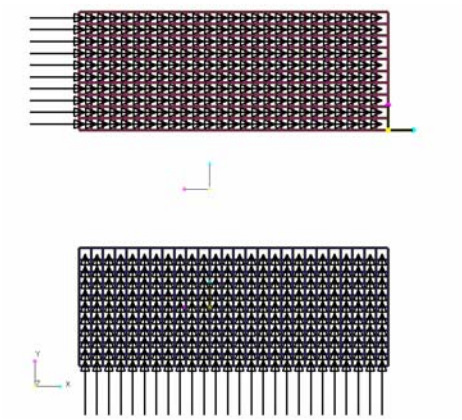
Now you can see that all the theta vectors are parallel to the X-Axis (in red) of the coordinate system called **Angle_Coordinate_System_Part1**

Clearing the Selection

45. Right-click the Viewport, select **Clear**→ **All**

Compare the Two PCOMP's Theta Vectors

46. Select the **Analysis** tab
47. From the category chooser, select **Elements**
48. Push the **Select All** button
49. Push the **Generate Theta Vectors** button
50. Study the two PCOMP



51. Is there a difference for the theta vectors?

    Reference answer: Yes, PCOMP 10's theta vectors are parallel to the X-Axis (in red) of the coordinate system called **Angle_Coordinate_System_Part1_1** and PCOMP 55's theta vectors are parallel to the X-Axis (in red) of the coordinate system called **Angle_Coordinate_System_Part1_2**

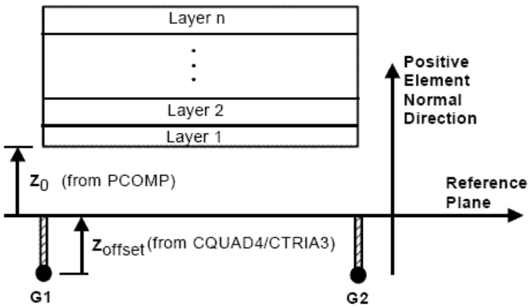
Quit Design Studio

52. From the main menu bar, select **File** → **Quit**
53. Push the **Don't Save** button

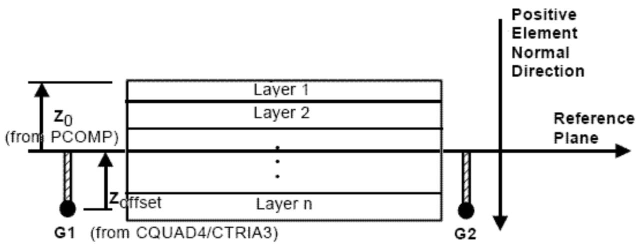
9.8.2 Part 2

The main purpose of this part of the example is to learn how to fix the mesh to align the orientation vectors.

You can define composites as shown on the following pictures:
On the picture below, the Normal direction is pointing to the top

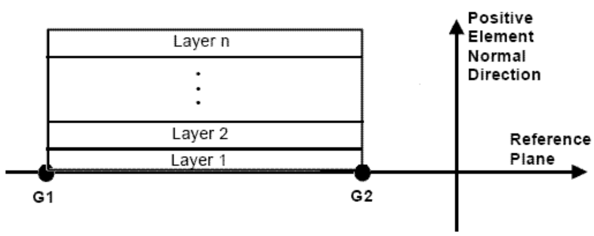


On the picture below, the Normal direction is pointing to the bottom.

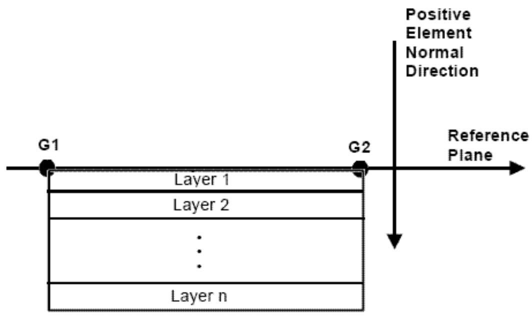


Note: Z0 Default = -1/2 the thickness of the element
Generally Z0=0 and Zoffset=0

On this picture the Normal direction is pointing to the top with Z0=0 and Zoffset=0:



On this one the Normal direction is pointing to the bottom with Z0=0 and Zoffset=0



Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `CMDSG008_2.dat`

Hide Groups

3. Select the **Display** tab
4. Push the **Show/Hide Groups** button
5. Push the **Hide All** button
6. Pick in the group list PCOMP 10 and PCOMP 55 to show

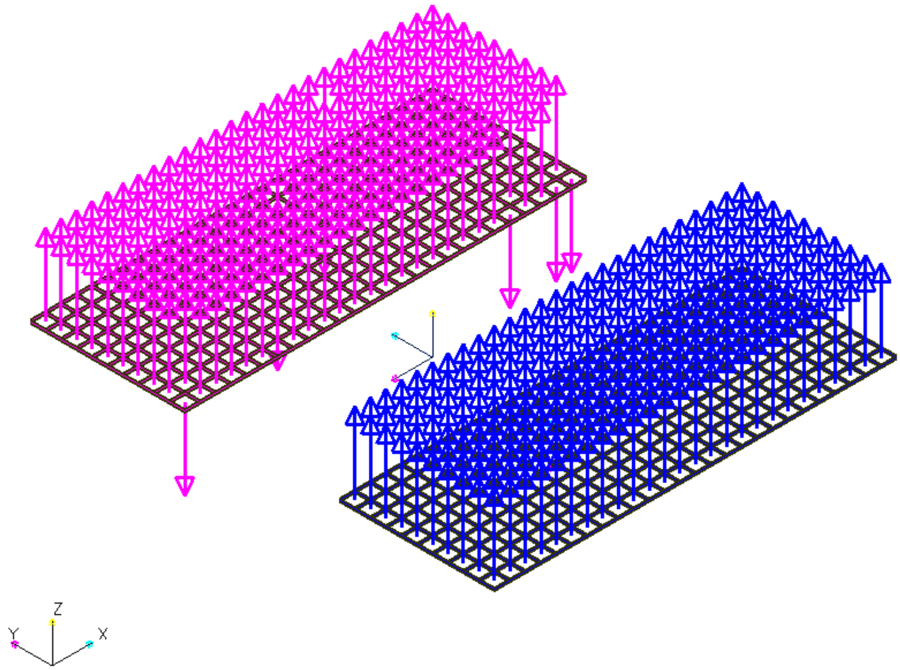
   You can also pick directly the group to hide in the Viewport. Use the **Show All**, **Hide All,** and **Invert All** buttons to easily select the group(s) to hide.

7. Push the **Up** button

Check the Orientation Vectors

8. Select the **Analysis** tab
9. From the category chooser, select **Elements**
10. Push the **Select All** button
11. Push the **Generate Orientation Vectors** button
12. In the Viewport, select the **Left** view icon
13. In the Viewport, select the **Iso-Front-Left-Top** view icon

14. Study the differences between the two PCOMP



15. Which one needs to be fixed?

 Reference answer: the PCOMP at the left in the viewport (PCOMP 55)

Clearing the Selection

16. Right-click the Viewport, select **Clear**→ **All**

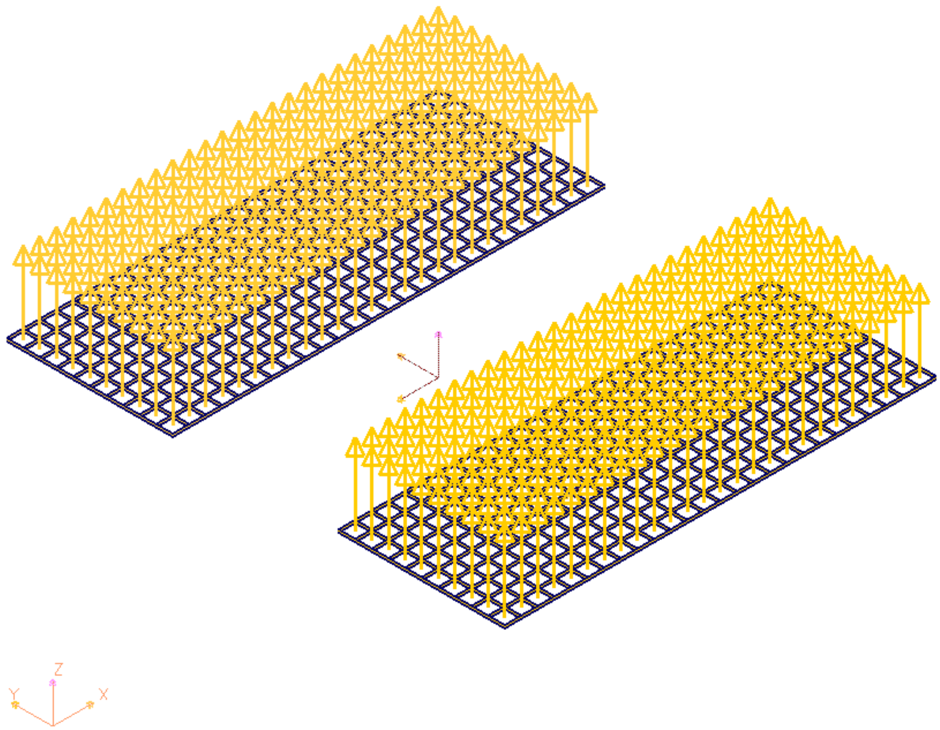
Change 2-D Elements' Orientation

17. From the **Analysis** category chooser, select **Elements**
18. Select all the elements of the PCOMP 55
19. Push the **Modify Elements** button from the Edit menu toolbar
20. Select the **Align 2-D Element's Orientation** radio button
21. Push the **Finish** button

22. Push the **Generate Orientation Vectors** button

    Now you can see that all the orientation vectors are pointing in the same direction**.**

23. Study the two PCOMP



Clearing the Selection

24. Right-click the Viewport, select **Clear**→ **All**

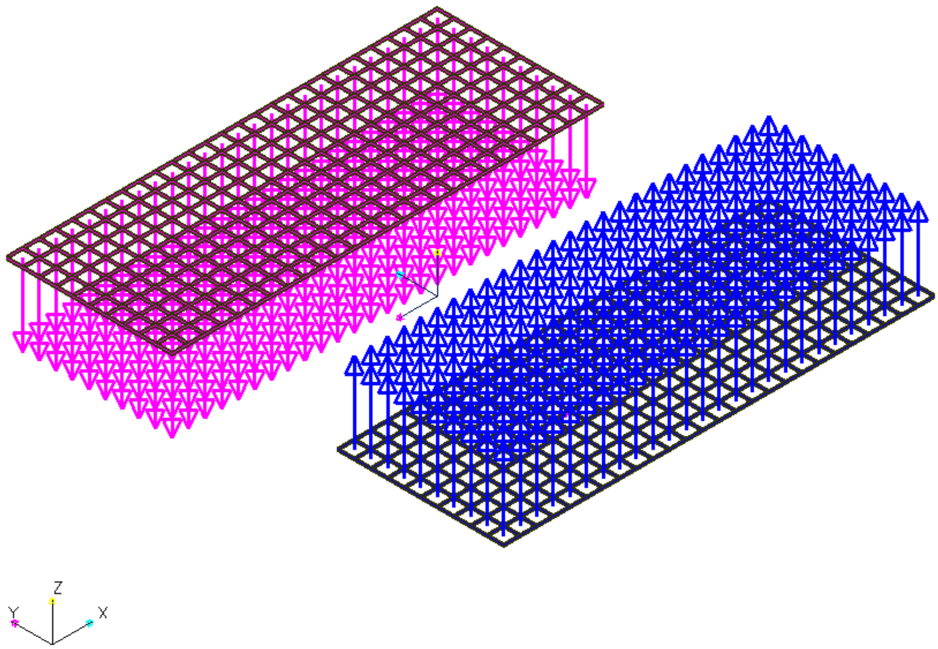
Flip 2-D Elements' Orientation

25. From the **Analysis** category chooser, select **Elements**
26. Select all the elements of the PCOMP 55
27. Push the **Modify Elements** button from the Edit menu toolbar
28. Select the **Flip 2-D Element's Orientation** radio button
29. Push the **Finish** button

30. Push the **Generate Orientation Vectors** button

    Now you can see that all the orientation vectors are pointing in the opposite direction**.**

31. Study the two PCOMP



Clearing the Selection

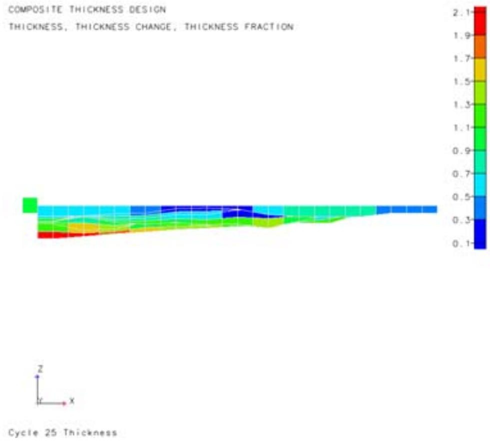
32. Right-click the Viewport, select Clear→ All

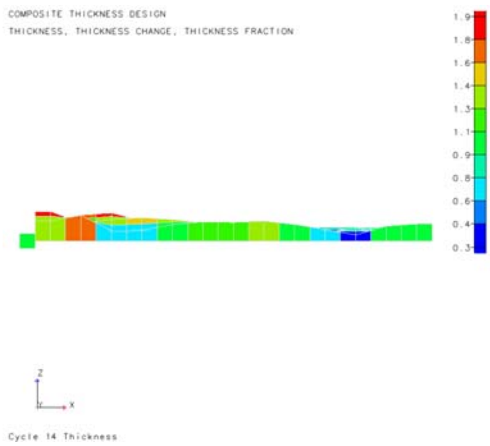
Quit Design Studio

33. From the main menu bar, select **File** → **Quit**
34. Push the **Don’t Save** button

Compare the Two Orientations for a Flat Plate

35. Study these two pictures





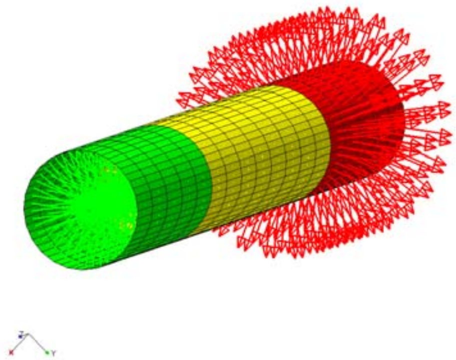
Results from the SSOL files

36. Which one has the orientation vectors pointing in the top?

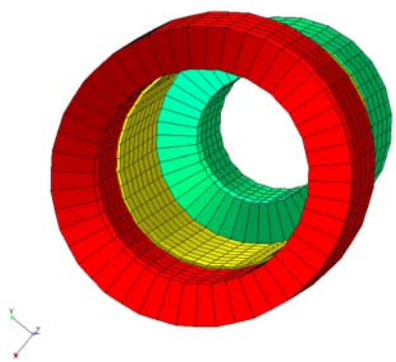
Reference answer: The second one

Compare the Two Orientations for a Cylinder

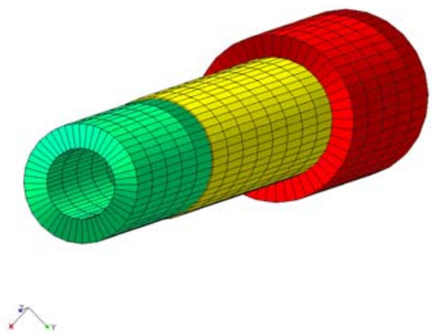
37. Study these following pictures:



This tube is divide in three parts. The orientation of the elements in green are pointing inside the tube whereas the orientation of the elements in red are pointing outside. The yellow part is a PSHELL. It is used to link the two PCOMP’s in green and red.



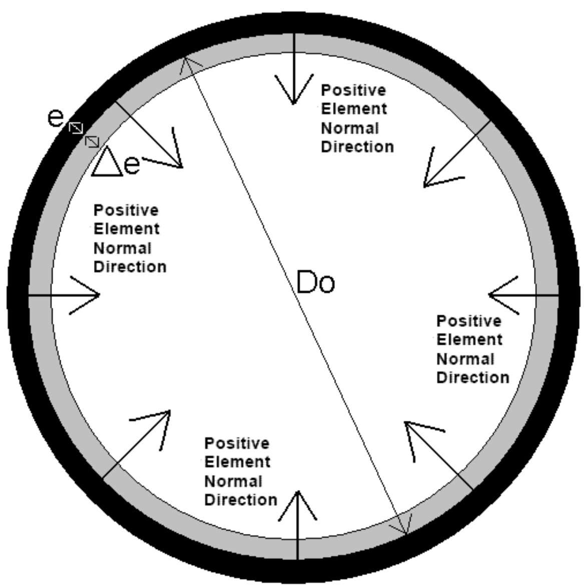
For the same pressure (loadcase), you can see the difference between the red and the green PCOMP



The orientation vector is useful to set up the direction of the extrusion.
On this example, you can use this orientation to design a tube if you have a constraint on the external diameter or on the internal diameter
Note: If you want to optimize this cylinder, open the file `CMDSG008_3.dat`

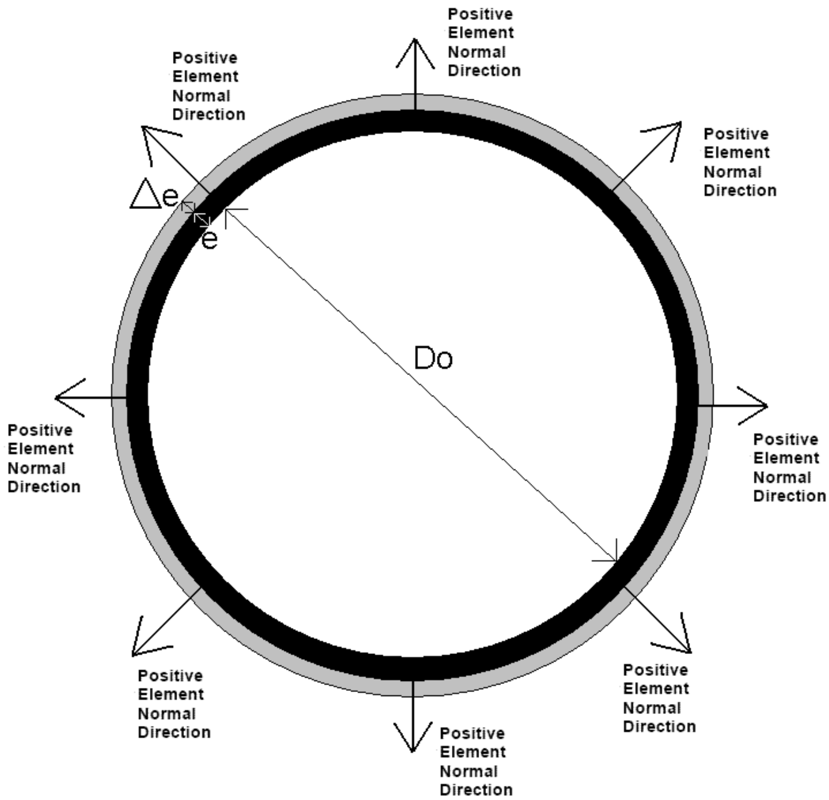
38. Which PCOMP has a constraint on the external diameter? Where are the orientation vectors pointing for the elements of this PCOMP?

Reference answer: The green PCOMP - Inside



39. Which PCOMP has a constraint on the internal diameter? Where are the orientation vectors pointing for the elements of this PCOMP?

    Reference answer: The red PCOMP - Outside



9.9 Topometry using Composite Failure Equations

Introduction

The main purpose of this exercise is to create an optimization problem by defining composite failure equations to calculate the failure index.

Objective function of the problem:

Maximize the first natural torsional frequency

Subject to:

Failure Index of each layer < 100.0

A failure equation synonymous to Von Mises for an isotropic material is used to calculate the failure index of each layer.

```
SQRT(S1**2+S2**2-S1*S2+3*S12**2)<= 100.0
```

Note: Even though VonMises criterion is applicable to isotropic materials, this was used as an example to demonstrate the definition of composite failure equations.

Designable region:

Thickness of each layer of every element in the composite plate

Analysis problem:

Static Analysis. The composite plate is simply supported at the four corners and a point load is applied at the center

Example ID

`CMDSG009`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. It is not necessary to study the list in detail at this point. The file listed will be introduced during the exercise. Later, this list can be used for verification.

File Name	Type	Description
CMDSG009.dat	Input data	Provided: This file is imported into Design Studio and contains the finite element mesh along with an static loadcase.
CMDSG009_dsg.dat	Input data	Created: Genesis input file including the composite failure definition and the topometry optimization data

CMDSG009_dsgxx.pch	Punch file	Created: Output file containing the responses at each design cycle
CMDSG009_dsgOPOSTxx.pch	Punch file	Created: Output file containing the thickness values at each design cycle
CMDSG009_ref.dat	Input data	Provided: File ready to be optimized. Same as CMDSG009_dsg.dat

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: CMDSG009.dat

Create a New Composite Failure Equation

3. From the **Analysis** category chooser, select **Composite Failure Equations**
4. Push the **New Findex** button from the Edit Menu toolbar
5. Enter VonMises for the **Name**
6. Check for the **Stress-based (FINDEX)** radio button

   One can also define a composite failure equation based on strains by selecting the **Strain-based (FINDEXN)** radio button. In this case, the quantities used in the equation correspond to the strains/strain-limits.

7. Enter F=SQRT(S1**2+S2**2-S1*S2+3*S12**2)+0.0*(XT+XC+YT+YC+S+F12)in the **Equation** field

   While defining a composite failure equation, all the quantities specified need to be used in the equation definition. That is the reason for multiplying the unnecessary quantities in the equation with a 0.0

8. Push **Finish**

Defining the failure equation on the PCOMP

9. From the **Analysis** category chooser, select **Group Properties**
10. Select the PCOMP property from the list
11. Push the **Modify Group Property** button from the Edit menu toolbar
12. Select the created failure equation (VonMises) from the drop-down listbox for the **Failure Theory**
13. Push **Finish**

Create six Independent Design Variables

14. Select the **Design** tab

15. From the **Design** category chooser, select **Design Variables**
16. Push the **New Design Variable** button from the Edit Menu toolbar to create a new design variable
17. Enter `T1` as the **Name** and select **Independent Design Variable**
18. Push **Next>**
19. Enter `0.5` for **Initial Value**, `0.01` for **Lower Bound**, and `2.0` for **Upper Bound**
20. Push the **Finish** button

    Notice that there is an asterisk in front of the independent design variable, it indicates that this design variable is not being used. The "**I**" indicates this is an **I**ndependent design variable.

21. Repeat steps 16-20 to create five more independent design variables

    Use `T2`, `T3`, `T4`, `T5`, `T6` for the names and the same lower and upper bounds

Create the Sizing Region

22. Select the **Design** tab
23. From the **Design** category chooser, select **Sizing**
24. Select PCOMP from the list
25. Push the **Modify Sizing Design** button from the Edit Menu toolbar to add design attributes
26. Select `T1` for the **Thickness** for **Layer 1**
27. Similarly, select `T2 - T6` for the **Thickness** for **Layer 2 - 6**
28. Push the **Finish** button

    Verify that the hammer icon is next to the PCOMP.
    The hammer icon indicates that the group is being size-designed.

Create the Topometry Region

29. Stay with the **Design** tab
30. From the category chooser, select **Topometry**
31. Select PCOMP
32. Push the **Modify Topometry Design** button from the Edit Menu toolbar

    One can add a coarsening to link adjacent elements thereby reducing the number of design variables.
    Symmetry conditions can also be added here to ensure symmetry in the optimal design.

33. Push the **Finish** button

    Verify that the hammer icon is next to the PCOMP.
    The hammer icon now indicates that the group is being topometry-designed.

Defining the Design Objective

34. From the category chooser, select **Objectives**
35. Push the **New Objective** button from the Edit Menu toolbar
36. Select **Mass** as the response type
37. Push the **Finish** button

    Verify that now there is one response in the objectives list.

Defining the Failure Index Constraint

38. From the category chooser, select **Constraints**
39. Push the **New Constraint** button from the Edit Menu toolbar
40. Select **Failure Index** as the response type
41. Enter `100.0` for the **Upper Bound**
42. Push **Next>**
43. Select the PCOMP from the list
44. Push **Next>**
45. Select the static loadcase from the list
46. Push the **Finish** button

Change the Maximum Number of Design Cycles

47. From the main menu bar, select **Genesis** → **Options**
48. Select the **Design Control** tab
49. For **Maximum Design Cycles:** enter `20`
50. Push the **Apply** button

Export the Input File

51. From the main menu bar, select **File** → **Export** → **Input Data...**
52. Enter `CMDSG009` and push the **Save** button

Save the Design Studio file

53. From the main menu bar, select **File** → **Save As...**
54. Enter `CMDSG009` as the Filename and push **Save (**as a Design Studio File)

Optimize the structure using Genesis

55. From the main menu bar, select **Genesis** → **Optimize**

    This optimization takes between 5 and 20 minutes, depends of your computer.

56. Study the **Genesis Console Output** window

Import the Post-Processing Files

57. From the **Genesis Console Output** window, select the **Import Post...** button
58. Select all the `CMDSG009_dsgxx.pch` files where `xx` is the design cycle number
59. Select all the `CMDSG009_dsgOPOSTxx.pch` files where `xx` is the design cycle number
60. Push the **Import** button
61. When done, select the **Close** button in the **Genesis Console Output** window

Viewing Failure Index Results

62. Select the **Post** tab
63. Push the **Deform Mesh/Color Mesh** button
64. Select the `Cycle 0 Loadcase 1 Composite Failure Index` result in the **Color Mesh**
65. Select the `Loadcase 1 Composite Failure Index` result for the last design cycle

    Notice that the failure index of values of the last design cycle for most of the elements are close to the constraint bound.

66. In the Viewport, click on an element to display its Failure Index value in the Design Studio Messages window
67. Select the `Loadcase 1 Composite Stresses` result for the last design cycle
68. In the listbox near **Options**, you can view different types of results for each layer

    Vonmises Stresses, Layer Thickness, Layer Angles etc. are some of the results

Viewing Thickness Distribution for each design cycle

69. Select the `Cycle 0 Thickness` result in the **Color Mesh**
70. Go through the thickness results for each design cycle
71. In the Viewport, click on an element to display its Thickness value in the Design Studio Messages window

    Viewport displays the total thickness distribution for selected design cycle.

Quit Design Studio

72. From the main menu bar, select **File** → **Quit**
73. Push the **Don’t Save** button

CHAPTER 10

Frequency Response Optimization

10.1 Frequency Response Optimization using Beta Method

Introduction

In this example, you will learn to optimize dynamic responses.

Dynamic responses are dependent on loading frequencies and we often need to minimize the peak (maximum) dynamic response over the applied loading frequencies. However, such a response, selected by DRESP1 entry, creates vector values rather than scalar values. Therefore, it cannot be directly optimized since the objective function needs to be a single value response. To overcome this difficulty, we introduce an artificial design variable called Beta and additional constraint equations using this Beta value. The objective function is set to minimize Beta and the scaled dynamic responses are constrained to be less than Beta. If the value of Beta is reduced, the peak (maximum) value of the dynamic response must reduce in order to satisfy the Beta constraints. This method is called the *Beta method* and is commonly used to solve the min-max (minimizing the maximum response) problem.

The following optimization problem will be created, solved and post-processed:

Minimize Beta

Subject to:

Mass$\leq$ 0.152

Beta-(Dynamic Response)/(Scale Factor)>0.0

Designable region:

Top side of the box

Example ID

FRDSG001

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description

FRDSG001.dat	Input data	Provided: Contains the finite element mesh with applied load and boundary conditions.
FRDSG001_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in FRDSG001.dat.
FRDSG001_dsg.out	Output data	Generated by a Genesis run within Design Studio. This is a Genesis output file.
FRDSG001_dsgxx.pch	Punch Output data	Generated using Genesis within Design Studio. This file contains the analysis outputs requested for a design cycle during the optimization

Start Design Studio

1. Start Design Studio
2. Load the Genesis data file: `FRDSG001.dat`

Study the Analysis Problem

3. Select the **Analysis** Tab
4. From the category chooser, select **Loadcases** to check boundary conditions and loading condition

   First loadcase shows the boundary conditions and the second loadcase shows the dynamic loading.

Analyze the structure using Genesis

5. From the main menu bar, select **Genesis** → **Single Analysis**
6. Study the **Genesis Console Output** window

Import the Post-Processing Files

7. From the **Genesis Console Output** window, select **Import Post...** button
8. Select the `FRDSG001_dsg00.pch` file
9. Push the **Import** button
10. From the **Genesis Console Output** window, select **Close** button

Post-Processing the Results (Element Force)

11. Select the **Post** tab
12. Push the **Freq. Resp. Plot** button
13. Push the **New Freq. Resp. Plot** button from the Edit Menu toolbar

14. Push **Next>**
15. Push the + button
16. Choose `Cycle 0 Loadcase 2 Displacement`
17. Push the **Next** > button
18. Choose `Grid 5002`
19. Select the **Translation 3** radio button
20. Push **Next>**

    The Frequency Response Plot will appear.
21. Study the plot and find the maximum value by highlighting around the maximum frequency.

    Maximum and minimum responses between the highlighted loading frequency range along with loading frequency values are shown in the bottom left corner in the plot. Write down the Maximum responses value.
22. Push the **Finish** button
23. Push the **Close** button
24. Push the **Up** button

Create Design Variables

25. Select the **Design** tab
26. From the Design category chooser, select **Design Variables**
27. Push the **New Design Variable** button from the Edit Menu toolbar
28. Enter `Beta` in the name and push the **Next>** button
29. Enter `1.0` for **Initial Value**, `0.0` for **Lower Bound**, and `1.0` for **Upper Bound**
30. Push the **Finish** button
31. Push the **New Design Variable** button from the Edit Menu toolbar
32. Enter name `Thickness`
33. Push **Next>**
34. Enter `0.5` for **Initial Value**, `5.0E-4` for **Lower Bound**, and `5.0` for **Upper Bound**
35. Push the **Finish** button

Create Sizing and Topometry Design Region

36. From the **Design** category chooser, select **Sizing**
37. Select PSHELL 1 from the Property list
38. Push the **Modify Sizing Design** button from the Edit Menu toolbar
39. For **Thickness**, select `Thickness` design variable from the menu
40. Push the **Finish** button
41. From the **Design** category chooser, select **Topometry**
42. Select PSHELL 1
43. Push the **Modify Topometry Design** button from the Edit Menu toolbar
44. Push the **Finish** button

Create Synthetic Response

45. From the **Design** category chooser, select **Synthetic Responses**
46. Push the **New Synthetic Response** button from the Edit Menu toolbar
47. Enter `beta_constraint` for the name
48. Push **Next>**
49. Push the **+** button

    The '+' button is used to add arguments to the equation. Notice that everytime you push the '+' button a trail is launched to add a argument. One can delete arguments from the equation by using the '**-**' button adjacent to the argument.

50. Make sure the **Fundamental Response** radio button is selected
51. Push **Next>**
52. Select the **More Response Types...** radio button as Response Type
53. Push **Next>**
54. Select the **Dynamic Displacement** radio button
55. Select **From Modal (MDISP)** option from the listbox
56. Push **Next**>
57. Enter `5002` in the Select by Grid ID field
58. Push the **Add** button

    You can also select applied dynamic loading grid in the Viewport window.
    Verify there is 1 grid selected.

59. Select the **Translation 3** radio button
60. Push **Next>**

61. Select the Loadcase `MDYN ID=2` from the list
62. Push **Next**>
63. Push the + button
64. Select the **Design Variable** radio button

    This will activate the design variable menu.

65. Select `1 Beta` from the design variable menu

    Now we have the two arguments required to state the synthetic response. You are free to change those argument names anyway you like.

66. Push **Next>**
67. For the equation, enter `F= Arg2-(Arg1/37.32)`

    Recall that 37.32 came from the maximum dynamic displacement and this is the scale value.

68. Push the **Finish** button
69. Push the **New Synthetic Response** button from the Edit Menu toolbar
70. Enter `beta_objective` for the name
71. Select the default **User Function (DRESP2)** radio button
72. Push **Next>**
73. Push the + button
74. From the design variable menu, select `Beta`
75. Push **Next>**
76. For the equation, enter `F= Arg1`
77. Push the **Finish** button

Define the Objective

78. From the **Design** category chooser, select **Objectives**
79. Push the **New Objective** button from the Edit Menu toolbar
80. Select the **Synthetic Response** radio button as Response Type
81. Push **Next>**
82. Select the Synthetic Response `beta_objective` from the list
83. Push the **Finish** button

Define the Constraints

84. From the **Design** category chooser, select **Constraints**
85. Push the **New Constraint** button from the Edit Menu toolbar
86. Select the **Synthetic Response** radio button as Response Type
87. Enter `0.0` as **Lower Bound**
88. Push **Next>**
89. Select the Synthetic Response `beta_constraint` from the list
90. Push the **Finish** button
91. Push the **New Constraint** button from the Edit Menu toolbar
92. Select the **Mass** radio button
93. Enter `0.151` as **Upper Bound**
94. Push the **Finish** button

Optimize the structure using Genesis

95. From the main menu bar, select **Genesis** → **Optimize**
96. Study the **Genesis Console Output** window

    This optimization may take some time to complete. If you wish to move on to the next step quickly, set the maximum number of design cycles to a smaller number (2 or 3).

Import the Post-Processing Files

97. From the **Genesis Console Output** window, select **Import Post...** button
98. Select all the file with names `FRDSG001_dsgxx.pch` where `xx` is the design cycle
99. Select all the file with names `FRDSG001_dsgOPOSTxx.pch` where `xx` is the design cycle
100. Push the **Import** button
101. From the **Genesis Console Output** window, select **Close** button

Post-Processing the Thickness Results

102. Select the **Post** tab
103. Push the **Deform Mesh/Color Mesh** button and display the thickness distributions
104. Push the **Up** button

Post-Processing the Frequency Response Results

105. Select the **Post** tab
106. Push the **Freq. Resp. Plot** button
107. Push the **New Freq. Resp. Plot** button from the Edit Menu toolbar
108. Push **Next>**
109. Push the + button
110. Choose `Cycle 00 Loadcase 2 Displacement`
111. Push **Next>**
112. Choose Grid `5002` from the Choose Grid/Component list
113. Select the **Translation 3** radio button
114. Push **Next>**
115. Push the + button
116. Choose `Cycle yy Loadcase 2 Displacement` (where yy corresponds to the last design cycle)
117. Push **Next>**
118. Choose Grid `5002` from the Choose Grid/Component list
119. Select the **Translation 3** radio button
120. Push **Next>**

    Study the plot and check the peak value.

121. Push the **Finish** button
122. Push the **Close** button

Quit Design Studio

123. From the main menu bar, select **File** → **Quit**
124. Push the **Don’t Save** button

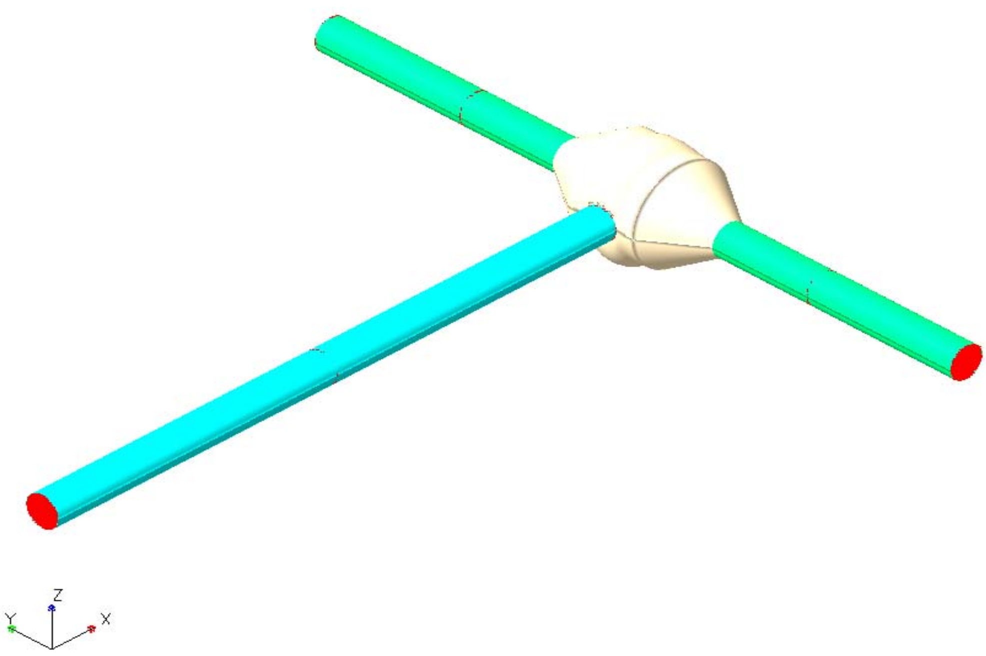
10.2 Driveline Vibrational and Modal Frequency Response Analysis

Introduction

The purpose of this example is to learn how to perform vibrational and modal frequency response analysis. In this example we will use a finite element model of an automobile driveline. The analysis is divided in two parts:

- Plot the mesh force of an element set and the modal contribution table of a grid set

- Calculate the eigenvector and the element strain energy



Analysis problem:

Natural Vibrations and Modal Frequency Response

Example ID

FRDSG002

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
FRDSG002.dat	Input data	Provided: Contains the finite element mesh of the driveline
FRDSG002_1.dat	Input data	Generated by Design Studio at the end of Part 1. This file contains all the data generated in Part 1 of the example plus the data in FRDSG002.dat
FRDSG002_1_ref.dat	Input data	Provided as a back up: This file contains all the same data as the file FRDSG002_1.dat
FRDSG002_1.dsg	DSG file	Generated by Design Studio. This is a Design Studio database file
FRDSG002_1_dsg.out	Output file	Generated using Genesis within Design Studio. This file contains FE output-results for analysis.
FRDSG002_1_dsgxy.pch	Punch file	Multiple files generated using Genesis within Design Studio. These file contains the element force for post-processing.
FRDSG002_2.dat	Input data	Generated by Design Studio at the end of Part 2. This file contains all the data generated in Part 2 of the example plus the data in FRDSG002.dat
FRDSG002_2_ref.dat	Input data	Provided as a back up: This file contains all the same data as the file FRDSG002_2.dat
FRDSG002_2.dsg	DSG file	Generated by Design Studio. This is a Design Studio database file
FRDSG002_2_dsg.out	Output file	Generated using Genesis within Design Studio. This file contains FE output results for analysis.
FRDSG002_2_dsgxy.pch	Punch file	Multiple files generated using Genesis within Design Studio. These file contains the element force for post-processing.

10.2.1 Part 1

The purpose of this part of the example is to learn how to plot the force of an element set and the modal contribution table of a grid set

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `FRDSG002.dat`

Create an element set

To plot the force in an element, you need to create an element set which contains this element.

3. Select the **Analysis** tab
4. From the category chooser, select **Element Sets**
5. Push the **New Element Set** button from the Edit Menu toolbar
6. Enter `Set 1` for the name
7. Type in Select by Element ID: `11000004`
8. Push the **Add** button

   Verify that there is the message "1 element selected".

9. Push the **Finish** button

Create a grid set

To get the modal contribution table, you need to create a grid set.

10. From the **Analysis** category chooser, select **Grid Sets**
11. Push the **New Grid Set** button from the Edit Menu toolbar
12. Enter `Set 2` for the name
13. Type in Select by Grid ID: `10000004`
14. Push the **Add** button

    Verify that there is the message "1 grid selected".

15. Push the **Finish** button

Create the Loadcase to plot the Element Force

You will learn how to set up a file in order to plot the Force in an element and get the Modal

Contribution table.

16. From the **Analysis** category chooser, select **Loadcases**
17. Select `Loadcase 1`
18. From the Edit menu toolbar, select the **Delete Loadcase** button
19. Push the **New Loadcase** button from the Edit Menu toolbar
20. Enter `Element_Force` for the name
21. Select the **Modal Frequency Response** radio button
22. Push **Next>**

    In this case, there are neither SPC nor MPC. In your model, you may have one of them or both. Don't forget to select them.

23. Push **Next>**
24. From the **Eigenvalue Method** category chooser, select `50 Method 50`
25. Push **Next>**
26. From the **Dynamic Load Set** category chooser, select `1 DLoad Set`
27. From the **Loading Frequency Set** category chooser, select `2 Frequency Set`

    Note: The step of the frequency set is 10 Hz. Usually, we use 1Hz but the computing takes more time.
    Homework: You can try to change 10 Hz into 1 Hz and see the impact on the curve and the value of the peaks.

28. From the **Modal Damping** category chooser, select `3 Modal Damping`
29. Push **Next>**
30. From the first category chooser for **Element Force**, select **Post**

    Post: element force for all elements will be output to the post processing file.
    Print: element force for all elements will be output to the output file.
    Both: element force for all elements will be output to both the output file and the post processing file.

31. From the second category chooser for **Element Force**, select `Set 1`

    All: element force for all elements will be output.
    Set 1: element force for all elements in the Set 1 will be output

32. From the first **Modal Contribution** category chooser, select **Print**
33. From the second **Modal Contribution** category chooser, select `Set 2`
34. Push the **Finish** button

    Verify that the loadcase you just created is in the list of Loadcases.

Genesis options

35. From the main menu bar, select **Genesis** → **Options...**
36. In the Output Control tab, select **Print Module Times: Both**
37. In the Output Control tab, select **Analysis Output: First & Last**
38. In the Output Control tab, select **Complex Output: Polar**
39. Push the **Apply** button

Save the Design Studio database

40. From the main menu bar, select **File** → **Save as...**
41. Enter `FRDSG002_1` for the name
42. Push the **Save** button

Export the input file

43. From the main menu bar, select **File** → **Export** → **Input data...**
44. Name the file `FRDSG002_1`
45. Push the **Save** button
46. In a text editor, open both files `FRDSG002.dat` and `FRDSG002_1.dat` and compare them.

Analyze the structure using Genesis

47. From the main menu bar, select **Genesis** → **Single Analysis**
48. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

49. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
50. Select the `FRDSG002_1_dsg00.pch` file
51. Push the **Open** button

Post-Processing the Results (Element Force)

52. Select the **Post** tab
53. Push the **Freq. Resp. Plot** button
54. Push the **New Freq. Resp. Plot** button from the Edit Menu toolbar

55. Push the **Magnitude + Phase** radio button
56. Push **Next>**
57. Push the + button
58. Choose `Cycle 0 Loadcase 2 Elas Force`
59. Push **Next>**
60. Choose `Element 11000004`
61. Push **Next>**

    The Frequency Response Plot will appear.

62. Right click on the graph
63. Push the **Save Image...** button
64. Enter the name `FRDSG002_1`
65. Push the **Save** button
66. Push the **Y Axis Options...** button
67. Push the **Log Scale** radio button
68. Push **Next>**
69. Find the maximum value of the 3 highest peaks by highlighting the plot around the peaks.

    Maximum and minimum responses between the highlighted loading frequency range along with loading frequency values are shown in the bottom left corner in the plot. Write down the Maximum responses values.

Frequency (Hz)	Magnitude Reference Solution (1)	Magnitude Solution (2)
340	7.376	
440	16.17	
840	7.251	

(1) Result from plotting `FRDSG002_2_ref_dsg00.pch`
(2) Result from plotting `FRDSG002_2_dsg00.pch`

70. Push the **Finish** button
71. Push the **Close** button in the Frequency Response window
72. Push the **Up** button

Quit Design Studio

73. From the main menu bar, select **File** → **Quit**
74. Push the **Don't Save** button

Modal Contribution Table

75. In a text editor, open the file `FRDSG002_2_dsg.out`
76. Search for `Modal Contributions`
77. The results are given in the following format

```
 TOTAL SOLUTION OF GRID 10000004, COMPONENT 1, LOADING FREQ. NUM.:        1


          REL MC          ID       EXC. FREQ       MAGNITUDE          PHASE

      1.0000E+00           0      1.0000E-03      6.0247E-09      1.8000E+02


 MODAL CONTRIBUTIONS,     MAGNITUDE > 1.00E-03* MAGNITUDE(TOTAL SOLUTION)


          REL MC          ID       NAT. FREQ       MAGNITUDE          PHASE


      1.2437E+01          12      4.3791E+02      7.4931E-08      1.8000E+02
     -1.1147E+01           3      2.2631E+02      6.7160E-08      3.6000E+02
      5.6915E+00          30      8.3763E+02      3.4290E-08      1.8000E+02
     -4.1993E+00          35      8.8657E+02      2.5300E-08      3.6000E+02
      4.1001E+00          17      6.4189E+02      2.4702E-08      1.8000E+02
     -3.9691E+00          45      1.6799E+03      2.3913E-08      0.0000E+00
      2.2408E+00          15      5.3714E+02      1.3500E-08      1.8000E+02
     -1.9507E+00          44      9.3679E+02      1.1753E-08      3.6000E+02
     -1.9279E+00           6      3.4307E+02      1.1615E-08      3.6000E+02
     -3.5659E-01          29      8.0691E+02      2.1483E-09      3.6000E+02
      8.4813E-02          13      4.5535E+02      5.1098E-10      1.8000E+02
      4.3982E-03           2      8.5667E+01      2.6498E-11      1.8000E+02
     -4.1498E-03          16      5.8205E+02      2.5002E-11      3.6000E+02
     -1.9495E-03           4      2.6009E+02      1.1745E-11      3.6000E+02
     -1.9338E-03          24      7.7355E+02      1.1651E-11      3.6000E+02
```

78. Study the modal contribution tables for the frequencies you wrote previously
79. Quit the file

10.2.2 Part 2

The purpose of this example is to introduce the basic steps to perform a modal analysis of a structure by calculating the eigenvector and the element strain energy. This example will show how to visualize the displacement and the element strain energy.

Analysis problem:

SMS method

Start Design Studio

80. Start Design Studio
81. Import the Genesis data file: `FRDSG002.dat`

Checking the Eigenvalue Methods (SMS)

82. Select the **Analysis** tab
83. From the category chooser, select **Eigenvalue Methods**
84. Select the existing Eigenvalue Method
85. Push the **Modify Eigenvalue Method** button from the Edit Menu toolbar

    The data you can see means that the calculated frequencies will be searched for all modes less than or equal to 1000
86. Push the **Cancel** button

Create the Loadcase to visualize the Displacement and the Element Strain Energy

87. From the category chooser, select **Loadcases**
88. Select `Loadcase 1`
89. From the Edit menu toolbar, select the **Delete Loadcase** button
90. Push the **New Loadcase** button from the Edit Menu toolbar
91. Enter `Disp_ESE` for the name
92. Select the **Normal Modes** radio button
93. Push **Next>**

    In this case, there are neither SPC nor MPC. In your model, you may have one of them or both. Don't forget to select them.

94. Push **Next>**
95. From the **Eigenvalue Method** category chooser, select `50 Method 50`
96. Push **Next>**
97. From the first category chooser for **Displacement**, select **Post**

    Post: displacements for all points will be output to the post processing file.
    Print: displacements for all points will be output to the output file.
    Both: displacements for all points will be output to both the output file and the post processing file.

98. From the second category chooser for **Displacement**, select **All**

    All: displacements for all points will be output.

99. Select **Post** and **All** from the first and second category chooser respectively for **Element Strain Energy**
100. Push the **Finish** button

    Note: Verify that the loadcase you just created is in the list of Loadcases.

Genesis options

101. From the main menu bar, select **Genesis** → **Options...**
102. In the Output Control tab, select **Print Module Times: Both**
103. In the Output Control tab, select **Analysis Output: First & Last**
104. Push the **Apply** button

Save the Design Studio database

105. From the main menu bar, select **File** → **Save as...**
106. Enter `FRDSG002_2` for the name
107. Push the **Save** button

Export the input file

108. From the main menu bar, select **File** → **Export** → **Input data...**
109. Enter `FRDSG002_2` for the name
110. Push the **Save** button
111. In a text editor, open both files `FRDSG002.dat` and `FRDSG002_2.dat` and compare them.

Analyze the structure using Genesis

112. From the main menu bar, select **Genesis** → **Single Analysis**
113. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

114. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
115. Select the `FRDSG002_2_dsg00.pch` file
116. Push the **Open** button

Post-Processing the Results (Displacement and Element Strain Energy Results)

117. Select the **Post** tab
118. Push the **Deform Mesh/Color Mesh** button
119. Push the **Oscillate** button
120. Select an Eigenvector from the **Deform Mesh** list
121. Select an ESE Result from the **Color Mesh** list
122. Push the **Ramp** button
123. Push the **Filled Contour** button
124. Select an Eigenvector from the **Color Mesh** list
125. Push the **Filled Elements** button
126. Select an Element Strain Energy result
127. From the category chooser, change **Element Strain Energy** into **ESE Percent**

Finding the Top 10 Displacements or Element Strain Energy:

Compare the maximum displacement or element strain energy values of any eigenvector

128. Right click the viewport and select **Top 10**

The 10 highest values are printed in the Design Studio Messages window.

129. Right click the viewport and select **Clear** → **All** to remove the selection

Post-Processing the Results in view of design

130. Study each mode ID you have written down in the previous part, write down the elements involved by searching for the Top 10.

```
Mode 12 (ESE)
CELAS1 11000004, 36.26625
CBUSH 11000003, 2.781401
CBUSH 11000006, 1.232711
CBUSH 11000007, 0.9792632
CBUSH 11000001, 0.2748655
CBUSH 11000002, 0.2126569
CQUAD4 100079, 0.1255991
CQUAD4 100083, 0.1253719
CQUAD4 101644, 0.09823395
CQUAD4 101643, 0.0957559
Mode 6 (ESE)
CELAS1 11000004, 16.86649
CBUSH 11000003, 2.999446
CBUSH 11000007, 0.2084633
CBUSH 11000006, 0.1666855
CQUAD4 16852, 0.1283768
CQUAD4 21733, 0.1278341
CQUAD4 17345, 0.1278312
CQUAD4 21240, 0.1274719
CQUAD4 21223, 0.1273965
CQUAD4 16835, 0.1268247
Mode 30 (ESE)
CELAS1 11000004, 15.12283
CBUSH 11000003, 0.2037344
CQUAD4 6574, 0.200969
CQUAD4 10334, 0.1986826
CQUAD4 8455, 0.1985766
CQUAD4 6575, 0.1980434
CQUAD4 8454, 0.1978911
CQUAD4 12215, 0.1975076
CQUAD4 10335, 0.1968341
```

```
CQUAD4 12214, 0.1958888
```

Quit Design Studio

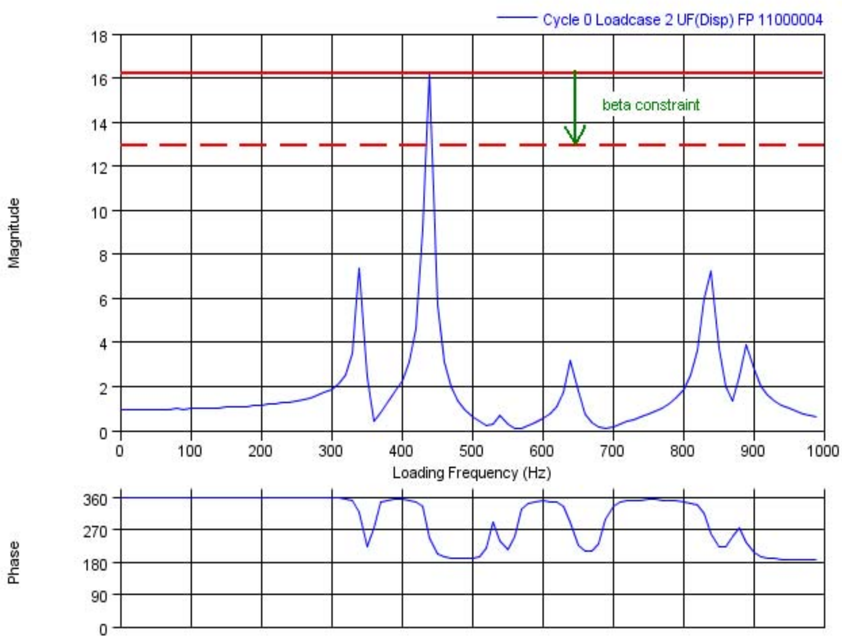
131. From the main menu bar, select **File** → **Quit**
132. Push the **Don’t Save** button

10.3 Driveline design using the beta method I

Introduction

In this example, you will learn to optimize dynamic responses using the beta method.

Dynamic responses are dependent on loading frequencies and we often need to minimize the peak (maximum) dynamic response over the applied loading frequencies. However, such a response, selected by DRESP1 entry, creates vector values rather than scalar values. Therefore, it cannot be directly optimized since the objective function needs to be a single value response. To overcome this difficulty, we introduce an artificial design variable called *beta* and additional constraint equations using the *beta* value. The objective function is set to minimize *beta* and the scaled dynamic responses are constrained to be less than *beta*. If the value of beta is reduced, the peak (maximum) value of the dynamic response must be reduced in order to satisfy the beta constraints. This method is called the *beta method* and is commonly used to solve the min-max (minimizing at the maximum response) problem.



In this example, you will also learn how to perform sizing.

The following optimization problem will be created, solved and post-processed:

Minimize *beta*

Subject to:

(Dynamic Response)/(Scale Factor) - *beta* < 0.0

Designable region:

The CELAS element

The CBUSH elements

The thickness of the PSHELL groups

Example ID

FRDSG003

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
FRDSG003.dat	Input data	Provided: Contains the finite element mesh of the driveline
FRDSG003_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in FRDSG003.dat
FRDSG003_ref.dat	Input data	Provided as a back up: This file contains the same data as the file FRDSG003_dsg.dat
FRDSG003.dsg	DSG file	Generated by Design Studio. This is the Design Studio database file
FRDSG003_dsg.out	Output file	Generated using Genesis within Design Studio. This file contains all the output data created while running genesis
FRDSG003_dsg00.pch	Punch file	File generated using Genesis within Design Studio. These file contains the element force for post-processing for the initial design.
FRDSG003_ref_dsg00.pch	Punch file	Provided as a back up: File generated using Genesis within Design Studio. These file contains the element force for post-processing.
FRDSG003_dsgxx.pch	Punch file	File generated using Genesis within Design Studio. These file contains the element force for post-processing for the final design, xx corresponds to the final design cycle number.

FRDSG003_ref_dsg07.pch	Punch file	Provided as a back up: File generated using Genesis within Design Studio. These file contains the element force for post-processing for the final design.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `FRDSG003.dat`

Create an element set

To plot the force in an element, you need to create an element set which contains this element.

3. Select the **Analysis** tab
4. From the category chooser, select **Element Sets**
5. Push the **New Element Set** button from the Edit Menu toolbar
6. Enter `Set 1` for the name
7. Type in Select by Element ID: `11000004`
8. Push the **Add** button

   Verify that there is the message "1 element selected".

9. Push the **Finish** button
10. Right click on the Viewport window and select **Clear** → **All**

Create the Loadcase to plot the Element Force

You will learn how to set up a file in order to plot the Force in an element.

11. Select the **Analysis** tab
12. From the category chooser, select **Loadcases**
13. Select `Loadcase 1`
14. From the Edit menu toolbar, select the **Delete Loadcase** button
15. Push the **New Loadcase** button from the Edit Menu toolbar
16. Enter `Element_Force` for the name
17. Select the **Modal Frequency Response** radio button
18. Push **Next>**

    In this case, there are neither SPC nor MPC. In your model, you may have one of them or both. Don't forget to select them.

19. Push **Next>**

20. From the **Eigenvalue Method** category chooser, select `50 Method 50`
21. Push **Next>**
22. From the **Dynamic Load Set** category chooser, select `1 DLoad Set`
23. From the **Loading Frequency Set** category chooser, select `2 Frequency Set`
24. From the **Modal Damping** category chooser, select `3 Modal Damping`
25. Push **Next>**
26. From the first **Element Force** category chooser, select **Post**

    Post: element force for all elements will be output to the post processing file.
    Print: element force for all elements will be output to the output file.
    Both: element force for all elements will be output to both the output file and the post processing file.

27. From the second **Element Force** category chooser, select `Set 1`

    All: element force for all elements will be output.
    Set 1: element force for all elements in the Set 1 will be output

28. Push the **Finish** button

    Verify that the loadcase you just created is in the list of Loadcases.

Designing the PSHELL groups

29. Select the **Design** tab
30. From the category chooser, select **Sizing**
31. Select PSHELL 3 and PSHELL 4

    When selecting from the list, to select more than one group, hold the **Ctrl** key while selecting the second group. When selecting from the viewport, you do not need to hold the **Ctrl** key.

32. Push the **Modify Sizing Design** button from the Edit Menu toolbar
33. Check the **Thickness** checkbox

    You need to create a design variable and then come back where you are. Do not push **Finish** or **Cancel** at this point.

34. From the **Design** category chooser, select **Design Variables**
35. Push the **New Design Variable** button from the Edit Menu toolbar
36. Enter `Tube` as the Name and accept the default selection of **Independent Design Variable**
37. Push **Next>**
38. Enter `1.0` for **Initial Value**, `0.5` for **Lower Bound** and `1.5` for **Upper Bound**
39. Push the **Finish** button

40. From the **Design** category chooser, select **Sizing**

    You are back where you left off before design variable creation

41. Select `1 Tube` in the **Thickness** category chooser
42. Push the **Finish** button

    Now, you see hammers icon next to the groups PSHELL 3 and PSHELL 4 in the list. This indicates that they are being size-designed.

43. Select PSHELL 6
44. Push the **Modify Sizing Design** button from the Edit Menu toolbar

    You need to create a design variable and then come back where you are. Do not push **Finish** or **Cancel** at this point.

45. From the **Design** category chooser, select **Design Variables**
46. Push the **New Design Variable** button from the Edit Menu toolbar
47. Enter `Box` as the Name and select **Independent Design Variable**
48. Push **Next>**
49. Enter `2.0` for **Initial Value**, `1.5` for **Lower Bound** and `2.5` for **Upper Bound**
50. Push the **Finish** button
51. From the **Design** category chooser, select **Sizing**

    You will be back where you left off before design variable creation

52. Select `2 Box` in the **Thickness** category chooser
53. Push the **Finish** button

    Now, you see hammers icon next to the groups PSHELL 6 in the list. This indicates that they are being size-designed.

Designing the PELAS group

54. Select PELAS 104
55. Push the **Modify Sizing Design** button from the Edit Menu toolbar

    You need to create a design variables and then come back where you are. Do not push **Finish** or **Cancel** at this point.

56. From the **Design** category chooser, select **Design Variables**
57. Push the **New Design Variable** button from the Edit Menu toolbar
58. Enter `DPELAS104` as the Name and select **Independent Design Variable**
59. Push **Next>**
60. Enter `1.0` for **Initial Value**, `0.6` for **Lower Bound** and `1.4` for **Upper Bound**

61. Push the **Finish** button
62. Push the **New Design Variable** button from the Edit Menu toolbar
63. Enter `PELAS104` as the Name and select **Synthetic Design Variable**
64. Push **Next>**
65. From the **Master Variable** Category chooser, select `DPELAS104`
66. Enter `35000.0` as coefficient
67. Push the **Finish** button
68. From the **Design** Category chooser, select **Sizing**

    You are back where you left off before design variable creation
69. From the **Stiffness** Category chooser, Select `PELAS104`
70. Push the **Finish** button

Designing the PBUSH groups

Some of the stiffness quantities on each of the PBUSH groups are designed. Below is a table with the designable quantities in each of the PBUSH groups along with the design variables used to control the quantities.

Property/Group	Designable Quantities	Design Variables	
		Synthetic	Independent
PBUSH 101	K1 K2 K3	PB101K1 PB101K2 PB101K3	DPB101K1 DPB101K2 DPB101K3
PBUSH 103	K3 K5 K6	PB103K3 PB103K5 PB103K6	DPB103K3 DPB103K5 DPB103K6
PBUSH 105	K1, K2, K3	PB105K123	DPB105K123
PBUSH 106	K1, K2,K4, K5, K6 K3	PB106K12456 PB106K3	DPB101K12456 DPB101K3

For scaling reasons, it is better to normalize the design variables. That is the reason why synthetic variables are used to determine the stiffness values from each of the PBUSH groups. Each of the independent variables are defined with a 30% variation around 1.0.

First all the Independent design variables are created. These independent variables are then used to create synthetic design variables. Finally the synthetic design variables are associated with the design of the PBUSH groups.

Creating the Independent Design Variables:

71. From the **Design** category chooser, select **Design Variables**
72. Push the **New Design Variable** button from the Edit Menu toolbar
73. Enter `DPB101K1` as the **Name**
74. Make sure the **Independent Design Variable** option is selected
75. Push **Next>**
76. Enter `1.0` for **Initial Value**, `0.7` for **Lower Bound** and `1.3` for **Upper Bound**

    You could also enter `15000.0` for **Initial Value**, `0.7*15000.0` for **Lower Bound** and `1.3*15000.0` for **Upper Bound**. However, for scaling reasons, it is better to normalize the design variables. If you create the design variables using the original values, the **Synthetic Design Variable** `PB101K1` need not be created
77. Push the **Finish** button
78. Repeat the steps 72 to 77 with the data given in the following table to design to define all the independent design variables

    The independent design variables in the first row has already been defined in steps 72 to 76.

Variable Name	Initial value	Lower Bound	Upper Bound
DPB101K1	1.0	0.7	1.3
DPB101K2	1.0	0.7	1.3
DPB101K3	1.0	0.7	1.3
DPB103K3	1.0	0.7	1.3
DPB103K5	1.0	0.7	1.3
DPB103K6	1.0	0.7	1.3
DPB105K123	1.0	0.7	1.3
DPB106K12456	1.0	0.7	1.3
DPB106K3	1.0	0.7	1.3

Creating the Synthetic Design Variables:

79. From the **Design** category chooser, select **Design Variables**
80. Push the **New Design Variable** button from the Edit Menu toolbar
81. Enter `PB101K1` as the **Name**
82. Select **Synthetic Design Variable** option

83. Push **Next>**
84. From the **Master Variable** Category chooser, select `DPB101K1`
85. Enter `15000.0` as coefficient
86. Push the **Finish** button
87. Repeat the steps 80 to 86 with the data given in the following table to create the synthetic design variables

    The synthetic design variables in the first row has already been defined.

Variable Name	Coefficient	Master Variable
PB101K1	15000	DPB101K1
PB101K2	50000	DPB101K2
PB101K3	40000	DPB101K3
PB103K3	30000	DPB103K3
PB103K5	50000	DPB103K5
PB103K6	75000	DPB103K6
PB105K123	1.0E+10	DPB105K123
PB106K12456	2500	DPB106K12456
PB106K3	1500	DPB106K3

Assigning design variables to PBUSH groups:

88. From the **Design** Category chooser, select **Sizing**
89. Select PBUSH 101
90. Push the **Modify Sizing Design** button from the Edit Menu toolbar
91. From the **K1** Category chooser, select `PB101K1`
92. From the **K2** Category chooser, select `PB101K2`
93. From the **K3** Category chooser, select `PB101K3`
94. Push the **Finish** button
95. Repeat the steps 89 to 94 with the data given in the following table to design the PBUSH groups

    The designable properties of PBUSH101 (first row in the table below) has already been defined in the earlier steps.

Group	Bushing Property Variable to be modified	Associated Variable
PBUSH101	K1	PB101K1
	K2	PB101K2
	K3	PB101K3
PBUSH103	K3	PB103K3
	K5	PB103K5
	K6	PB103K6
PBUSH105	K1	PB105K123
	K2	PB105K123
	K3	PB105K123
PBUSH106	K1	PB106K12456
	K2	PB106K12456
	K3	PB106K3
	K4	PB106K12456
	K5	PB106K12456
	K6	PB106K12456

Defining the beta method constraint equation

The *beta method* equation is: (Dynamic Response)/(Scale Factor) - *beta* < 0.0
The Scale Factor is calculated during the modal analysis process.
The Dynamic Response is calculated by Genesis for each design cycle.
The *beta* design variable is created.

96. From the **Design** category chooser, select **Synthetic Responses**
97. Push the **New Synthetic Response** button from the Edit Menu toolbar
98. Enter `beta constraint` for the **Name**
99. Make sure the **Type** selected is **User Function (DRESP2)**
100. Push **Next>**
101. Push the **+** button
102. Make sure the **Fundamental Response** radio button is selected

103. Push **Next>**
104. Select **More Response Types...** from the Response Type radio button
105. Push **Next>**
106. Select **Dynamic Force** from the **Response Type** radio button and select **Selected Elements** from the category chooser
107. Select **Magnitude** from the **Select Dynamic Component**
108. Push **Next>**
109. Enter `11000004` in **Select by Element ID**
110. Push the **Add** button

     Verify that there is 1 element selected.

111. Push **Next>**
112. Push **Next>**
113. Select the Loadcase `Element_Force`
114. Push **Next>**

     You need to create a design variable (*beta*) and then come back where you are. Do not push **Finish** or **Cancel** at this point.

115. From the **Design** category chooser, select **Design Variables**
116. Push the **New Design Variable** button from the Edit Menu toolbar
117. Enter `beta` as the Name and select **Independent Design Variable**
118. Push **Next>**
119. Enter `1.0` for **Initial Value**, `0.0` for **Lower Bound** and `1.0` for **Upper Bound**
120. Push the **Finish** button
121. From the **Design** category chooser, select **Synthetic Responses**

     You are back where you left off before design variable creation.

122. Push the **+** button
123. Select **Design Variable** from the **Response Type** radio button and select `beta`
124. Push **Next>**

     Now we have the two arguments required to state the synthetic response.

125. For the equation, enter `F = (Arg1/16.17) - Arg2`

     You are free to change those argument names anyway you like.
     Scale Factor = 16.17 (value found during the analysis).

126. Push the **Finish** button

Verify that the synthetic response you just created is listed

Defining the Design Constraints

127. From the **Design** category chooser, select **Constraints**
128. Push the **New Constraint** button from the Edit Menu toolbar
129. Select **Synthetic Response** as Response Type.
130. Enter `0.0` for **Upper bound**
131. Push the **Next>** button
132. Select the `beta constraint` Synthetic Response
133. Push the **Next>** button
134. Push the **Finish** button

Verify that the constraint you just created is listed

Defining the beta method objective equation

135. From the **Design** category chooser, select **Synthetic Responses**
136. Push the **New Synthetic Response** button from the Edit Menu toolbar
137. Enter `beta objective` in the **Name**
138. Make sure the **Type** selected is **User Function (DRESP2)**
139. Push the **Next>** button
140. Push the + button
141. Push the **Design Variable** radio button and select `beta`
142. Push the **Next** > button
143. For the equation, enter `F = 16.17*Arg1`

Scale Factor = 16.17

144. Push the **Finish** button

Verify that the synthetic response you just created is listed

Defining the Design Objective

145. From the **Design** category chooser, select **Objectives**
146. Push the **New Objective** button from the Edit Menu toolbar

147. Select **Synthetic Response** as Response Type.
148. Select **Min** as Objective definition
149. Push the **Next>** button
150. Select the `beta objective` synthetic response
151. Push the **Next>** button
152. Push the **Finish** button

    Verify that the objective you just created is listed

Clearing the Selection

153. Right click the Viewport, Select **Clear**, **All**

Setup the Genesis Options

154. From the main menu bar, select **Genesis** → **Options...**
155. In the **Output Control** tab, select **Print Module Times: Both**
156. In the **Output Control** tab, select **Analysis Output: First & Last**
157. In the **Output Control** tab, select **Design Output: All Cycles**
158. Push the **Apply** button

Save the Design Studio database

159. From the main menu bar, select **File** → **Save as...**
160. Name the file `FRDSG003`
161. Push the **Save** button

Optimize the structure using Genesis

162. From the main menu bar, select **Genesis** → **Optimize**
163. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files (Element Force)

164. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
165. Select the `FRDSG003_dsg00.pch` file

166. Check the **Import Similar Results for All Design Cycles** check box

   This will allow all similar files, e.g., `FRDSG003_dsgxx.pch` where `xx` is the design cycle, to import in one shot.

167. Push the **Open** button

Post-Processing the Results (Element Force)

168. Select the **Post** tab
169. Push the **Freq. Resp. Plot** button
170. Push the **New Freq. Resp. Plot** button from the Edit Menu toolbar
171. Push the **Magnitude + Phase** radio button
172. Push **Next>**
173. Push the + button
174. Choose `Cycle yy Loadcase 2 Elas force` (where yy corresponds to the last design cycle)
175. Push **Next>**
176. Choose `Element 11000004`
177. Push **Next>**

   A Frequency Response Plot will appear.

178. Right click on the graph
179. Push the **Save Image...** button
180. Enter the name `FRDSG003`
181. Push the **Save** button
182. Find the maximum value of the 3 highest peaks by highlighting around the maximum frequencies.

   Maximum and minimum responses between the highlighted loading frequency range along with loading frequency values are shown in the bottom left corner in the plot. Write down the

Maximum responses values

Frequency (Hz)	Magnitude Reference Solution (1)	Magnitude Solution (2)
350	5.118	
430	11.22	
810	11.25	

(1) Result from plotting `FRDSG003_ref_dsg00.pch`
(2) Result from plotting `FRDSG003_dsgxx.pch`

183. Push the + button
184. Choose `Cycle 0 Loadcase 2 Elas force`
185. Push **Next>**
186. Choose `Element 11000004`
187. Push **Next>**

    An other Frequency Response Plot will appear on the same graph.

188. Right click on the graph
189. Push the **Save Image...** button
190. Enter the name `FRDSG003_comparison`
191. Push the **Save** button
192. Push the **Finish** button
193. Push the **Close** button
194. Push the **Up** button

Study the results

195. In a text editor, open the file `FRDSG003_dsg.out`

    Study the Design Variables History

196. Go to the end of the file

197. Write down the last value of the design variables

Design Variable	Value Reference Solution (1)	Value Solution (2)
Tube	6.8415E-01	
DPB101K1	9.9995E-01	
DPB101K2	1.0005E+00	
DPB101K3	1.3000E+00	
DPB103K3	1.3000E+00	
DPB103K5	1.0026E+00	
DPB103K6	1.0000E+00	
DPB105K123	9.9999E-01	
DPB106K12456	1.0005E+00	
DPB106K3	7.0006E-01	
PELAS104	7.9592E-01	
beta	6.9703E-01	
Box	1.5124E+00	

(1) Result from `FRDSG003_ref_dsg.out`
(2) Result from your run, `FRDSG003_dsg.out`

Quit Design Studio

198. From the main menu bar, select **File** → **Quit**

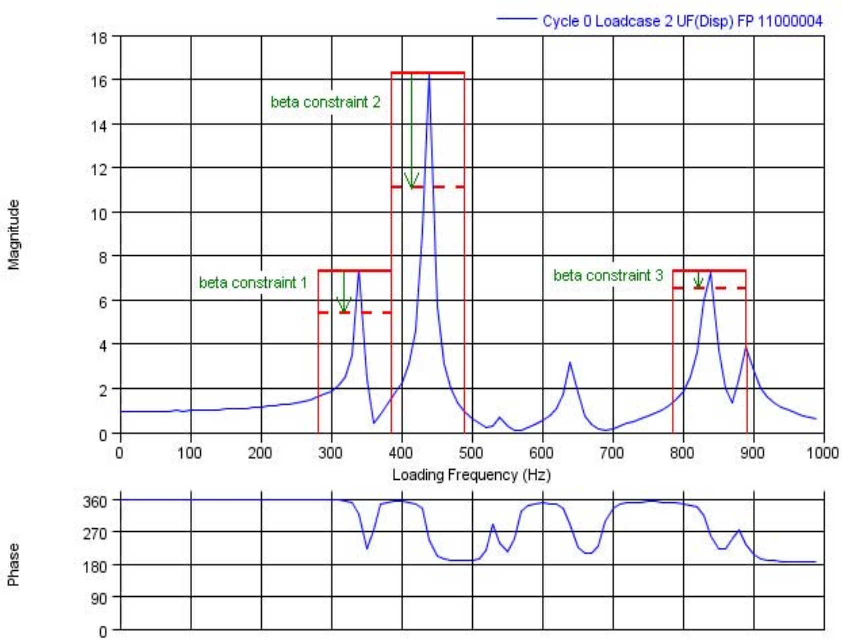
199. Push the **Don’t Save** button

10.4 Driveline design using the beta method IIa

Introduction

In this example, you will learn to optimize dynamic responses using the multiple beta method.

Dynamic responses are dependent on loading frequencies and we often need to minimize the peaks (maximums) dynamic response over the applied loading frequencies. However, such a response, selected by DRESP1 entry, creates vector values rather than scalar values. Therefore, it cannot be directly optimized since the objective function needs to be a single value response. To overcome this difficulty, we introduce 3 artificial design variables called *beta1*, *beta2*, *beta3* and additional constraint equations using the *betas* design variables. The objective function is set to minimize a linear combination of *beta1*, *beta2*, *beta3*. If the values of the betas are reduced, the peaks (maximums) value of the dynamic response must be reduced in order to satisfy the beta constraints. This method is called the *multiple beta method* and is commonly used to solve the min-max (minimizing at the maximum response) problem.



In this example, you will also learn how to perform sizing.

The following optimization problem will be created, solved and post-processed:

Minimize *F(beta1,beta2,beta3)*

where, *F(beta1,beta2,beta3) = (Scale Factor1*beta1 + Scale Factor2*beta2 + Scale Factor3*beta3)/(Scale Factor1 + Scale Factor2 + Scale Factor3)*

Subject to:

(Dynamic Response 1)/(Scale Factor 1) - *beta1* < 0.0

(Dynamic Response 2)/(Scale Factor 2) - *beta2* - *0.1* < 0.0

(Dynamic Response 3)/(Scale Factor 3) - *beta3* - *0.5* < 0.0

Designable region:

The CELAS element

The CBUSH elements

The thickness of the PSHELL groups

IMPORTANT NOTE: If you have already worked through Example 3 (Driveline design using the beta method I), you should go directly to Example 5 (Driveline design using the beta method IIb), which uses the files created in Example 3 and allows you to save time.

Example ID

`FRDSG004`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`FRDSG004.dat`	`Input data`	Provided: Contains the finite element mesh of the driveline
`FRDSG004_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `FRDSG004.dat`

FRDSG004_ref.dat	Input data	Provided as a back up: This file contains the same data as the file FRDSG004_dsg.dat
FRDSG004.dsg	DSG file	Generated by Design Studio. This is the Design Studio database file
FRDSG004_dsg.out	output file	Generated using Genesis within Design Studio. This file contains all the output data created while running genesis
FRDSG004_dsgxx.pch	punch file	File generated using Genesis within Design Studio. These file contains the element force for post-processing for the final design, xx corresponds to the design cycle number.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `FRDSG004.dat`

Create an element set

To plot the force in an element, you need to create an element set which contains this element.

3. Select the **Analysis** tab
4. From the category chooser, select **Element Sets**
5. Push the **New Element Set** button from the Edit Menu toolbar
6. Enter `Set 1` for the name
7. Type in Select by Element ID: `11000004`
8. Push the **Add** button

   Verify that there is the message "1 element selected".
9. Push the **Finish** button

   Verify that the Element Set you just created is listed.

Create the Loadcase to plot the Element Force

You will learn how to set up a file in order to plot the Force in an element.

10. Select the **Analysis** tab
11. From the category chooser, select **Loadcases**
12. Select `Loadcase 1`
13. From the Edit menu toolbar, select the **Delete Loadcase** button
14. Push the **New Loadcase** button from the Edit Menu toolbar
15. Enter `Element_Force` for the name

16. Select the **Modal Frequency Response** radio button
17. Push **Next>**

    In this case, there are nor SPC neither MPC. In your model, you may have one of them or both. Don't forget to select them.

18. Push **Next>**
19. From the **Eigenvalue Method** category chooser, select `50 Method 50`
20. Push **Next>**
21. From the **Dynamic Load Set** category chooser, select `1 DLoad Set`
22. From the **Loading Frequency Set** category chooser, select `2 Frequency Set`
23. From the **Modal Damping** category chooser, select `3 Modal Damping`
24. Push **Next>**
25. From the first category chooser for **Element Force**, select **Post**

    Post: element force for all elements will be output to the post processing file.
    Print: element force for all elements will be output to the output file.
    Both: element force for all elements will be output to both the output file and the post processing file.

26. From the second category chooser for **Element Force**, select `Set 1`

    All: element force for all elements will be output.
    Set 1: element force for all elements in the Set 1 will be output

27. Push the **Finish** button

Create the Loadcases to the optimization problem

You will need one loadcase per constraint.

28. From the **Analysis** category chooser, select **Loadcases**
29. Push the **New Loadcase** button from the Edit Menu toolbar
30. Enter `Element_Force1` for the name
31. Select the **Modal Frequency Response** radio button
32. Push **Next>**

    In this case, there are neither SPC nor MPC. In your model, you may have one of them or both. Don't forget to select them.

33. Push **Next>**
34. From the **Eigenvalue Method** category chooser, select `50 Method 50`
35. Push **Next>**

36. From the **Dynamic Load Set** category chooser, select `1 DLoad Set`
37. From the **Loading Frequency Set** category chooser, select `21 Frequency Set 21`

    The frequency set 21 goes from 290 Hz to 390 Hz.

38. From the **Modal Damping** category chooser, select `3 Modal Damping`
39. Push **Next>**

    This loadcase will be use for the computing. That's why we leave blank the Request Results window.

40. Push the **Finish** button
41. Push the **New Loadcase** button from the Edit Menu toolbar
42. Enter `Element_Force2` for the name
43. Select the **Modal Frequency Response** radio button
44. Push **Next>**

    In this case, there are neither SPC nor MPC. In your model, you may have one of them or both. Don't forget to select them.

45. Push **Next>**
46. From the **Eigenvalue Method** category chooser, select `50 Method 50`
47. Push **Next>**
48. From the **Dynamic Load Set** category chooser, select `1 DLoad Set`
49. From the **Loading Frequency Set** category chooser, select `22 Frequency Set 22`

    The frequency set 22 goes from 400 Hz to 500 Hz.

50. From the **Modal Damping** category chooser, select `3 Modal Damping`
51. Push **Next>**

    This loadcase will be use for the computing. That's why we leave blank the Request Results window.

52. Push the **Finish** button
53. In the list, select the loadcase `Element_Force2`
54. From the Edit Menu toolbar, select the **Copy Loadcase** button
55. From the Edit Menu toolbar, select the **Paste Loadcase** button
56. In the list, select the loadcase `Copy of Element_Force2`
57. Push the **Modify Loadcase** button from the Edit Menu toolbar

58. Enter `Element_Force3` for the name

    Notice that the loadcase type is Modal Frequency Response and that you cannot change it.

59. Push **Next>**

    In this case, there are neither SPC nor MPC. In your model, you may have one of them or both. Don't forget to select them.

60. Push **Next>**
61. Push **Next>**
62. From the **Loading Frequency Set** category chooser, select `23 Frequency Set 23`

    The frequency set 23 goes from 690 Hz to 900 Hz.

63. Push **Next>**

    This loadcase will be used for the computing. That's why we leave blank the Request Results window.

64. Push the **Finish** button

    Verify that you have 4 Loadcases listed.
    The Loadcases `Element_Force1, Element_Force2 and Element_Force3` will be used for the computing.
    The loadcase `Element_Force` will be used for the plotting.

Designing the PSHELL groups

65. Select the **Design** tab
66. From the category chooser, select **Sizing**
67. Select PSHELL 3 and PSHELL 4

    When selecting from the list, to select more than one group, hold the **Ctrl** key while selecting the second group. When selecting from the viewport, you do not need to hold the **Ctrl** key.

68. Push the **Modify Sizing Design** button from the Edit Menu toolbar
69. Check the **Thickness** checkbox

    You need to create a design variable and then come back where you are. Do not push **Finish** or **Cancel** at this point.

70. From the **Design** category chooser, select **Design Variables**
71. Push the **New Design Variable** button from the Edit Menu toolbar
72. Enter `Tube` as the Name and accept the default selection of **Independent Design Variable**
73. Push **Next>**
74. Enter `1.0` for **Initial Value**, `0.5` for **Lower Bound** and `1.5` for **Upper Bound**

75. Push the **Finish** button
76. From the **Design** category chooser, select **Sizing**

    You are back where you left off before design variable creation

77. Select `1 Tube` in the **Thickness** category chooser
78. Push the **Finish** button

    Now, you see hammers icon next to the groups PSHELL 3 and PSHELL 4 in the list. This indicates that they are being size-designed.

79. Select PSHELL 6
80. Push the **Modify Sizing Design** button from the Edit Menu toolbar

    You need to create a design variable and then come back where you are. Do not push **Finish** or **Cancel** at this point.

81. From the **Design** category chooser, select **Design Variables**
82. Push the **New Design Variable** button from the Edit Menu toolbar
83. Enter `Box` as the Name and select **Independent Design Variable**
84. Push **Next>**
85. Enter `2.0` for **Initial Value**, `1.5` for **Lower Bound** and `2.5` for **Upper Bound**
86. Push the **Finish** button
87. From the **Design** category chooser, select **Sizing**

    You will be back where you left off before design variable creation

88. Select `2 Box` in the **Thickness** category chooser
89. Push the **Finish** button

    Now, you see hammers icon next to the groups PSHELL 6 in the list. This indicates that they are being size-designed.

Designing the PELAS group

90. Select PELAS 104
91. Push the **Modify Sizing Design** button from the Edit Menu toolbar

    You need to create a design variables and then come back where you are. Do not push **Finish** or **Cancel** at this point.

92. From the **Design** category chooser, select **Design Variables**
93. Push the **New Design Variable** button from the Edit Menu toolbar
94. Enter `DPELAS104` as the Name and select **Independent Design Variable**
95. Push **Next>**

96. Enter `1.0` for **Initial Value**, `0.6` for **Lower Bound** and `1.4` for **Upper Bound**
97. Push the **Finish** button
98. Push the **New Design Variable** button from the Edit Menu toolbar
99. Enter `PELAS104` as the Name and select **Synthetic Design Variable**
100. Push **Next>**
101. From the **Master Variable** Category chooser, select `DPELAS104`
102. Enter `35000.0` as coefficient
103. Push the **Finish** button
104. From the **Design** Category chooser, select **Sizing**

     You are back where you left off before design variable creation
105. From the **Stiffness** Category chooser, Select `PELAS104`
106. Push the **Finish** button

Designing the PBUSH groups

Some of the stiffness quantities on each of the PBUSH groups are designed. Below is a table with the designable quantities in each of the PBUSH groups along with the design variables used to control the quantities.

Property/Group	Designable Quantities	Design Variables	
		Synthetic	Independent
PBUSH 101	K1 K2 K3	PB101K1 PB101K2 PB101K3	DPB101K1 DPB101K2 DPB101K3
PBUSH 103	K3 K5 K6	PB103K3 PB103K5 PB103K6	DPB103K3 DPB103K5 DPB103K6
PBUSH 105	K1, K2, K3	PB105K123	DPB105K123
PBUSH 106	K1, K2,K4, K5, K6 K3	PB106K12456 PB106K3	DPB101K12456 DPB101K3

For scaling reasons, it is better to normalize the design variables. That is the reason why synthetic variables are used to determine the stiffness values from each of the PBUSH groups. Each of the independent variables are defined with a 30% variation around 1.0.

First all the Independent design variables are created. These independent variables are then used to create synthetic design variables. Finally the synthetic design variables are associated with the design of the PBUSH groups.

Creating the Independent Design Variables:

107. From the **Design** category chooser, select **Design Variables**
108. Push the **New Design Variable** button from the Edit Menu toolbar
109. Enter `DPB101K1` as the **Name**
110. Make sure the **Independent Design Variable** option is selected
111. Push **Next>**
112. Enter `1.0` for **Initial Value**, `0.7` for **Lower Bound** and `1.3` for **Upper Bound**

    You could also enter `15000.0` for **Initial Value**, `0.7*15000.0` for **Lower Bound** and `1.3*15000.0` for **Upper Bound**. However, for scaling reasons, it is better to normalize the design variables. If you create the design variables using the original values, the **Synthetic Design Variable** `PB101K1` need not be created

113. Push the **Finish** button
114. Repeat the steps 108 to 113 with the data given in the following table to design to define all the independent design variables

    The independent design variables in the first row has already been defined in steps 72 to 76.

Variable Name	Initial value	Lower Bound	Upper Bound
DPB101K1	1.0	0.7	1.3
DPB101K2	1.0	0.7	1.3
DPB101K3	1.0	0.7	1.3
DPB103K3	1.0	0.7	1.3
DPB103K5	1.0	0.7	1.3
DPB103K6	1.0	0.7	1.3
DPB105K123	1.0	0.7	1.3
DPB106K12456	1.0	0.7	1.3
DPB106K3	1.0	0.7	1.3

Creating the Synthetic Design Variables:

115. From the **Design** category chooser, select **Design Variables**
116. Push the **New Design Variable** button from the Edit Menu toolbar
117. Enter `PB101K1` as the **Name**
118. Select **Synthetic Design Variable** option

119. Push **Next>**
120. From the **Master Variable** Category chooser, select `DPB101K1`
121. Enter `15000.0` as coefficient
122. Push the **Finish** button
123. Repeat the steps 116 to 122 with the data given in the following table to create the synthetic design variables

    The synthetic design variables in the first row has already been defined.

Variable Name	Coefficient	Master Variable
PB101K1	15000	DPB101K1
PB101K2	50000	DPB101K2
PB101K3	40000	DPB101K3
PB103K3	30000	DPB103K3
PB103K5	50000	DPB103K5
PB103K6	75000	DPB103K6
PB105K123	1.0E+10	DPB105K123
PB106K12456	2500	DPB106K12456
PB106K3	1500	DPB106K3

Assigning design variables to PBUSH groups:

124. From the **Design** Category chooser, select **Sizing**
125. Select PBUSH 101
126. Push the **Modify Sizing Design** button from the Edit Menu toolbar
127. From the **K1** Category chooser, select `PB101K1`
128. From the **K2** Category chooser, select `PB101K2`
129. From the **K3** Category chooser, select `PB101K3`
130. Push the **Finish** button
131. Repeat the steps 125 to 130 with the data given in the following table to design the PBUSH groups

    The designable properties of PBUSH101 (first row in the table below) has already been defined in the earlier steps.

Group	Bushing Property Variable to be modified	Associated Variable
PBUSH101	K1	PB101K1
	K2	PB101K2
	K3	PB101K3
PBUSH103	K3	PB103K3
	K5	PB103K5
	K6	PB103K6
PBUSH105	K1	PB105K123
	K2	PB105K123
	K3	PB105K123
PBUSH106	K1	PB106K12456
	K2	PB106K12456
	K3	PB106K3
	K4	PB106K12456
	K5	PB106K12456
	K6	PB106K12456

Defining the beta method II constraint equations

The *beta method* equations are:

(Dynamic Response 1)/(Scale Factor 1) - *beta1* < 0.0

(Dynamic Response 2)/(Scale Factor 2) - *beta2 -0.1* < 0.0

(Dynamic Response 3)/(Scale Factor 3) - *beta3* - *0.5* < 0.0

The Scale Factors are calculated during the modal analysis process.

The Dynamic Responses are calculated by Genesis for each design cycle.

The design variables *beta1, beta2,* and *beta3* are created.

132. From the **Design** category chooser, select **Synthetic Responses**
133. Push the **New Synthetic Response** button from the Edit Menu toolbar
134. Enter `beta constraint 1` in the **Name**
135. Make sure the **Type** selected is **User Function (DRESP2)**

136. Push **Next>**
137. Push the + button
138. Make sure the **Fundamental Response** radio button is selected
139. Push **Next>**
140. Select **More Response Types...** from the **Response Type** radio button
141. Push **Next>**
142. Select **Dynamic Force** from the **Response Type** radio button and select **Selected Elements** from the category chooser
143. Select **Magnitude** from the **Select Dynamic Component**
144. Push **Next>**
145. Enter `11000004` in **Select by Element ID**
146. Push the **Add** button
147. Push **Next>**
148. Push **Next>**
149. Select the Loadcase `Element_Force1`
150. Push **Next>**

    You need to create a design variable (`beta1`) and then come back where you are. Do not push **Finish** or **Cancel** at this point.

151. From the **Design** category chooser, select **Design Variables**
152. Push the **New Design Variable** button from the Edit Menu toolbar
153. Enter `beta1` as the **Name** and select **Independent Design Variables**
154. Push **Next>**
155. Enter `1.0` for **Initial Value**, `0.0` for **Lower Bound** and `1.0` for **Upper Bound**
156. Push the **Finish** button
157. From the **Design** category chooser, select **Synthetic Responses**

    You are back where you left off before design variable creation.

158. Push the + button
159. Select **Design Variable** radio button and select `beta1`
160. Push **Next>**

    Now we have the two arguments required to state the synthetic response.

161. For the equation, enter `F = Arg1/(Scale Factor1) - Arg2`

    You are free to change those argument names anyway you like.
    The value of Scale Factor is usually the magnitude of the peak obtained in the initial analysis.
    The table below contains the Scale Factors for this problem.

162. Push the **Finish** button

163. Repeat the step 133 to 162 twice for defining the 2 other constraint equations using the following table (replace `beta1` by `beta2` the first time and `beta1` by `beta3` the second time. Also use `Element_Force2` loadcase the first time and `Element_Force3` loadcase the second time)

    Verify that you have 3 synthetic response listed.
    Important: As you can see in the definition of the second and third equation (linked to the second and third peak), we introduce a pre-existent constraint of -10% and -50%. In this case, the problem is defined such that even if the second and third peaks increase, the results are still acceptable as long as they do not increase more than 10% and 50% respectively.
    Homework: Optimize the problem without including the pre-existent constraints.

Scale Factor Name	Scale Factor value	Equation
Scale Factor 1	7.376	F = Arg1/(Scale Factor1) - Arg2
Scale Factor 2	16.17	F = Arg1/(Scale Factor2) - Arg2 - 0.1
Scale Factor 3	7.251	F = Arg1/(Scale Factor3) - Arg2 - 0.5

Defining the Design Constraints

164. From the **Design** category chooser, select **Constraints**
165. Push the **New Constraint** button from the Edit Menu toolbar
166. Select **Synthetic Response** as Response Type.
167. Enter `0.0` for **Upper bound**
168. Push **Next>**
169. Select the `beta constraint 1` Synthetic Response
170. Push the **Finish** button
171. Push the **New Constraint** button from the Edit Menu toolbar
172. Select **Synthetic Response** as Response Type.
173. Enter `0.0` for **Upper bound**
174. Push **Next>**
175. Select the `beta constraint 2` Synthetic Response
176. Push the **Finish** button

177. Push the **New Constraint** button from the Edit Menu toolbar
178. Select **Synthetic Response** as Response Type.
179. Enter `0.0` for **Upper bound**
180. Push **Next>**
181. Select the `beta constraint 3` Synthetic Response
182. Push the **Finish** button

    Verify there are 3 constraints listed.

Defining the beta method II objective equation

183. From the **Design** category chooser, select **Synthetic Responses**
184. Push the **New Synthetic Response** button from the Edit Menu toolbar
185. Enter `beta objective` in the **Name**
186. Make sure the **Type** selected is **User Function (DRESP2)**
187. Push **Next>**
188. Push the **+** button
189. Select the **Design Variable** radio button and select `beta1`
190. Push **Next>**
191. Push the **+** button
192. Select the **Design Variable** radio button and select `beta2`
193. Push **Next>**
194. Push the **+** button
195. Select the **Design Variable** radio button and select `beta3`
196. Push **Next>**
197. For the equation, enter `F = (Scale Factor1*Arg1 + Scale Factor2*Arg2 + Scale Factor3*Arg3)/(Scale Factor1 + Scale Factor2 + Scale Factor3)`

    Use the scale factors defined in the table earlier.

198. Push the **Finish** button

Defining the Design Objectives

199. From the **Design** category chooser, select **Objectives**

200. Push the **New Objective** button from the Edit Menu toolbar
201. Select **Synthetic Response** as Response Type
202. Select **Min** as Objective definition
203. Push **Next>**
204. Select the `beta objective` Synthetic Response
205. Push **Next>**
206. Select the `Element_Force` Loadcase
207. Push the **Finish** button

    Verify there is 1 objective listed.

Clearing the Selection

208. Right click the Viewport, select **Clear** → **All**

Setup the Genesis Options

209. From the main menu bar, select **Genesis** → **Options...**
210. In the **Output Control** tab, select **Print Module Times: Both**
211. In the **Output Control** tab, select **Analysis Output: First & Last**
212. In the **Output Control** tab, select **Design Output: All Cycles**
213. In the **Design Control** tab, select **Maximum Design Cycles** checkbox and enter `30` for the maximum cycles
214. Push the **Apply** button

Save the Design Studio database

215. From the main menu bar, select **File** → **Save as...**
216. Name the file `FRDSG004`
217. Push the **Save** button

Optimize the structure using Genesis

218. From the main menu bar, select **Genesis** → **Optimize**
219. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

220. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
221. Select the `FRDSG004_dsg00.pch` file
222. Check the **Import Similar Results for All Design Cycles** check box

    This will allow all similar files, e.g., `FRDSG004_dsgxx.pch` where `xx` is the design cycle, to import in one shot.

223. Push the **Open** button

Post-Processing the Results (Element Force)

224. Select the **Post** tab
225. Push the **Freq. Resp. Plot** button
226. Push the **New Freq. Resp. Plot** button from the Edit Menu toolbar
227. Push the **Magnitude + Phase** radio button
228. Push **Next>**
229. Push the **+** button
230. Choose `Cycle yy Loadcase 2 Elas Force` (where `yy` corresponds the last design cycle)
231. Push **Next>**
232. Choose `Element 11000004`
233. Push **Next>**

    A Frequency Response Plot will appear.

234. Right click on the graph
235. Push the **Save Image...** button
236. Enter the name `FRDSG004`
237. Push the **Save>** button
238. Find the maximum value of the 3 highest peaks by highlighting around the maximum frequencies.

    Maximum and minimum responses between the highlighted loading frequency range along with loading frequency values are shown in the bottom left corner in the plot. Write down the

Maximum responses values

Frequency (Hz)	Magnitude Reference Solution (1)	Magnitude Solution (2)
340	7.373	
420	13.44	
740	5.089	

(1) Result from plotting `FRDSG004_ref_dsg00.pch`
(2) Result from plotting `FRDSG004_dsgxx.pch`

239. Push the + button
240. Choose `Cycle 0 Loadcase Elas Force`
241. Push **Next>**
242. Choose `Element 11000004`
243. Push **Next>**

    An other Frequency Response Plot will appear on the same graph.

244. Right click on the graph
245. Push the **Save Image...** button
246. Enter the name `FRDSG004_comparison`
247. Push the **Save>** button
248. Push the **Finish** button
249. Push the **Close** button
250. Push the **Up** button

Quit Design Studio

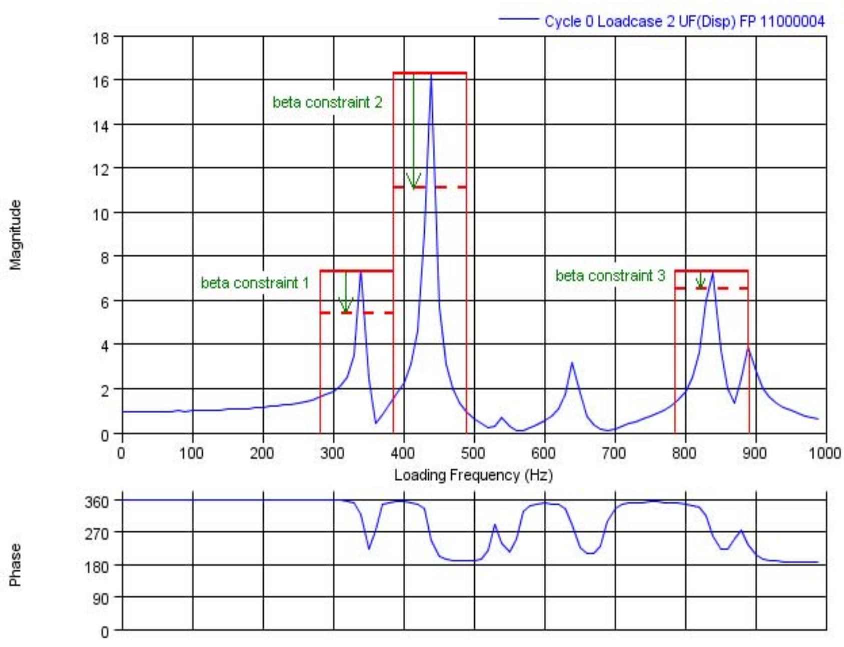
251. From the main menu bar, select **File** → **Quit**
252. Push the **Don't Save** button

10.5 Driveline design using the beta method IIb

Introduction

In this example, you will learn to optimize dynamic responses using the multiple beta method.

Dynamic responses are dependent on loading frequencies and we often need to minimize the peaks (maximums) dynamic response over the applied loading frequencies. However, such a response, selected by DRESP1 entry, creates vector values rather than scalar values. Therefore, it cannot be directly optimized since the objective function needs to be a single value response. To overcome this difficulty, we introduce 3 artificial design variables called *beta1*, *beta2*, *beta3* and additional constraint equations using the *betas* design variables. The objective function is set to minimize a linear combination of *beta1*, *beta2*, *beta3.* If the values of the betas are reduced, the peaks (maximums) value of the dynamic response must be reduced in order to satisfy the beta constraints. This method is called the *multiple beta method* and is commonly used to solve the min-max (minimizing at the maximum response) problem.



In this example, you will also learn how to perform sizing.

The following optimization problem will be created, solved and post-processed:

Minimize *F(beta1,beta2,beta3)*

where, *F(beta1,beta2,beta3) = (Scale Factor1*beta1 + Scale Factor2*beta2 + Scale Factor3*beta3)/(Scale Factor1 + Scale Factor2 + Scale Factor3)*

Subject to:

(Dynamic Response 1)/(Scale Factor 1) - *beta1* < 0.0

(Dynamic Response 2)/(Scale Factor 2) - *beta2* - *0.1* < 0.0

(Dynamic Response 3)/(Scale Factor 3) - *beta3* - *0.5* < 0.0

Designable region:

The CELAS element

The CBUSH elements

The thickness of the PSHELL groups

Example ID

`FRDSG005`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`FRDSG003.dat`	`Input data`	Created in Example 3: contains all the finite element data plus the optimization data created in example 3
`FRDSG005.dsg`	`DSG file`	Generated by Design Studio. This is the Design Studio database file
`FRDSG005_dsg.dat`	`Input data`	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in `FRDSG003.dat`
`FRDSG005_ref.dat`	`Input data`	Provided as a back up: This file contains the same data as the file `FRDSG005_dsg.dat`
`FRDSG005_dsg.out`	`output file`	Generated using Genesis within Design Studio. This file contains all the output data created while running genesis

`FRDSG005_dsgxx.pch`	`punch file`	File generated using Genesis within Design Studio. These file contains the element force for post-processing for the final design, `xx` corresponds to the design cycle number.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `FRDSG003.dat`

   Note: you can also import the file `FRDSG003_ref.dat`.

Create the Loadcases to the optimization problem

You will need one loadcase per constraint.

3. Select the **Analysis** tab
4. From the category chooser, select **Loadcases**
5. Push the **New Loadcase** button from the Edit Menu toolbar
6. Enter `Element_Force1` for the name
7. Select the **Modal Frequency Response** radio button
8. Push **Next>**

   In this case, there are nor SPC neither MPC. In your model, you may have one of them or both. Don't forget to select them.

9. Push **Next>**
10. From the **Eigenvalue Method** category chooser, select `50 Method 50`
11. Push **Next>**
12. From the **Dynamic Load Set** category chooser, select `1 DLoad Set`
13. From the **Loading Frequency Set** category chooser, select `21 Frequency Set 21`

    The frequency set 21 goes from 290 Hz to 390 Hz.

14. From the **Modal Damping** category chooser, select `3 Modal Damping`

    This loadcase will be used for the computing. That's why we leave the Request Results blank.

15. Push the **Finish** button
16. Push the **New Loadcase** button from the Edit Menu toolbar
17. Enter `Element_Force2` for the name
18. Select the **Modal Frequency Response** radio button

19. Push **Next>**

    In this case, there are nor SPC neither MPC. In your model, you may have one of them or both. Don't forget to select them.

20. Push **Next>**
21. From the **Eigenvalue Method** category chooser, select `50 Method 50`
22. Push **Next>**
23. From the **Dynamic Load Set** category chooser, select `1 DLoad Set`
24. From the **Loading Frequency Set** category chooser, select `22 Frequency Set 22`

    The frequency set 22 goes from 400 Hz to 500 Hz.

25. From the **Modal Damping** category chooser, select `3 Modal Damping`
26. Push the **Finish** button
27. In the list, select the loadcase `Element_Force2`
28. From the Edit Menu toolbar, select the **Copy Loadcase** button
29. From the Edit Menu toolbar, select the **Paste Loadcase** button
30. In the list, select the loadcase `Copy of Element_Force2`
31. Push the **Modify Loadcase** button from the Edit Menu toolbar
32. Enter `Element_Force3` for the name

    Notice that the loadcase type is Modal Frequency Response and that you cannot change it.

33. Push **Next>**

    In this case, there are nor SPC neither MPC. In your model, you may have one of them or both. Don't forget to select them.

34. Push **Next>**
35. Push **Next>**
36. From the Loading Frequency Set category chooser, select `23 Frequency Set 23`

    The frequency set 23 goes from 690 Hz to 900 Hz.

37. Push the **Finish** button

    Verify that you have 4 Loadcases listed.
    The Loadcases `Element_Force1, Element_Force2 and Element_Force3` will be used for the computing.
    The loadcase `Element_Force` will be used for the plotting.

Defining the beta method II constraint equations

The *beta method* equations are:

(Dynamic Response 1)/(Scale Factor 1) - *beta1* < 0.0

(Dynamic Response 2)/(Scale Factor 2) - *beta2* - *0.1* < 0.0

(Dynamic Response 3)/(Scale Factor 3) - *beta3* - *0.5* < 0.0

The Scale Factors are calculated during the modal analysis process.

The Dynamic Responses are calculated by Genesis for each design cycle.

The *beta* design variables are created.

38. From the **Design** category chooser, select **Synthetic Responses**
39. Select the Synthetic Response `beta constraint`
40. From the Edit menu toolbar, select the **Delete Synthetic Response** button
41. Push the **New Synthetic Response** button from the Edit Menu toolbar
42. Enter `beta constraint 1` in the Name
43. Make sure the **Type** selected is **User Function (DRESP2)**
44. Push **Next>**
45. Push the **+** button
46. Make sure the **Fundamental Response** radio button is selected
47. Push **Next>**
48. Select **More Response Types...** from the **Response Type** radio button
49. Push **Next>**
50. Select **Dynamic Force** radio button for the Additional Response and select **Selected Elements** from the category chooser
51. Select **Magnitude** from the **Select Dynamic Component**
52. Push **Next>**
53. Enter `11000004` in **Select by Element ID**

    Check if 1 element is selected.
54. Push the **Add** button
55. Push **Next>**
56. Push **Next>**
57. Select the Loadcase `Element_Force1`
58. Push **Next>**

    You need to create a design variable (`beta1`) and then come back where you are. Do not push **Finish** or **Cancel** at this point.
59. From the **Design** category chooser, select **Design Variables**

60. Select the Design Variable `beta`
61. From the Edit menu toolbar, select the **Delete Design Variable** button
62. Push the **New Design Variable** button from the Edit Menu toolbar
63. Enter `beta1` as the Name and select **Independent Design Variable**
64. Push **Next>**
65. Enter `1.0` for **Initial Value**, `0.0` for **Lower Bound** and `1.0` for **Upper Bound**
66. Push the **Finish** button
67. From the **Design** category chooser, select **Synthetic Responses**

    You are back where you left off before design variable creation.

68. Push the + button
69. Select **Design Variable** from the Response Type radio button and Select `beta1`
70. Push **Next>**

    Now we have the two arguments required to state the synthetic response.

71. For the equation, enter `F = Arg1/(Scale Factor1) - Arg2`

    You are free to change those argument names anyway you like.
    Use the values for `Scale Factor`, from the table below.

72. Push the **Finish** button
73. Repeat the step 42 to 72 for defining the 2 other constraint equations using the Scale Factors table above and the following note (replace `beta1` by `beta2` the first time and `beta1` by `beta3` the second time. Also use `Element_Force2` loadcase the first time and `Element_Force3` loadcase the second time)

    Verify that you have 3 synthetic response listed.
    Important: As you can see in the definition of the second and third equation (linked to the second and third peak), we introduce a pre-existent constraint of -10% and -50%. In this case, the problem is defined such that even if the second and third peaks increase, the results are still acceptable as long as they do not increase more than 10% and 50% respectively.
    Homework: Optimize the problem without including the pre-existent constraints.

Scale Factor Name	Scale Factor value	Equation
`Scale Factor 1`	`7.376`	`F = Arg1/(Scale Factor1) - Arg2`
`Scale Factor 2`	`16.17`	`F = Arg1/(Scale Factor2) - Arg2 - 0.1`
`Scale Factor 3`	`7.251`	`F = Arg1/(Scale Factor3) - Arg2 - 0.5`

Defining the Design Constraints

74. From the **Design** category chooser, select **Constraints**
75. Push the **New Constraint** button from the Edit Menu toolbar
76. Select **Synthetic Response** as Response Type.
77. Enter `0.0` for **Upper bound**
78. Push **Next>**
79. Select the `beta constraint 1` Synthetic Response
80. Push the **Finish** button
81. Push the **New Constraint** button from the Edit Menu toolbar
82. Select **Synthetic Response** as Response Type.
83. Enter `0.0` for **Upper bound**
84. Push **Next>**
85. Select the `beta constraint 2` Synthetic Response
86. Push the **Finish** button
87. Push the **New Constraint** button from the Edit Menu toolbar
88. Select **Synthetic Response** as Response Type.
89. Enter `0.0` for **Upper bound**
90. Push **Next>**
91. Select the `beta constraint 3` synthetic response
92. Push the **Finish** button

    Verify there are 3 constraints listed.

Defining the beta method II objective equation

93. From the **Design** category chooser, select **Synthetic Responses**
94. Select the Synthetic Response `beta objective`
95. From the Edit menu toolbar, select the **Delete Synthetic Response** button
96. Push the **New Synthetic Response** button from the Edit Menu toolbar
97. Enter `beta objective` in the Name
98. Make sure the **Type** selected is **User Function (DRESP2)**
99. Push **Next>**
100. Push the + button
101. Select the **Design Variable** radio button and select `beta1`

102. Push **Next>**
103. Push the + button
104. Select the **Design Variable** radio button and select `beta2`
105. Push **Next>**
106. Push the + button
107. Select the **Design Variable** radio button and select `beta3`
108. Push **Next>**
109. For the equation, enter `F = (Scale Factor1*Arg1 + Scale Factor2*Arg2 + Scale Factor3*Arg3)/(Scale Factor1 + Scale Factor2 + Scale Factor3)`

    Use the values for `Scale Factor`, from the table above.

110. Push the **Finish** button

Defining the Design Objective

111. From the **Design** category chooser, select **Objectives**
112. Push the **New Objective** button from the Edit Menu toolbar
113. Select **Synthetic Response** as Response Type.
114. Select **Min** as Objective definition
115. Push **Next>**
116. Select the `beta objective` Synthetic Response
117. Push the **Finish** button

    Verify there is 1 objective listed.

Setup the Genesis Options

118. From the main menu bar, select **Genesis** → **Options...**
119. In the **Design Control** tab, select **Maximum Design Cycles** checkbox and enter `30` for the maximum cycles
120. Push the **Apply** button

Save the Design Studio database

121. From the main menu bar, select **File** → **Save as...**
122. Name the file `FRDSG005`

123. Push the **Save** button

Optimize the structure using Genesis

124. From the main menu bar, select **Genesis** → **Optimize**
125. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

126. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
127. Select the `FRDSG005_dsg00.pch` file
128. Check the **Import Similar Results for All Design Cycles** check box

    This will allow all similar files, e.g., `FRDSG005_dsgxx.pch` where xx is the design cycle, to import in one shot.

129. Push the **Open** button

Post-Processing the Results (Element Force)

130. Select the **Post** tab
131. Push the **Freq. Resp. Plot** button
132. Push the **New Freq. Resp. Plot** button from the Edit Menu toolbar
133. Push the **Magnitude + Phase** radio button
134. Push **Next>**
135. Push the **+** button
136. Choose `Cycle yy Loadcase 2 Elas Force` (where yy corresponds the last design cycle)
137. Push **Next>**
138. Choose `Element 11000004`
139. Push **Next>**

    A Frequency Response Plot will appear.

140. Right click on the graph
141. Push the **Save Image...** button
142. Enter the name `FRDSG005`
143. Push the **Save>** button

144. Find the maximum value of the 3 highest peaks by highlighting around the maximum frequencies.

    Maximum and minimum responses between the highlighted loading frequency range along with loading frequency values are shown in the bottom left corner in the plot. Write down the Maximum responses values

Frequency (Hz)	Magnitude Reference Solution (1)	Magnitude Solution (2)
340	7.373	
420	13.44	
740	5.089	

    (1) Result from plotting `FRDSG005_ref_dsg00.pch`
    (2) Result from plotting `FRDSG005_dsgyy.pch`

145. Push the + button
146. Choose `Cycle 0 Loadcase 2 Elas Force`
147. Push **Next>**
148. Choose `Element 11000004`
149. Push **Next>**
150. Right click on the graph
151. Push the **Save Image...** button
152. Enter the name `FRDSG005_comparison`
153. Push the **Save>** button
154. Push the **Finish** button
155. Push the **Close** button
156. Push the **Up** button

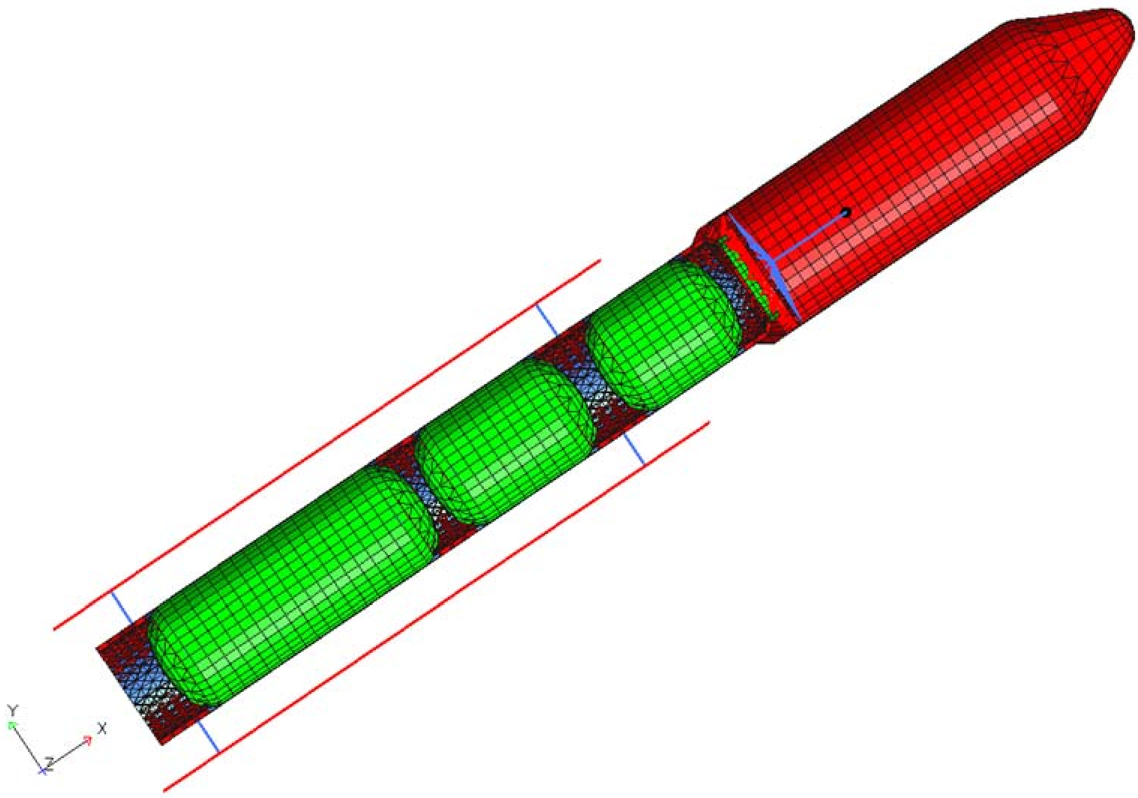
Quit Design Studio

157. From the main menu bar, select **File** → **Quit**
158. Push the **Don’t Save** button

10.6 Payload Integration Design of a Launch Vehicle

Introduction

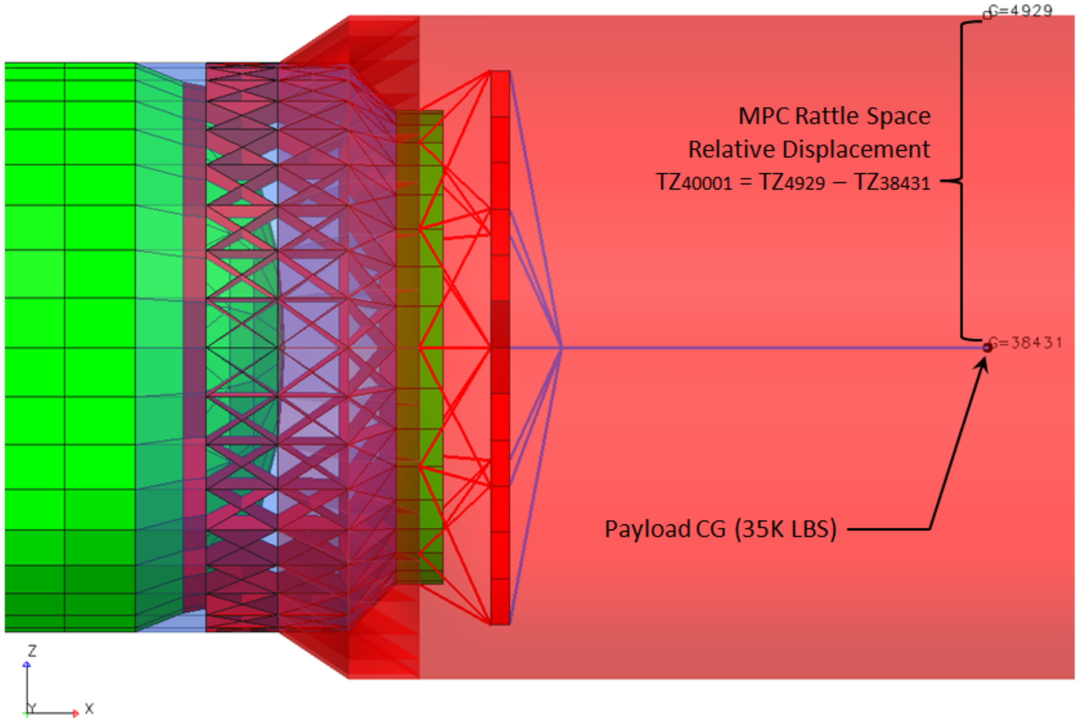
The purpose of this example is to learn how to perform modal frequency response analysis and optimization based on dynamic response. In this example we will use a finite element model of an aerospace launch vehicle (LV) with it's payload (PL) simulated as a CG mass connected at the LV/PL interface via rigid elements. A cut-away picture of the FEM is shown below:



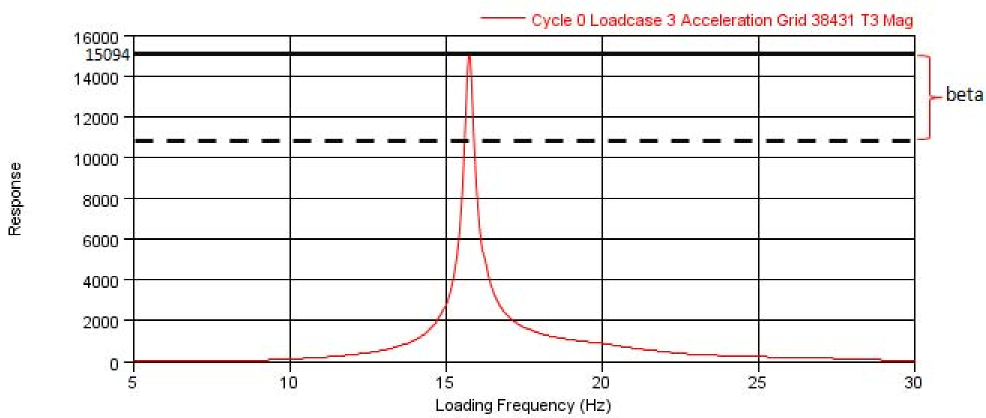
One of several design Launch environments is the rocket engine ignition overpressure pulse that is applied laterally to the structure. Other lateral loads may be caused by both ground winds or local aerodynamic turbulance from winds aloft. Lateral loading tends to excite the payload and the fairing because both are cantilevered from the top of the launch vehicle.

This first part of this example will run a frequency response analysis with a simple 1 psi half cosine pressure applied to just the -Z side of the payload fairing. It consists of low frequency content that starts to drop off at 20Hz to better illustrate the optimization method and results in part 2 of this example.

For payload integration the primary response of interest is the CG acceleration of the rigid payload. Additionally the rattle space relative displacement between the payload CG and a grid point on the fairing at the same X station in line with the applied loads should be monitored. The PL CG and rattle space grids are shown below:



In the second part of the example, topometry optimization is carried out to minimize the dynamic Z-accelerations of the CG of the payload. As this response is a vector quantity *beta* method is used.



The following optimization problem will be created, solved and post-processed:

Minimize *beta*

Subject to:

The rigid payload CG acceleration

(Dynamic ACCE Response)/(Scale Factor) - ***beta*** **< 0.0**

and the rattle space relative displacement

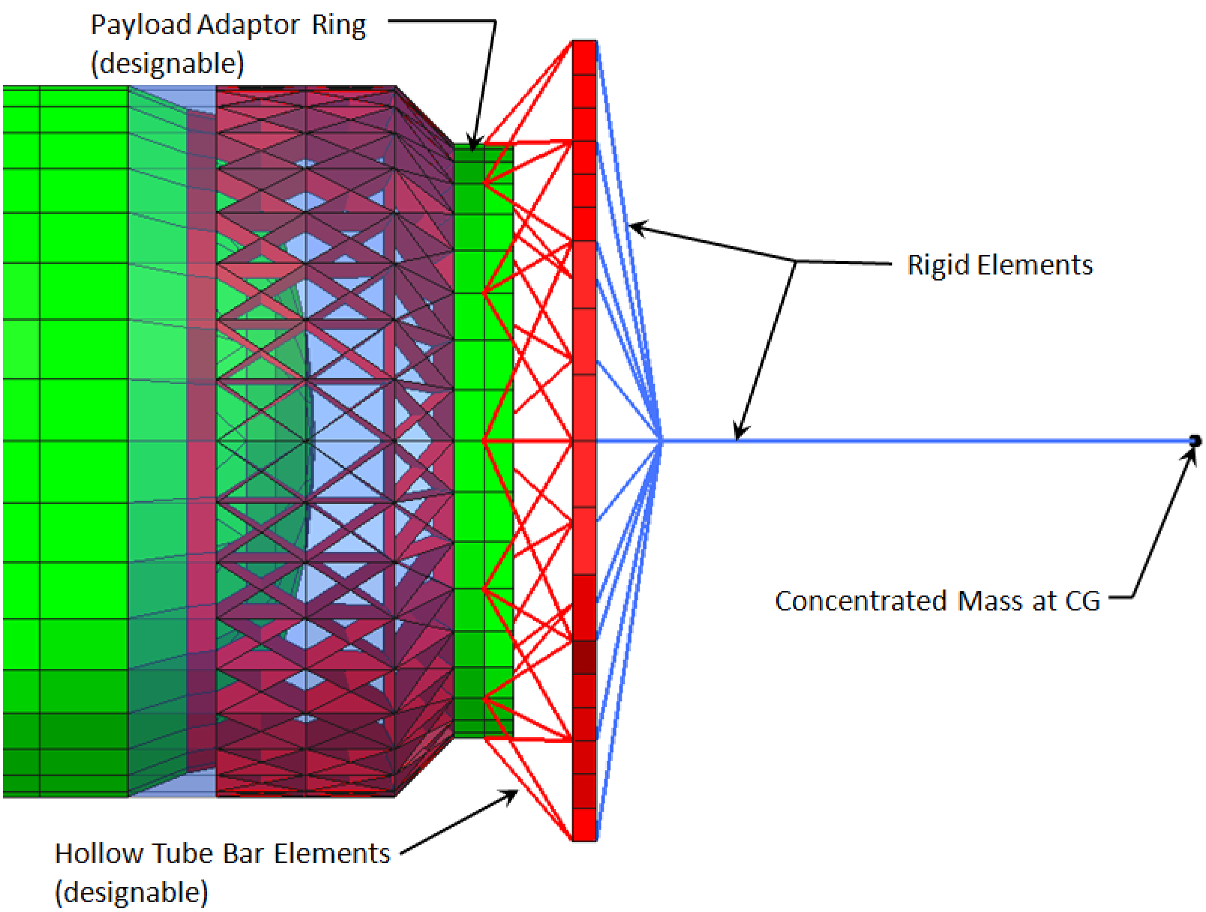
Dynamic DISP Response < 1.0 inch

Designable region for payload integration:

The thickness of the PSHELL group for the payload adaptor ring

The diameter and thickness of the PBARL hollow tubes that connect the PL adaptor ring to the payload platform

The designable regions are shown below:



Along with it's actual mission, payload design always involves investigating the ability of the space vehicle to survive the launch environment. It is common in preliminary design to rigidly couple the mass properties of the payload to the Launch vehicle to evaluate the maximum accelerations expected, and the rattle space between the fairing and the payload.

Part 1 of this problem shows that with the current design the payload CG is subject to approximately 38 G's lateral acceleration. During preliminary payload integration it is desirable to reduce this as much as possible without exceeding 1 inch of rattle space at the CG station (the clearance envelope would thus be greater near the PLF tip). In general, when the elastic payload is coupled to the launch vehicle during the final loads analysis cycle the body accelerations are reduced (i.e. get better), but the rattle space displacements may get worse. Requirements should be written with this in mind.

Example ID

FRDSG006

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
FRDSG006_1.dat	Input data	Provided: Contains the finite element mesh of the launch vehicle and rigid payload
FRDSG006_1.dsg	DSG file	Generated by Design Studio to run Design Studio. This file contains all the data generated in this example plus the data in FRDSG006_1.dat
FRDSG006_1_dsg.out	output file	Generated using Genesis within Design Studio. This file contains FE output-results for analysis.
FRDSG006_2.dat	Input data	Exported using Design Studio in part 1.
FRDSG006_2_ref.dat	Input data	Provided as a back up: This file contains all the data generated in this example plus the data in FRDSG006_2.dat
FRDSG006_2.dsg	DSG file	Generated by Design Studio to run Design Studio. This file contains all the data generated in this example plus the data in FRDSG006_2.dat
FRDSG006_2_dsg.out	output file	Generated using Genesis within Design Studio. This file contains FE post-processing results for analysis.

10.6.1 Part 1

This part of the example goes through the steps of creating and running a frequency response analysis on the launch vehicle model. The bulk data file provided contains all the data required to define the loading, eigensolution method, and MPC rattle space equation. The Design Studio steps below show how to set up the normal modes and modal frequency response loadcases, execute the analysis, and post process the desired response data.

When you finish this part, you should have created a file named: `FRDSG006_2.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `FRDSG006_1.dat`

Examine the Pressure Load on the Payload Fairing

3. Select the **Analysis** tab
4. From the category chooser, select **Static Loads**
5. Select the **HalfCOS_FairingPressure** load

   The arrows on the screen are all the same length even though the pressure is maximum at the center (in the +Z direction) and drops off as a half cosine function towards the edges.

Examine the Eigenvalue Method

6. From the **Analysis** category chooser, select **Eigenvalue Methods**
7. Select the `Lanczos` method
8. Push the **Modify Eigenvalue Method** button from the Edit menu toolbar

   Notice that V1 is set to 1.0 Hz. so that rigid body modes are not recovered.

9. Push the **Finish** button

Create the Frequency Response Data

Design Studio allows the user to input the frequency response data we will use for this problem.

10. From the **Analysis** category chooser, select **Frequency Response Data**
11. Push the **New Freq.Resp.Data** button from the Edit menu toolbar

12. Enter `TC Table` for the **Name**
13. Select the **Freq. Function Table (TABLEDi)** radio button
14. Push the **Next>** button
15. For the **Form**, select **TABLED1** option
16. Keep the first X-Y pair of X=1.0, Y=1.0 and push the **+** button repeatedly to add the following X-Y pairs:

X	Y
20.0	1.0
23.0	0.5
30.0	0.1
31.0	0.1

17. Push the **Finish** button

    Verify that the TABLED data is listed.
18. Push the **New Freq.Resp.Data** button
19. Enter `Lateral Pressure Load` for the **Name**
20. Select the **Dynamic Load Set (RLOADi)** radio button
21. Push the **Next>** button
22. For the **TC Table**, select `TC Table`
23. For the **Static Load**, select `HalfCOS_FairingPressure`
24. Push the **Finish** button

    Verify that the DLOAD data is listed.
25. Push the **New Freq.Resp.Data** button
26. Enter `Modal Damping` for the **Name**
27. Select the **Modal Damping Table (TABDMP1)** radio button
28. Push the **Next>** button
29. For the **Form**, select **CRIT**
30. Fill in two X-Y pairs to define 1% of critical damping:

X	Y
1.0	0.01
30.0	0.01

31. Push the **Finish** button

    Verify that the SDAMP data is listed.

32. Push the **New Freq.Resp.Data** button
33. Enter `Frequencies` for the **Name**
34. Select the **Loading Frequency Set (FREQi)** radio button
35. Push the **Next>** button
36. From the category chooser, select **FREQ1**
37. Type in **F1**=`5.0`, **DF**=`0.05`, and **N**=`500`
38. Push the **Finish** button

    Verify that the FREQ data is listed.

Create the Normal Loads Loadcase

39. From the **Analysis** category chooser, select **Loadcases**
40. Select `Loadcase 1`
41. Push the **Delete Loadcase** button
42. Push the **New Loadcase** button
43. Enter `Unrestrained Normal Modes` for the **Name**
44. Select the **Normal Modes** radio button
45. Push the **Next>** button
46. For the MPC, select `Rattle Space T3 Relative Displ`
47. Push **Next>**
48. For the Eigenvalue method, select `Lanczos`
49. Push **Next>**
50. For the Displacement, select **Post**

    Verify that All appears in the 2nd category chooser.

51. Push the **Finish** button

Create the Frequency Response Loadcase

52. From the **Analysis** category chooser, select **Loadcases**
53. Push the **New Loadcase** button
54. Enter `Modal_Freq_Resp, 1% critical damping` for the **Name**

55. Select the **Modal Frequency Response** radio button
56. Push the **Next>** button
57. Push the **Next>** button

    Do not choose an eigenvalue method. We will use the normal modes from the first loadcase for the frequency response.

58. Push the **Next>** button
59. Choose the Frequency Response Controls from the category chooser according to the following table:

Dynamic Load Set:	`Lateral Pressure Load`
Loading Frequency Set:	`Frequencies`
Modes Loadcase:	`Unrestrained Normal Modes`
Modal Damping:	`Modal Damping`
Random Set:	`None`
Cross Correlation Set:	`None`

60. Push the **Next>** button
61. For Displacement, select **Post** and `Grid Set 2` in the two pull down menu's respectively
62. For Acceleration, select **Post** and `Grid Set 1` in the two pull down menu's respectively
63. Push the **Finish** button

Genesis options

64. From the main menu bar, select **Genesis** → **Options...**
65. In the Output Control tab, select **Print Module Times: Both**
66. In the Output Control tab, select **Analysis Output: First & Last**
67. In the Output Control tab, select **Complex Output: Polar**
68. Push the **Apply** button

Save the Design Studio database

69. From the main menu bar, select **File** → **Save**

Export the input file

70. From the main menu bar, select **File** → **Export** → **Input data...**
71. Name the file `FRDSG006_2`
72. Push the **Save** button

Analyze the structure using Genesis

73. From the main menu bar, select **Genesis** → **Single Analysis**
74. Study the **Genesis Console Output**
75. From the Genesis Console Output window push the **Import Post...** button
76. Select the punched output file and push the **Import** button; when done, push the **Close** button

Post-Processing the Results

77. Select the **Post** tab
78. Push the **Freq. Resp. Plot** button
79. Push the **New Freq. Resp. Plot** button
80. Make sure the **Choose Dynamic Component** radio button is selected
81. Push **Next>**
82. Push the + button
83. Choose `Cycle 0 Loadcase 3 Acceleration`
84. Push **Next>**
85. Choose `Grid 38431`

    This is the rigid Payload CG grid
86. Push the **Translation 3** radio button
87. Push the **Magnitude** radio button
88. Push **Next>**

    The Frequency Response Plot will appear. Notice that the peak is approximately 15000 in/sec**2 which is about 38 G's acceleration of the payload CG at about 16 Hz.
89. Push the **Finish** button
90. Right click on the graph
91. Push the **Save Image...** button
92. Enter the name `FRDSG006_1_ACCE`

93. Push the **Save** button
94. Right click on the graph
95. Push the **Export Curve Data...** button
96. Enter the name `FRDSG006_1_ACCE`
97. Push the **Save Image...** button
98. Push the **New Freq. Resp. Plot** button
99. Push the **Choose Dynamic Component** radio button
100. Push **Next>**
101. Push the + button
102. Choose `Cycle 0 Loadcase 3 Displacement`
103. Push **Next>**
104. Choose all three `Grid 4929, Grid 38431,` and `Grid 40001`

    The displacement at 40001 T3 is the rattle space relative displacement between grids 4929 T3 and 38431 T3.

105. Push the **Translation 3** radio button
106. Push the **Magnitude** radio button
107. Push **Next>**

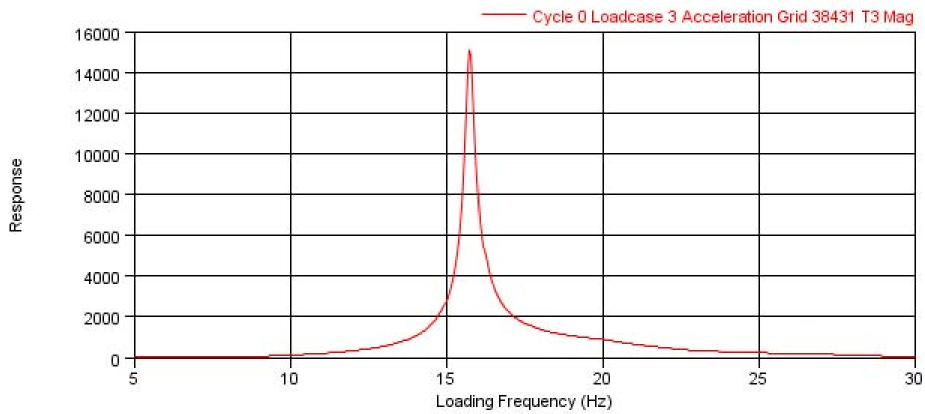
    The Frequency Response Plot will appear. Notice that the rattle space peak is approximately 0.55 inches at about 16 Hz.

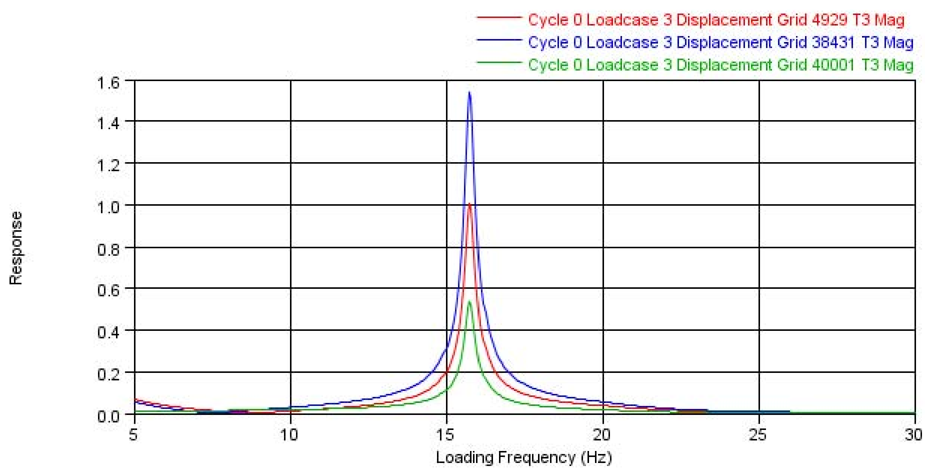
108. Push the **Finish** button
109. Right click on the graph
110. Push the **Save Image...** button
111. Enter the name `FRDSG006_1_DISP`
112. Push the **Up** button on the DSG window
113. Push the **Deform Mesh/Color Mesh** button
114. Push the **Oscillate** radio button, and examine the mode shapes

    Mode 2 at 15.77 Hz is the first bending in the Z direction. This is the mode that is excited by the payload fairing pressure load at 15.77 Hz. Compare the rattle space displacement frequency response plot to this mode shape. Does it make sense? Since the fairing and the Payload motions are in phase the relative peak turns out to be the smaller of the three peaks.

115. Push the **Up** button

The Following two frequency response plots show the expected response of the LV/PL FEM:





Quit Design Studio

116. From the main menu bar, select **File** → **Quit**
117. Push the **Don't Save** button

10.6.2 Part 2

In this part, topometry optimization is carried out to minimize the dynamic accelerations of the CG of the payload. The optimization example below shows how just the components that attach the payload to the launch vehicle can be designed to tune the interface for minimal destructive response.

If you do not have the `FRDSG006_2.dat` file generated in part 1, copy the file `FRDSG006_1_ref.dat` to `FRDSG006_2.dat`

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `FRDSG006_2.dat`

Create the Design Variables

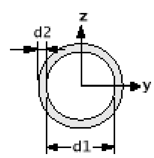
3. Select the **Design** tab
4. From the category chooser, select **Design Variables**
5. Push the **New Design Variable** button
6. For **Name**, enter `T3`

   Design variable T3 will represent the thickness of the payload adaptor shells PID=3
7. Make sure the **Independent Design Variable** radio button is selected
8. Push the **Next>** button
9. Type in the following for design variable **T3**:

Initial Value:	0.75
Lower bound:	0.5
Upper bound:	1.5

10. Push the **Finish** button

    The hollow tubes (PBARL PID=23) have two dimensions that will be designed



11. Repeat steps 5-10 for new design variable **P23 d1**:

DV Name:	`P23 d1`
Initial Value:	`3.0`
Lower bound:	`0.5`
Upper bound:	`3.0`

12. Repeat steps 5-10 for new design variable **P23 d2**:

DV Name:	`P23 d2`
Initial Value:	`0.5`
Lower bound:	`0.2`
Upper bound:	`0.6`

Create the Designable Regions

13. From the **Design** category chooser, select **Sizing**
14. Select PSHELL 3
15. Push the **Modify Sizing Design** button
16. For **Thickness,** select `T3 I=0.75`
17. Push the **Finish** button
18. Select PBARL 23
19. Push the **Modify Sizing Design** button
20. For **d1,** select `P23 d1 I=3.0`
21. For **d2,** select `P23 d2 I=0.5`
22. Push the **Finish** button

    Verify that a hammer icon appears next to the designed properties, and 2 sizing regions are defined

Define the Topometry Sizing Regions

23. Select the **Design** tab
24. From the category chooser, select **Topometry**
25. Select both PBARL 23 and PSHELL 3 groups
26. Push the **Modify Topometry Design** button

27. Examine the Topometry Attributes worksheet and push the **Finish** button
28. From the main menu bar, select **Genesis** → **Model Summary...**
29. Verify the pop up window shows 2 topometry designed groups and then push the **Close** button

Defining the beta method constraint equation

The *beta method* equation is: (Dynamic Response)/(Scale Factor) - *beta* < 0.0
The Scale Factor is calculated during the modal analysis process.
The Dynamic Response is calculated by Genesis for each design cycle.
The *beta* design variable is created.

30. From the **Design** category chooser, select **Synthetic Responses**
31. Push the **New Synthetic Response** button
32. Enter `beta constraint` for the **Name**
33. Make sure **User Function (DRESP2)** radio button is selected for the Type
34. Push **Next>**
35. Push the **+** button
36. Make sue the **Fundamental Response..** radio button is selected
37. Push **Next>**
38. Select **More Response Types...** from the Response Type radio button
39. Push **Next>**
40. Select **Dynamic Acceleration** from the Response Type radio button and select **From Modal (MACCE)** from the category chooser
41. Select **Magnitude** from the Select Dynamic Component
42. Push **Next>**
43. Push the **Select None** button
44. Enter `38431` in Select by Grid ID:
45. Select the **Translation 3** radio button
46. Push the **Add** button

    Verify that there is 1 grid selected.

47. Push **Next>**
48. Select the **MDYN** loadcase
49. Push **Next>**

You need to create a design variable (*beta*) and then come back where you are. Do not push **Finish** or **Cancel** at this point.

50. From the **Design** category chooser, select **Design Variables**
51. Push the **New Design Variable** button
52. Enter `beta` as the **Name**
53. Select **Independent Design Variable** radio button for Type
54. Push **Next>**
55. Enter `1.0` for **Initial Value**, `0.0` for **Lower Bound** and `1.0` for **Upper Bound**
56. Push the **Finish** button
57. From the **Design** category chooser, select **Synthetic Responses**

    You are back where you left off before design variable creation.

58. Push the + button
59. Select **Design Variable** from the Response Type radio button and select `beta`
60. Push **Next>**

    Now we have the two arguments required to state the synthetic response.

61. Rename **Arg1** to `ACCE`
62. Rename **Arg2** to `beta`
63. For the equation, enter `F = (ACCE/15094.0) - beta`

    Scale Factor = 15093.54 (value found during the analysis)

64. Push the **Finish** button

    Verify that the synthetic response you just created is listed

Defining the beta Design Constraint

65. From the **Design** category chooser, select **Constraints**
66. Push the **New Constraint** button from the Edit menu toolbar
67. Select **Synthetic Response** as Response Type.
68. Enter `0.0` for **Upper bound**
69. Push the **Next>** button
70. Select the `beta constraint` Synthetic Response
71. Push the **Next>** button
72. Select the MDYN loadcase

73. Push the **Finish** button

    Verify that the constraint you just created is listed

Defining the Rattle Space Design Constraint

74. From the **Design** category chooser, select **Constraints**
75. Push the **New Constraint** button from the Edit menu toolbar
76. For **Name:** type in `Rattle Space Constraint`
77. Select **More Response Types...** as Response Type.
78. Enter `1.0` for **Upper bound**
79. Push the **Next>** button
80. Select **Dynamic Displacement** from Additional Responses radio button and select **From Modal (MDISP)** from the category chooser
81. Select **Magnitude** from the Select Dynamic Component
82. Push **Next>**
83. Push the **Select None** button
84. Enter `40001` in Select by Grid ID:
85. Select the **Translation 3** radio button
86. Push the **Add** button

    Verify that there is 1 grid selected

87. Push **Next>**
88. Select the **MDYN** loadcase
89. Push **Finish**

    Verify that the constraint you just created is listed

Defining the beta method objective equation

90. From the **Design** category chooser, select **Synthetic Responses**
91. Push the **New Synthetic Response** button
92. Enter `beta objective` in the Name
93. Make sure **User Function (DRESP2)** radio button is selected for the Type
94. Push the **Next >** button
95. Push the + button

96. Push the **Design Variable** radio button and select `beta`
97. Push the **Next** > button
98. Rename **Arg1** to `beta`
99. For the equation, enter `F = beta`
100. Push the **Finish** button

    Verify that the synthetic response you just created is listed

Defining the Design Objective

101. From the **Design** category chooser, select **Objectives**
102. Push the **New Objective** button
103. Select **Synthetic Response** as Response Type.
104. Select **Min** as Objective definition
105. Push the **Next>** button
106. Select the `beta objective` Synthetic Response
107. Push the **Next>** button
108. Select the **MDYN** loadcase
109. Push the **Finish** button

    Verify that the objective you just created is listed

Clearing the Selection

110. Right click the Viewport, Select **Clear** → **All**

Setup the Genesis Options

111. From the main menu bar, select **Genesis** → **Options...**
112. In the Output Control tab, select **Design Output: First & Last**
113. In the Design Control tab, select **Max Design Cycles: 75**
114. In the Design Control tab, select **Advanced**
115. In the Methods tab, select **More Complex** option for **MDISP,MVELO,MACCE Appr.(MODAPP)**
116. Push the **Close** button
117. Push the **Apply** button

Save the Design Studio database

118. From the main menu bar, select **File** → **Save**

Optimize the structure using Genesis

119. From the main menu bar, select **Genesis** → **Optimize**
120. Study the **Genesis Console Output**
121. From the Genesis Console Output window push the **Import Post...** button
122. Select all the punched output files and push the **Import** button; when done, push the **Close** button

Post-Processing the Results

123. Select the **Post** tab
124. Push the **Deform Mesh/Color Mesh** button
125. Push the **Oscillate** radio button, and examine the mode shapes

    Notice that the modes for the final design cycle have changed considerably. Our design objective was to minimize the PL CG acceleration for the T3 direction, but as the modes change there may be acceleration in the T2 direction also. For this reason we will also plot the T2 acceleration along with the T3 acceleration in the following frequency response plots.

126. Push the **Up** button
127. Push the **Freq. Resp. Plot** button
128. Push the **New Freq. Resp. Plot** button
129. Push the **Choose Dynamic Component** radio button
130. Push **Next>**
131. Push the + button
132. Choose both `Loadcase 3 Acceleration` for the first and last design cycles
133. Push **Next>**
134. Choose `Grid 38431`

    This is the rigid Payload CG grid

135. Push the **Translation 3** radio button
136. Push the **Magnitude** radio button
137. Push **Next>**

    The Frequency Response Plot will appear. Notice the single peak for design cycle 0 has been split into multiple smaller peaks for the T3 acceleration response.

138. Push the + button
139. Choose both `Loadcase 3 Acceleration` for the first and last design cycles
140. Push **Next>**
141. Choose `Grid 38431`

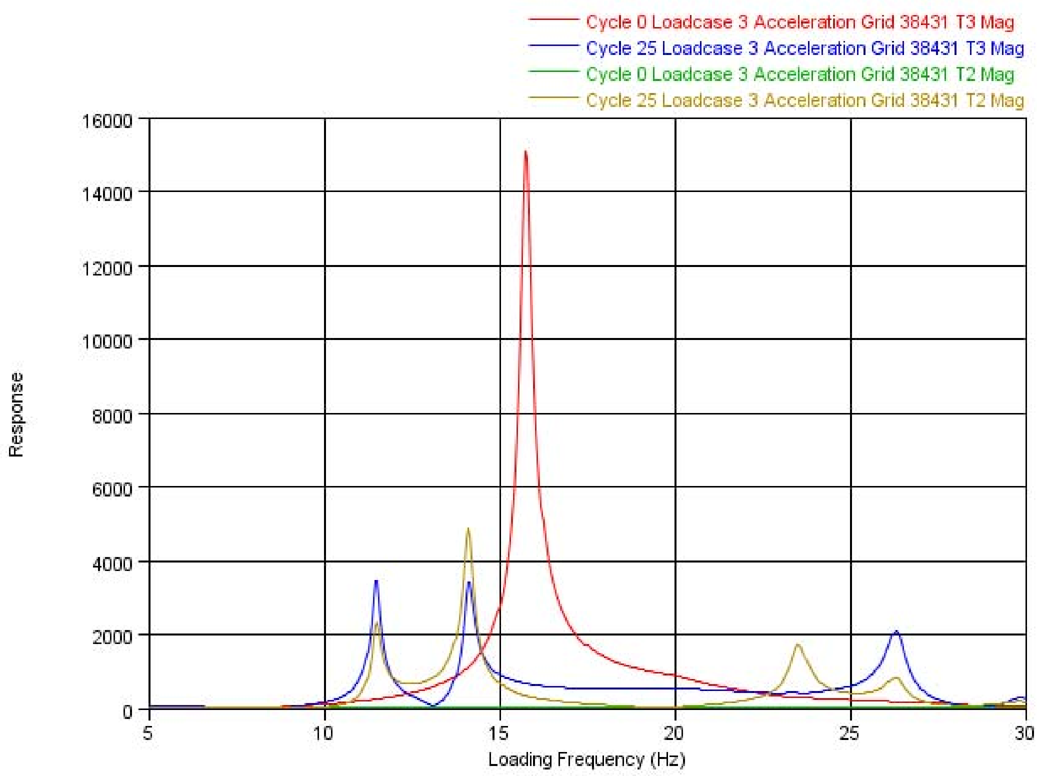
    This is the rigid Payload CG grid

142. Push the **Translation 2** radio button
143. Push the **Magnitude** radio button

    Push **Next>**

144. Push the **Finish** button

    The PL CG acceleration frequency response plot should look like the following plot:



Notice how the peak payload CG acceleration for the final design cycle is now in the T2 direction at about 10 G's. The Genesis optimization has tuned the LV/PL interface structure so that the pure T3 system mode at 15.77 Hz. for design cycle 0 has been split into multiple modes with both T2 and T3 participation neither of which resonate at 38 G's for the applied loading.

It should be noted that the T2 acceleration response at the PL CG was not constrained for this optimization, but it could have been.

145. Push the **New Freq. Resp. Plot** button
146. Push the **Choose Dynamic Component** radio button
147. Push **Next>**
148. Push the **+** button
149. Choose both `Loadcase 3 Displacement` for the first and last design cycles
150. Push **Next>**
151. Choose just `Grid 40001`

    The displacement at 40001 T3 is the rattle space relative displacement between grids 4929 T3 and 38431 T3. This has only been defined for the T3 displacements.

152. Push the **Translation 3** radio button
153. Push the **Magnitude** radio button
154. Push **Next>**

    The rattle space Frequency Response Plot for T3 will appear. Notice that the rattle space peak has not violated the 1.0 inch design constraint for T3. However, because the final design now shows some PL CG acceleration in the T2 direction it would have been prudent to include T2 in the rattle space calculation as well.

155. Push the **Finish** button
156. Push the **Up** button

Post-Processing the Topometry Results

157. Push the **Deform Mesh/Color Mesh** button
158. From the Color Mesh, select the PBARL d1 dataset for first design cycle
159. Push the **Options...** button
160. Select the **Hide Elements With No Value** checkbox
161. Push the **Close** button

    Notice the color plot of the property value for each of the PBARs

162. From the Color Mesh, select the PBARL d1 dataset for last design cycle
163. Click on an element in the Viewport to view the corresponding property value of the element in the Design Studio Messages window
164. Study the changes in design between the first and last design cycles
165. Similarly study the changes for PBARL d2 dataset

166. Select the PSHELL thickness for the first design cycle
167. Select the PSHELL thickness for the last design cycle
168. From the listbox near options, change from **Value** to **Value Change** to plot the change during the optimization
169. Push the **Up** button

Quit Design Studio

170. From the main menu bar, select **File** → **Quit**
171. Push the **Don't Save** button

CHAPTER 11

Modal Test-FEA Correlation with Design Optimization

- Compare Test Modes to Analytical Modes
- Test Data Correlation with FEM Using Sizing Optimization
- Add the Exact Target Frequencies to the Design Objective
- Adding Design Constraints to Tune the Airfoil Spar Stiffness Profile

11.1 Compare Test Modes to Analytical Modes

Introduction

The correlation of an FEM with Ground Vibration Test (GVT) data is often a laborious task generally performed manually. The task is usually composed of sequential steps including the comparison of analytic to test frequencies and mode shapes, an assessment of what parameter(s) need to change and by how much, making these changes by manually editing the bulk data entries, and rerunning the FEM analysis. This iterative process continues as the analyst works to tune the FEM, essentially one mode at a time. Once the first target mode has been tuned the analyst then repeats the tuning process for the second target mode trying not to obliterate the correlation obtained for the first mode.

These examples outline how Genesis and Design Studio may be used to reduce some of the difficulties associated with tuning a model by utilizing design optimization. The correlation is posed as a structural sizing optimization problem. The test frequencies are assigned as design constraints and element stiffness properties are assigned as design variables. The design objective is to minimize the differences between the test and analytical mode shapes by varying the beam properties subject to the frequency constraints. Once the optimization problem is set up Genesis optimization will correlate all target modes simultaneously.

This example is similar to the process outlined in the paper: Mundt, C. and Quinn, G., "Test-Analysis Correlation with Design Optimization," presented at the Aerospace Testing Expo 2005 North America, Long Beach Convention Center, Long Beach, CA, November 8-10, 2005. The FEM and test data used here is not the same as the actual data the paper describes.

The purpose of example 1 is to show how the mode shapes and frequencies acquired during a GVT can be brought into Design Studio and visually compared to the analytical mode shapes and frequencies calculated in Genesis. This example will first show how to create a modal load case for the FEM that simulates the GVT set up. Then, this example will describe a method that brings in the GVT data into Design Studio. Finally, the two sets of mode shapes and frequencies will be compared in order to identify how well our initial FEM matches the modal test data.

Normal Modes Analysis:

No restraints, Omit Rigid Body Modes, Display First 20 Elastic Modes

GVT Data:

Convert to Punch File Format and Display 7 Target Modes.

Example ID

`MCDSG001`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
MCDSG001.dat	Input data	Provided: Contains the finite element mesh.
MCDSG001_dsg.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus the data in MCDSG001_1.dat.
MCDSG001_dsg00.pch	Punch Output data	Generated using Genesis within Design Studio. This file contains the mode shapes requested as analysis outputs
MCDSG001_GVT.pch	Punch File	Provided. Contains the mode shape information obtained by testing in punch format.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `MCDSG001.dat`

Create the Eigenvalue Method

You will create a Lanczos method to calculate the first 20 elastic modes.

3. Select the **Analysis** tab
4. From the category chooser, select **Eigenvalue Methods**
5. Push the **New Eigenvalue Method** button from the Edit menu toolbar
6. Enter `Lanczos Method` for the **Name**
7. Push the **Lanczos** button
8. Enter `2.0` for the **V1**

   V2 is chosen as a frequency that is greater than the sixth rigid body mode and less than the frequency of the first elastic mode. This way only the elastic normal modes of the FEM is considered when comparisons are made to the GVT data.

9. For **V2** delete the default value and leave blank
10. Enter `20` for the **Number of Modes**

11. Push the **Finish** button

Create the Loadcase

The eigenvalue method you just created is not used unless it is in a loadcase. You will now create one loadcase that uses the previously created eigenvalue method. Since the normal modes are to be calculated for a free unrestrained structure there are no boundary conditions included in the loadcase.

12. Select the **Analysis** tab
13. From the category chooser, select **Loadcases**
14. Select the default loadcase
15. From the Edit menu toolbar, select the **Delete Loadcase** button

    Design Studio uses a static loadcase as the default which is not needed for this example.

16. Push the **New Loadcase** button from the Edit menu toolbar
17. Enter `Normal Modes` for the **Name**
18. Push the **Normal Modes** button
19. Push the **Next>** button
20. Push the **Next>** button

    No boundary conditions are required for this loadcase.

21. For **Eigenvalue Method**: Select `Lanczos Method`
22. Push the **Next>** button
23. For **Displacement**: Select `Post`
24. Push the **Finish** button

    Verify that the loadcase you just created is in the list of Loadcases.
    Note that there is a **FREQ** label on the left part of the line item. FREQ refers to modal load-case.

Perform the Modal Analysis Using Genesis

25. From the main menu bar, select **Genesis→Single Analysis**
26. Study the **Genesis Console Output** window

Post-Processing the Results (Modal Analysis Results)

27. From the **Genesis Console Output** window, select the **Import Post..** button
28. Select `MCDSG001_dsg00.pch` punch file
29. Push the **Import** button

30. When done select the **Close** button from the **Genesis Console Output** window
31. Select the **Post** tab
32. Push the **Deform Mesh/Color Mesh** button
33. Select a Mode Result for design cycle 0

    Study the results. Verify that there are no rigid body modes. Only the first 7 elastic modes are considered target modes that we wish to match with test data (both frequencies and mode shapes). The measured GVT test mode shapes may, or may not correspond to the first seven analytical modes. Therefore it is good practice to solve for more modes than are required and then identify which ones correspond to the measured data. This is why we solved for 20 modes in the Genesis analysis.

Import the GVT Test Data (Mode Shapes and Frequencies)

The measured mode shape data and frequencies for the first seven modes must be converted to punch or output2 format for Design Studio to recognize it. The normalized displacement data for the airfoil was gathered at 38 accelerometer locations in the X, Y, and Z directions for seven modes. Microsoft Excel was used to convert this data into Genesis punch format so that file `MCDSG001_GVT.pch` can be imported into Design Studio.

34. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
35. For Files of Type, select **Punch/Output2 Files (*.pch),etc.**
36. Select the `MCDSG001_GVT.pch` file
37. Push the **Open** button

    The test modes from the punch file `MCDSG001_GVT.pch` are imported into Design Studio and appended onto the end of the first 20 analytical modes. The last seven modes are the GVT modes.

38. Select one of the test mode shapes from the last seven listed
39. Push the **Oscillate** button

    Study the results. Notice that the test data is only defined for the accelerometer locations on the leading and trailing edges of the airfoil. The spar grid points do not have displacements defined for them so the test mode shapes show zero displacement for these grids. This model has PLOTEL elements defined to outline only the accelerometer grids in order to plot the test modes in a visually uncluttered manner.

40. Select the **Display** tab
41. Push the **Show/Hide Groups** button
42. Push the **Hide All** button
43. Select the **PLOTEL** Property ID
44. Push the **Up** button

45. Select the **Post** tab

Now the analysis modes and the test modes may be studied and identified. Upon inspection the FEM model does not simulate the test results adequately. The correspondence of the first seven target modes is shown below:

ORIGINAL ANALYTICAL DATA			TEST MODE DATA		
Mode	**Freq**	**Description**	**Mode**	**Freq**	**Description**
1	20.60	1st Bending SYM	1	20.00	1st Bending SYM
2	27.54	2nd Bending ASYM	2	27.06	2nd Bending ASYM
3	49.84	3rd Bending SYM	3	49.13	3rd Bending SYM
4	55.90	In Plane Fore & Aft	4	55.09	In Plane Fore & Aft
5	56.42	1st Torsion ASYM	5	58.84	1st Torsion ASYM
6	78.35	2nd Torsion SYM	6	75.74	4th bending ASYM
7	81.14	4th bending ASYM	7	87.64	2nd Torsion SYM

The first five mode shapes correspond one-to-one and their frequencies are within 2Hz or better. However, the frequencies of the second torsion and fourth bending modes are such that these modes have changed places in their respective sequence.

The two sets of data (analysis versus test modes) could also be compared side-by-side by opening Design Studio twice and importing the analysis and test responses separately.

The next section will demonstrate how design optimization can be used to correlate the FEM with the test data so that a test verified FEM can be produced.

Quit Design Studio

46. From the main menu bar, select **File** → **Quit**

47. Push the **Don’t Save** button

11.2 Test Data Correlation with FEM Using Sizing Optimization

Introduction

The purpose of this example is to show how design optimization is used to modify the I1, I2, and J properties of the horizontal stabilizer spar so that the FEM mode shapes and frequencies better match the GVT test data. Structural tests usually generate massive amounts of data, and this small example is no exception. It will be shown how to enter a sampling of the test data into Design Studio from data files that contain most, but not all of the design data.

Objective function of the problem:

The design objective is defined as the sum of the square of the differences between all analytical and measured mode shapes

$$MinimizeObj = \sum (U_{Calc} - U_{Test})^2$$

Where U = normalized displacements at accelerometer locations.

Subject to:

All target modal frequencies should match $F_{Calc} = F_{Test}$

Designable region:

All spar bar elements in the structure...I1, I2, & J

Analysis problem:

Normal modes

Free unrestrained structure

Example ID

`MCDSG002`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
MCDSG002_1.dat	Input data	Provided: Contains the finite element mesh along with the eigenvalue loadcase

MCDSG002_2.dat	Input data	Provided: Contains the FE model along with most of the design variable and sizing data defined.
MCDSG002_3.dat	Input data	Provided: Contains the FE model along with the design variable and sizing data defined.Also contains most of the objective and constraints definition.
MCDSG002_3_dsg.dat	Input data	Generated by Design Studio. This file contains all the data created and ready for optimization.
MCDSG002_3_dsg00.pch	Punch Output data	Generated using Genesis within Design Studio. This file contains the mode shapes requested as analysis outputs
MCDSG002_3_ref.dat	Input data	Provided: Reference file with all the data needed for the problem. Similar to `MCDSG002_3_dsg.dat` file

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `MCDSG002_1.dat`

Modify the Eigenvalue Method

The calculated eigenvectors must be consistent with the measured test data or comparison within the objective function would be impossible. The measured eigenvectors are extracted from accelerometer and are point normalized to 1.0 at the grid component with the largest deflection for each mode. The analytical shapes should also be point normalized in exactly this same manner. This step shows how to modify the eigenvalue method to match the point normalization scheme of the test data. Don't forget that Example 1 shows that analytical mode 6 (2nd torsion) is test mode 7, and analytical mode 7 (4th bending) is test mode 6. This mode identification step is very important to correlate the FEM.

3. Select the **Analysis** tab
4. From the category chooser, select **Eigenvalue Methods**
5. Select `Lanczos Method` from the list
6. Push the **Modify Eigenvalue Method** button from the Edit Menu toolbar
7. For **Normalization**, select `Point`
8. Push the **Next>** button
9. Push the + button at the lower left portion of the window

    According to the Excel table of test data shown below, mode one should be point normalized to component 3 of grid 31001

GVT data
Normalized to Positive 1.0
Airfoil mode shapes

		TEST	GENESIS	
		Mode No	Mode No	Freq(Hz)
		1	1	20.00
	Node Id	X	Y	Z
1	31001	0.009	-0.001	1.000
2	31002	0.008	0.002	0.760
3	31003	0.008	0.002	0.391
*	*	*	*	*
*	*	*	*	*
*	*	*	*	*
36	131036	0.007	-0.002	-0.611
37	131037	0.006	-0.002	-0.568
38	131038	0.006	-0.002	-0.462
	MAX=	0.091	0.002	1.000
	MIN=	-0.251	-0.003	-0.616
	Norm. Grid			31001
	Component	1	2	3

10. For **Grid ID**: type in `31001`
11. For **Component**: select `3`
12. Push the + button at the lower left portion of the window
13. For **Mode**: type in `2`
14. For **Grid ID**: type in `131031`
15. For **Component**: select `3`
16. Repeat steps 12-15 for modes 3-7 using the data in the table below.

Make sure the component for mode 4 is 1 as shown in the table below

Mode	Grid ID	Component
3	31031	3
4	31901	1
5	131034	3
6	131033	3
7	131031	3

17. Push the **Finish** button

Defining the Design Variables

In order to better match the test mode shapes and frequencies the airfoil spar stiffness will be modified. Design variables will be assigned to each of 16 bar elements for I1, I2, and J. These 48 design variables will design the +Y and -Y spars in a symmetric manner. The design variables are scaled in this example with the original values of I1, I2, and J. This is done for convenience so that the user can see at a glance the fractional change for each bar during optimization.

Rather than take the time to enter all data here this tutorial will read in a file that contains all but three DVAR and DVPROP1 entries. The user will complete DVAR and DVPROP1 entries for PBAR 3075 by following the steps below.

Start a New Design Studio Database

18. From the main menu bar, select **File** → **New**
19. Push the **Don't Save** button

Import a New Input Data File

20. Import the Genesis data file `MCDSG002_2.dat`

Defining the Independent Design Variables for PID=3075

21. Select the **Design** tab
22. From the category chooser, select **Design Variables**

    Examine the list of design variables. The prefix "**I**" and "**S**" designates independent and synthetic design variables respectively.

23. Push the **New Design Variable** button from the Edit menu toolbar
24. Enter `P3075 I1` for the **Name**
25. Check the **Independent Design Variable** radio button
26. Push the **Next** button
27. For **Lower Bound**, enter `0.5`

    Accept the initial value of 1.0 and upper bound of 2.0. All design variables will be set up so that the bar stiffness properties can all vary from one half to twice their initial values.

28. Push the **Finish** button

    Independent Design Variable P3075 I1 has been defined. The asterisk preceding the "I" means this design variable has not been referenced to the model yet.

29. Repeat steps 23 to 28 to create two Design Variables using the following table

Design Variable Name	Initial Value	Lower Bound	Upper Bound
P3075 I2	1.0	0.5	2.0
P3075 J	1.0	0.5	2.0

Defining the Synthetic Design Variables for PID=3075

The 3 independent design variables above will be referenced to the following 3 synthetic design variables defined below.

30. Push the **New Design Variable** button from the Edit menu toolbar
31. Enter `DVP1 P3075 I1` for the **Name**
32. Push the **Synthetic Design Variable** button
33. Push the **Next>** button
34. From the Master Variable list, select `P3075 I1 I=1.0`
35. For Coefficient, enter `6.00`

    For PID=3075 I1=6.00, I2=148.96, and J=8.95. The three synthetic design variables will scale the independent design variables by their initial stiffness properties.

36. Push the **Finish** button

    Synthetic Design Variable P3075 I1 has been defined

37. Push the **New Design Variable** button from the Edit menu toolbar
38. Enter `DVP1 P3075 I2` for the **Name**
39. Push the **Next>** button
40. From the Master Variable list, select `P3075 I2 I=1.0`
41. For Coefficient, enter `148.96`
42. Push the **Finish** button

    Synthetic Design Variable P3075 I2 has been defined

43. Push the **New Design Variable** button from the Edit menu toolbar
44. Enter `DVP1 P3075 J` for the **Name**
45. Push the **Next>** button
46. From the Master Variable, select `P3075 J I=1.0`
47. For Coefficient, enter `8.95`

48. Push the **Finish** button

    Synthetic Design Variable P3075 J has been defined

Associate the Bar Properties with the Synthetic Design Variables

The 3 synthetic design variables above will be referenced to PBAR 3075 I1, I2, & J

49. From the Category Chooser, select **Sizing**

    Notice that PBAR 3075 does not have a hammer icon next to it meaning it has not yet been designed.

50. Select PBAR 3075
51. Push the **Modify Sizing Design** button from the Edit Menu toolbar
52. For **I1**, select `DVP1 P3075 I1`
53. For **I2**, select `DVP1 P3075 I2`
54. For **J**, select `DVP1 P3075 J`
55. Push the **Finish** button

    Notice the hammer icon next to PBAR 3075 designating it as a designable property

Defining the Design Objective

The design objective for this problem is to minimize the square of the differences between the FEM and the measured test data of the eigenvector response for all target modes. For this problem the accelerometer responses show a predominant response in either the X, Y, or Z direction that is different for each target mode. For instance, the in-plane bending mode 4 is predominant in X while the rest of the modes are predominant in Z. To tune this model we need only match the predominant model response component and disregard the off-component response. The test data to match for target mode 6 is shown highlighted in dark green below.

This tutorial will read in a new file that has all but the last three response entries for FEM mode seven. Initially, FEM mode 7 corresponds to test mode 6 so the data that will be entered is: Grid 131036 Z=0.17569, Grid 131037 Z=0.18713, & Grid 131038 Z=0.07118.

GVT data
Normalized to Positive 1.0
Airfoil mode shapes

		TEST	GENESIS	
		Mode No	Mode No	Freq(Hz)
		6	7	75.74
	Node Id	X	Y	Z
1	31001	-0.00284	0.00199	-0.69180
2	31002	-0.00255	0.00096	-0.09851
3	31003	-0.00135	0.00087	0.45328
*	*	*	*	*
*	*	*	*	*
*	*	*	*	*
*	*	*	*	*
36	131036	-0.00007	0.00105	0.17569
37	131037	-0.00003	0.00084	0.18713
38	131038	0.00014	0.00072	0.07118
	MAX=	0.17585	0.00292	1.00000
	MIN=	-0.04973	0.00072	-0.96573
	Norm. Grid			131031
	Component	1	2	3

Start a New Design Studio Database

56. From the main menu bar, select **File** → **New**
57. Push the **Don't Save** button

Import a New Input Data File

58. Import the Genesis data file `MCDSG002_3.dat`

Input 3 Entries to Complete the DMATCH2 Objective Function

59. Select the **Design** tab
60. From the category chooser, select **Objectives**

    Examine the data for the objective function.

61. Push the **New Objective** button from the Edit menu toolbar
62. Enter `Obj264` for the **Name**

63. Select the **More Response Types...** radio button
64. Select the **Target** radio button
65. For **Target**, enter: `0.17569`
66. For **Weight**, enter: `-1.0`
67. Push the **Next>** button
68. Select the **Mode Shape Number** radio button and enter `7` for Mode Shape Number
69. Push the **Next>** button
70. For **Select by Grid ID**, enter: `131036`
71. Push the **Add** button

    Verify that "1 grid selected" is shown at the bottom of window and that a single grid is highlighted on the trailing edge of the airfoil. You may need to hide the CONM2 elements to see the selected grid.

72. Push the **Translation 3** button
73. Push the **Next>** button
74. Select the loadcase **FREQ ID=2**
75. Push the **Finish** button

    "Obj264" has been created.

76. Right hand click in the display window and select: **Clear** → **Grid Selection**

    This deselects grid 131036

77. Repeat the steps from 61 to 76 to create other objective using the data in the following table

Name	Target	Weight	Mode Shape Number	Grid ID
`Obj265`	`0.18713`	`-1.0`	`7`	`131037`
`Obj266`	`0.07118`	`-1.0`	`7`	`131038`

Defining the Design Constraints

The design constraints for matching the GVT data will be the lower and upper bounds on the modal frequencies for each target mode. For this case the lower bound will be a tenth of a Hertz below the target frequency, and the upper bound will be a tenth of a Hertz above the target frequency as shown below.

Mode Description	Analytical Mode Sequence	Test Mode Sequence	Target Frequency (Hz.)	Freq Constraints LB	UB
1st Bending SYM	1	1	20.00	19.90	20.10
2nd Bending ASYM	2	2	27.06	26.96	27.16
3rd Bending SYM	3	3	49.13	49.03	49.23
In Plane Fore & Aft	4	4	55.09	54.99	55.19
1st Torsion ASYM	5	5	58.84	58.74	58.94
2nd Torsion SYM	6	7	87.64	87.54	87.74
4th bending ASYM	7	6	75.74	75.64	75.84

78. Select the **Design** tab
79. From the category chooser, select **Constraints**
80. Push the **New Constraint** button from the Edit menu toolbar
81. Enter `Mode1 LB` for the **Name**
82. Push the **Frequency Mode Number** button
83. For Frequency Mode Number, enter `1`
84. For **Lower Bound**, enter `19.90`

    Leave Upper Bound blank
85. Push the **Next>** button
86. Select the existing loadcase **FREQ ID=2**
87. Push the **Finish** button

    The lower bound frequency constraint for mode 1 has been created.
88. Push the **New Constraint** button from the Edit menu toolbar
89. Enter `Mode1 UB` for the **Name**
90. Push the **Frequency Mode Number** button
91. For Frequency Mode Number, enter `1`
92. For Upper Bound, enter `20.10`

    Leave Lower Bound blank
93. Push the **Next>** button

94. Select the existing loadcase **FREQ ID=2**

95. Push the **Finish** button

    The upper bound frequency constraint for mode 1 has been created.

96. Repeat the steps for the lower and upper bound design constraints for modes 2-7 as described in the table below.

Name	Mode Number	Bound
Mode2 LB	2	26.96
Mode2 UB	2	27.16
Mode3 LB	3	49.03
Mode3 UB	3	49.23
Mode4 LB	4	54.99
Mode4 UB	4	55.19
Mode5 LB	5	58.74
Mode5 UB	5	58.94
Mode6 LB	6	87.54
Mode6 UB	6	87.74
Mode7 LB	7	75.64
Mode7 UB	7	75.84

97. From the main menu bar, select **Genesis**→ **Model Summary**

    The model statistics should include:
    266 design objectives,
    14 design constraints,
    48 design variables, and
    16 sizing-designed groups.

98. Push the **Close** button on the Design Studio Model Statistics viewport

Set The Eigensolution to Track the Modes

When changing the properties of an FEM during the normal modes analysis the sequence of the modes sometimes will change as modal frequencies increase and/or decrease. In fact, for this particular design optimization problem we want modes 6 and 7 to change their sequence to better match the test data. The Genesis parameter MODTRK is used to do this.

99. Select the **Analysis** tab

100. From the category chooser, select **Loadcases**
101. Select the FREQ ID=2 loadcase
102. Push the **Modify Loadcase** button from the Edit Menu toolbar
103. Push the **Next>** button
104. Push the **Next>** button
105. For Mode Tracking, select `All`
106. Push the **Finish** button

Set the Optimizer to BIGDOT

The default optimization algorithm in Genesis for sizing optimization is DOT. This problem works better using the BIGDOT optimization algorithm.

107. From the main menu bar, select **Genesis**→ **Options**
108. From the Genesis Options menu bar, select **Design Control**
109. Push the **Optimizer** checkbox
110. For Optimizer, select **BIGDOT**
111. Push the **Apply** button

Set the Post-Processing Output Control to Only Create Punch Files for the First and Last Design Cycles

112. From the main menu bar, select **Genesis**→ **Options**
113. From the Genesis Options menu bar, select **Output Control**
114. Push the **Analysis Output** checkbox
115. For Analysis Output, select **First & Last**
116. Push the **Design Output** checkbox
117. For Design Output, select **First & Last**
118. Push the **Apply** button

Set the Post-Processing File Control to Output an Updated Input File for the Last Design Cycle

119. From the main menu bar, select **Genesis**→ **Options**
120. From the Genesis Options menu bar, select **File Control**

121. Push the **Updated Input File** checkbox
122. For Updated Input File, select **Last Cycle**
123. Push the **Apply** button

Request for Element Sizing data (OPOST) File

124. From the main menu bar, select **Genesis**→ **Options**
125. From the Genesis Options menu bar, select **File Control**
126. Push the **Element Sizing File** checkbox and select **Create** option
127. Push the **Apply** button

Optimize the FEM Using Genesis

128. From the main menu bar, select **Genesis**→ **Optimize**

    The Genesis console window will appear. Two design history charts will display the objective and the maximum constraint violation versus design cycle. These charts will update as the Genesis run progresses.

129. Study the **Design History** charts; when done, push the **Close** button
130. Study the **Genesis Console Output** window

Import the Post-Processing Files

131. From the **Genesis Console Output** window, select the **Import Post...** button
132. Select all the existing punch files (`MCDSG002_3_dsgxx.pch` and `MCDSG002_3_dsgOPOSTxx.pch`)
133. Push the **Import** button
134. When done, push the **Close** button in the **Genesis Console Output** window

Post-Processing the Results (Modal Displacement Results)

135. Select the **Post** tab
136. Push the **Manage Result Datasets** button

    We have imported the first 20 modes for 2 design cycles, but only wish to view the first 7 modes for Cycle 0 and the last design cycle.

137. Highlight the modes you wish to delete (keep modes 1-7 for the first and last design cycle); from the Edit menu toolbar, select the **Delete Result Set** button
138. Push the **Up** button

139. Push the **Deform Mesh/Color Mesh** button

140. Select **Cycle 0 Loadcase 2 Mode 1 Eigenvector** result

141. Push the **Oscillate** button

Compare mode 1 for the first and last design cycles. Make sure the mode shape is the same (1st bending SYM) and the frequency is closer to the constrained target frequency. Comparing modes 1-7 in this way shows:

Mode Description	Analytical Mode Sequence	Target Frequency (Hz.)	1st Design Frequency (Hz.)	Last Design Frequency (Hz.)	Freq Constraints LB	Freq Constraints UB
1st Bending SYM	1	20.00	20.60	20.04	19.90	20.10
2nd Bending ASYM	2	27.06	27.54	27.04	26.96	27.16
3rd Bending SYM	3	49.13	49.84	49.17	49.03	49.23
In Plane Fore & Aft	4	55.09	55.90	55.15	54.99	55.19
1st Torsion ASYM	5	58.84	56.42	58.76	58.74	58.94
2nd Torsion SYM	6	87.64	78.35	87.47	87.54	87.74
4th bending ASYM	7	75.74	81.14	75.87	75.64	75.84

Notice that all frequencies for the last design cycle lie between the lower and upper bound constraints except mode 7. It's frequency is 75.87 Hz which is above the upper bound constraint of 75.84 Hz. Genesis employs a tolerance of 0.03 on its constraints which allows the mode 7 constraint to be considered active at its UB+0.03, but not violated. Notice how the 2nd torsion mode 6 is now at a higher frequency than the 4th bending mode 7. Genesis tracked all the modes as their frequencies migrated to the constrained target frequencies.

Post-Processing the Sizing Results

142. From the Color Mesh, select any of the properties of the PBAR for Cycle 0

143. Push the **Options...** button

144. Select the **Hide Elements With No Value** checkbox

145. Push the **Close** button

    Notice the color plot of the property value for each of the PBARs

146. Study the results for last design cycle

147. Compare the property value between the first and last design cycles

148. Click on an element in the Viewport to view the corresponding property value of the element in the Design Studio Messages window

149. From the listbox near options, change from **Value** to **Value Change** to plot the change during the optimization
150. From the listbox near options, change from **Value Change** to **Value/Original** to plot the ratio of the change during the optimization
151. Push the **Up** button

Quit Design Studio

152. From the main menu bar, select **File** → **Quit**
153. Push the **Don’t Save** button

11.3 Add the Exact Target Frequencies to the Design Objective

Introduction

As shown in the previous example the airfoil FEM has been optimized to better match the mode shapes and frequencies of the GVT modal test data. The design objective in the previous example was to minimize the sum of the square of the differences between the calculated and measured modal displacements at the accelerometer locations for all seven target modes. The design constraints were to match the modal frequencies within a ± 0.1 Hz. tolerance.

Depending on how well an initial FEM actually simulates the target modes the upper and lower bounds for the frequency constraints may have to employ larger tolerances. This is OK to help tune the FEM to the test data, but eventually it may be desirable to zero in on FEM modal frequencies that better match the measured modal frequencies.

There is no reason why we cannot also put the modal frequencies into the design objective function as well as the design constraints. Sometimes it is better to allow Genesis to first satisfy frequency constraints with a large tolerance, and then minimize the design objective function that also includes the differences between the calculated and measured target frequencies.

This example shows how to add the modal frequency match to the design objective function of the optimization set up defined in Example 2.

Mode Description	Analytical Mode Sequence	Target Frequency (Hz.)	1st Design Frequency (Hz.)	Last Design Frequency (Hz.)	Freq Constraints LB	UB
1st Bending SYM	1	20.00	20.60	20.04	19.90	20.10
2nd Bending ASYM	2	27.06	27.54	27.04	26.96	27.16
3rd Bending SYM	3	49.13	49.84	49.17	49.03	49.23
In Plane Fore & Aft	4	55.09	55.90	55.15	54.99	55.19
1st Torsion ASYM	5	58.84	56.42	58.76	58.74	58.94
2nd Torsion SYM	6	87.64	78.35	87.47	87.54	87.74
4th bending ASYM	7	75.74	81.14	75.87	75.64	75.84

Example ID

MCDSG003

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
MCDSG003.dat	Input data	Provided: Contains the finite element mesh along optimization data for sizing.
MCDSG003_dsg.dat	Input data	Generated by Design Studio. This file contains all the data created along with the data in MCDSG003.dat
MCDSG003_dsg00.pch	Punch Output data	Generated using Genesis within Design Studio. This file contains the mode shapes requested as analysis outputs
MCDSG003_ref.dat	Input data	Provided: Reference file with all the data for the problem. Similar to MCDSG003_dsg.dat file

Start Design Studio

1. Start **Design Studio**
2. Import the `MCDSG003.dat` file

Add the Target Frequencies to the Objective Function

3. Select the **Design** tab
4. From the category chooser, select **Objectives**

   Scroll to the bottom of the list. The last entry should be Obj266. We will add 7 more, one for each target frequency.
5. Push the **New Objective** button from the Edit menu toolbar
6. Push the **Frequency Mode Number** button
7. For Frequency Mode Number, enter `1`
8. Push the **Target** button
9. For **Target**, enter: `20.00`
10. For **Weight**, enter `-10.0`

    Weighting the frequency match by a factor of 10 will prioritize the eigenvalue match over the eigenvector match in this objective function. This also allows the test engineer to independently assign larger weighting factors to “problem” modes that are difficult to match.
11. Push the **Next>** button
12. Select `FREQ ID=2` loadcase
13. Push the **Finish** button

    “Obj267” has been created.

14. Repeat the above steps with the data in the table below to create the objectives for modes 2-7.

Mode Number	Target	Weight
2	27.06	-10.0
3	49.13	-10.0
4	55.09	-10.0
5	58.84	-10.0
6	87.64	-10.0
7	75.74	-10.0

Request for Element Sizing data (OPOST) File

15. From the main menu bar, select **Genesis**→ **Options**
16. From the Genesis Options menu bar, select **File Control**
17. Push the **Element Sizing File** checkbox and select **Create** option
18. Push the **Apply** button

Optimize the FEM Using Genesis

19. From the main menu bar, select **Genesis**→ **Optimize**

    The Genesis console window will appear. Two design history charts will display the objective and the maximum constraint violation versus design cycle. These charts will update as the Genesis run progresses.

20. Study the **Design History** charts; when done, push the **Close** button
21. Study the **Genesis Console Output** window

Import the Post-Processing Files

22. From the **Genesis Console Output** window, select the **Import Post...** button
23. Select all the existing punch files (`MCDSG003_dsgxx.pch` and `MCDSG003_dsgOPOSTxx.pch`)
24. Push the **Import** button
25. When done, push the **Close** button in the **Genesis Console Output** window

Post-Processing the Results (Modal Displacement Results)

26. Select the **Post** tab
27. Push the **Manage Result Datasets** button

    We have imported the first 20 modes for 2 design cycles, but only wish to view the first 7 modes for Cycle 0 and the last design cycle.

28. Highlight the modes you wish to delete (keep modes 1-7 for the first and last design cycle); from the Edit menu toolbar, select the **Delete Result Set** button
29. Push the **Up** button
30. Push the **Deform Mesh/Color Mesh** button
31. Select **Cycle 0 Loadcase 2 Mode 1 Eigenvector** result
32. Push the **Oscillate** button

Examine all the modes and frequencies. The table below shows how the designed frequencies now match the target frequencies much better.

Mode Description	Analytical Mode Sequence	Target Frequency (Hz.)	1st Design Frequency (Hz.)	Last Design Frequency (Hz.)	Freq Constraints LB	Freq Constraints UB
1st Bending SYM	1	20.00	20.60	20.00	19.90	20.10
2nd Bending ASYM	2	27.06	27.54	27.06	26.96	27.16
3rd Bending SYM	3	49.13	49.84	49.13	49.03	49.23
In Plane Fore & Aft	4	55.09	55.90	55.09	54.99	55.19
1st Torsion ASYM	5	58.84	56.42	58.84	58.74	58.94
2nd Torsion SYM	6	87.64	78.35	87.64	87.54	87.74
4th bending ASYM	7	75.74	81.14	75.74	75.64	75.84

Post-Processing the Sizing Results

33. From the Color Mesh, select any of the properties of the PBAR for last design cycle
34. Push the **Options...** button
35. Select the **Hide Elements With No Value** checkbox
36. Push the **Close** button

    Notice the color plot of the property value for each of the PBARs

37. Study the results for last design cycle
38. Click on an element in the Viewport to view the corresponding property value of the element in the Design Studio Messages window
39. From the listbox near options, change from **Value** to **Value Change** to plot the change during the optimization

40. Push the **Up** button

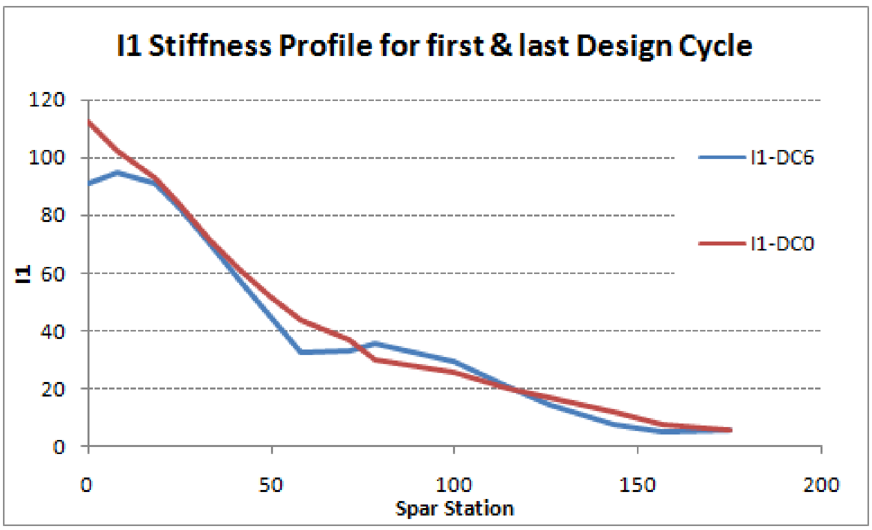
Quit Design Studio

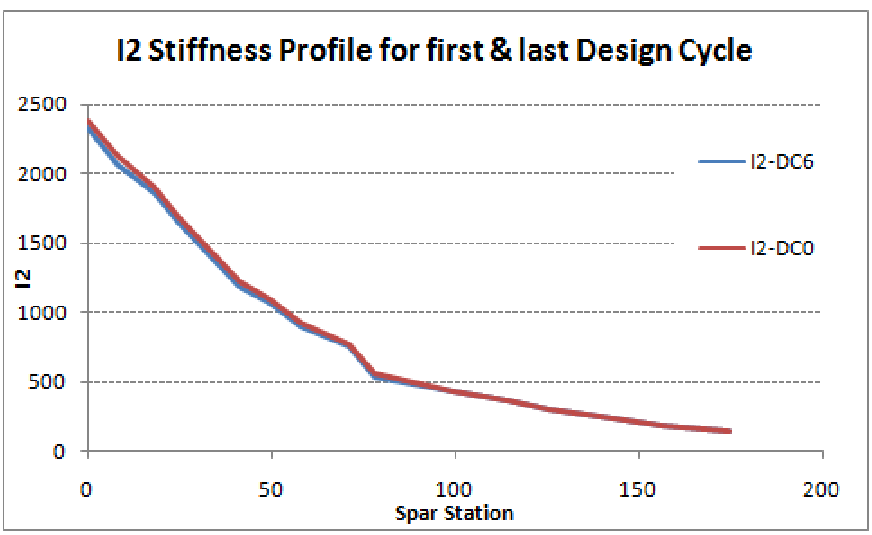
41. From the main menu bar, select **File** → **Quit**
42. Push the **Don’t Save** button

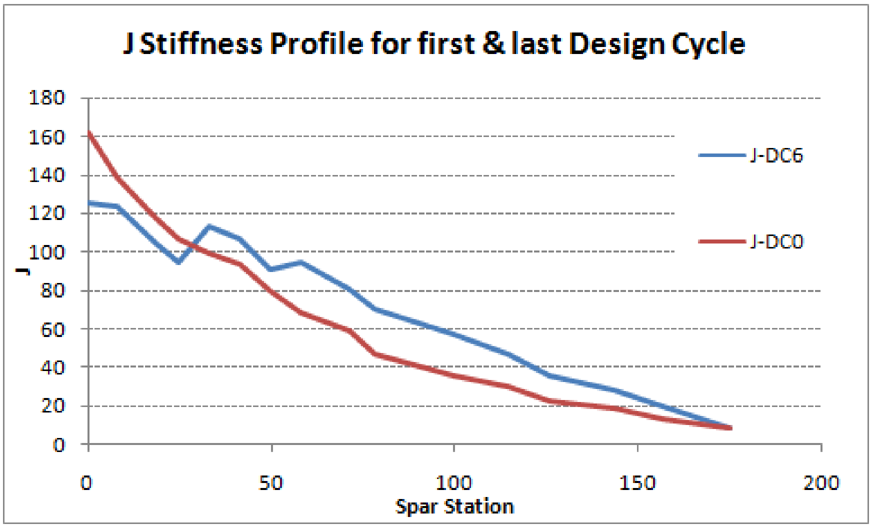
11.4 Adding Design Constraints to Tune the Airfoil Spar Stiffness Profile

Introduction

During the model correlation process it is possible that many different designs can satisfy the target frequency and mode shape goals. The previous examples have done this. However, test engineers must also consider the airfoil beam sections should resemble a tapered beam for the spar. For this particular model the root of the airfoil should have the stiffest properties for I1, I2, and J that gradually decrease towards the tip. This is called the airfoil stiffness profile and is plotted from the root to the tip for I1, I2, and J below:







For I1 and J notice how the stiffness profiles for the original data (in red) are greatest at the root (station 0) and always decrease when moving towards the tip. Once the model has been optimized to match the test mode shapes and frequencies this is no longer the case.

Additional design constraints that enforce each successive inboard bar to be at least 5% stiffer than it's outboard neighbor help the test engineer tune this FEM for what would be considered a more reasonable stiffness profile by engineering standards.

This design constraint is of the form: $\frac{K_{inboard}}{K_{outboard}} \geq 1.05$ for each bar pair from root to tip.

The plots show we only need to define these design constraints for I1, and J.

Example ID

MCDSG004

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
MCDSG004.dat	Input data	Provided: Contains all the data needed except for one stiffness profile constraint definition.

`MCDSG004_dsg.dat`	`Input data`	Generated by Design Studio. This file contains all the data created along with the data in `MCDSG004.dat`
`MCDSG004_dsg00.pch`	`Punch Output data`	Generated using Genesis within Design Studio. This file contains the mode shapes requested as analysis outputs
`MCDSG004_ref.dat`	`Input data`	Provided: Reference file with all the data for the problem. Similar to `MCDSG004_dsg.dat` file

Defining the Stiffness Profile Design Constraints

Rather than take the time to enter all data here this tutorial will read in a file that contains all but one constraint for J at the tip of the airfoil. The user will complete DCONS, DRESP2, DEQATN, and DTABLE entries for PBAR 3070/3075 by following the steps below.

Start Design Studio

1. Start Design Studio
2. Import the `MCDSG004.dat` file

Create a Constant Parameter for the Initial Value of J for BAR 3075

3. Select the **Design** tab
4. From the category chooser, select **Design Variables**

   Scroll to the bottom of the list. The last entry should be C Constant=6.0 for P3075I1. We will add the initial value of J=8.95 for P3075J. Note: The J value for a given design cycle is this constant multiplied by its associated design variable.

5. Push the **New Design Variable** button from the Edit menu toolbar
6. Push the **Constant Value (DTABLE)** button
7. For **Label**, enter `P3075J`
8. For **Value**, enter `8.95`
9. Push the **Finish** button

Create a Synthetic Response that gives the Stiffness Ratio for P3070J/P3075J

10. Select the **Design** tab
11. From the category chooser, select **Synthetic Responses**

12. Make a copy of J_14 by selecting it and then from the Edit menu toolbar, select **Copy Synthetic Response** button, and then **Paste Synthetic Response** button
13. Select `Copy of J_14`
14. Push the **Modify Synthetic Response** button from the Edit Menu toolbar
15. Enter `J_15` for the **Name**
16. Push the **Next>** button

    The synthetic response equation on the bottom at the bottom of the viewport will stay the same as it gives the stiffness ratio of any two adjacent bars with the inboard K in the numerator and the outboard K in the denominator. For J_15 the inboard bar will be P3070J and the outboard bar will be P3075J

17. For argument DVI, push the **Modify** button
18. For Design Variable, select the design variable `P3070J I=1.0`
19. Push the **Next>** button
20. For argument DVO, push the **Modify** button
21. For Design Variable, select the design variable `P3075J I=1.0`
22. Push the **Next>** button
23. For argument CI, push the **Modify** button
24. For Design Variable, select the design variable `Constant P3070J=13.39`
25. Push the **Next>** button
26. For argument CO, push the **Modify** button
27. For Design Variable, select the design variable `Constant P3075J=8.95`
28. Push the **Next>** button
29. Push the **Finish** button

Create the Stiffness Ratio Design Constraint for P3070J/P3075J

30. From the category chooser, select **Constraints**
31. Push the **New Constraint** button from the Edit menu toolbar
32. Enter `J_15` for the **Name**
33. Push the **Synthetic Response** button
34. For **Lower Bound**, enter `1.05`
35. Push the **Next>** button
36. Select the Synthetic Response for `J_15`

37. Push the **Next>** button
38. Select the existing **FREQ ID=2** loadcase
39. Push the **Finish** button

Request for Element Sizing data (OPOST) File

40. From the main menu bar, select **Genesis**→ **Options**
41. From the Genesis Options menu bar, select **File Control**
42. Push the **Element Sizing File** checkbox and select **Create** option
43. Push the **Apply** button

Optimize the FEM Using Genesis

44. From the main menu bar, select **Genesis**→ **Optimize**

    The Genesis console window will appear. Two design history charts will display the objective and the maximum constraint violation versus design cycle. These charts will update as the Genesis run progresses.

45. Study the **Design History** charts; when done, push the **Close** button
46. Study the **Genesis Console Output** window

Import the Post-Processing Files

47. From the **Genesis Console Output** window, select the **Import Post...** button
48. Select the punch file: `MCDSG003_dsgOPOSTxx.pch`
49. Push the **Import** button
50. When done, push the **Close** button in the **Genesis Console Output** window

Post-Processing the Sizing Results

51. From the Color Mesh, select any of the properties of the PBAR for last design cycle
52. Push the **Options...** button
53. Select the **Hide Elements With No Value** checkbox
54. Push the **Close** button

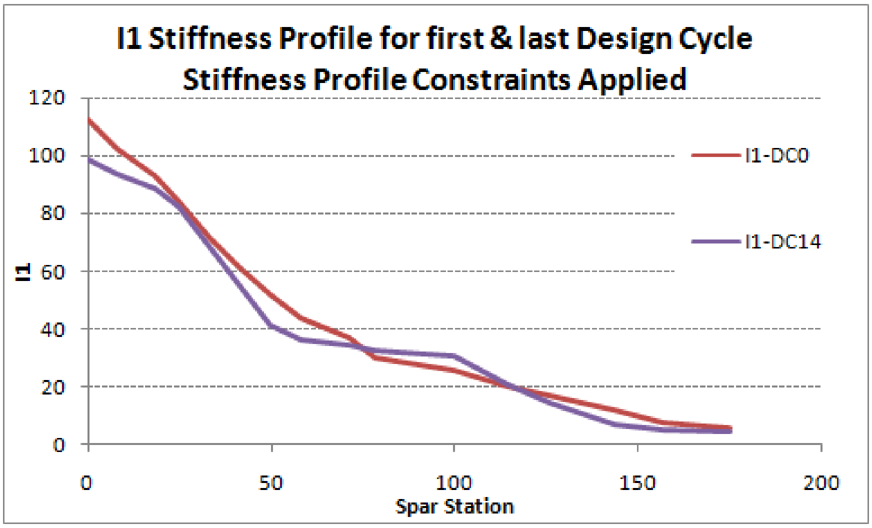
    Notice the color plot of the property value for each of the PBARs

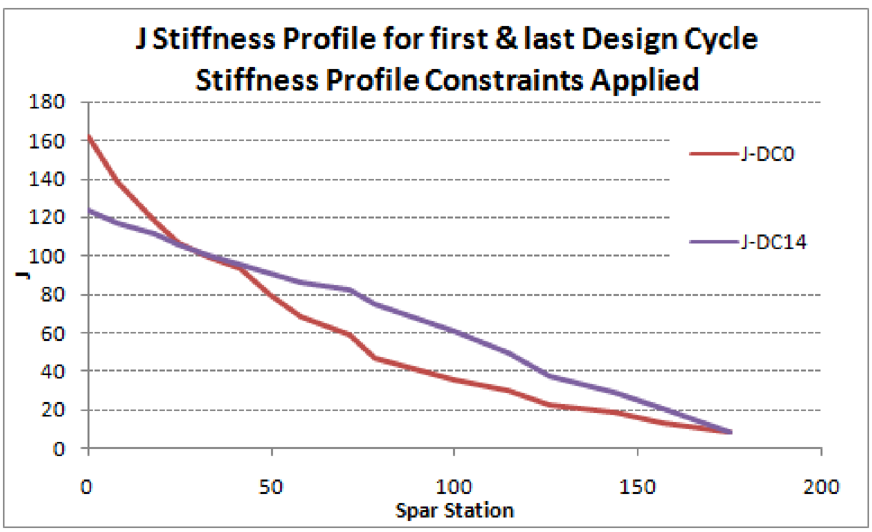
55. Study the results for last design cycle

    Note that the values of the stiffness for each of the three properties decrease, as specified by the constraints, while going from the root to the tip of the structure.

56. Push the **Up** button

The I1 & J PBAR values for the last design cycle (found in file `MCDSG004dsgUPDATExx` for the last design cycle) are plotted below:





Because of the stiffness profile design constraints there are no longer any troughs in the curves. This was exactly the intention for including these constraints, but one side effect is that the final frequencies don't match the target frequencies as well, but are still within an acceptable tolerance for test data correlation.

Mode Description	Analytical Mode Sequence	Target Frequency (Hz.)	1st Design Frequency (Hz.)	Last Design Frequency (Hz.)	Freq Constraints LB	Freq Constraints UB
1st Bending SYM	1	20.00	20.60	20.00	19.90	20.10
2nd Bending ASYM	2	27.06	27.54	27.05	26.96	27.16
3rd Bending SYM	3	49.13	49.84	49.08	49.03	49.23
In Plane Fore & Aft	4	55.09	55.90	55.02	54.99	55.19
1st Torsion ASYM	5	58.84	56.42	58.76	58.74	58.94
2nd Torsion SYM	6	87.64	78.35	87.48	87.54	87.74
4th bending ASYM	7	75.74	81.14	75.61	75.64	75.84

It may be possible to improve the frequency match by individually adjusting the constraint bounds and the weighting factors for the frequencies in the objective function. This is left as an exercise for the user.

CHAPTER 12

Super Element Examples

12.1 Truck Cabin - Full System Design

Introduction

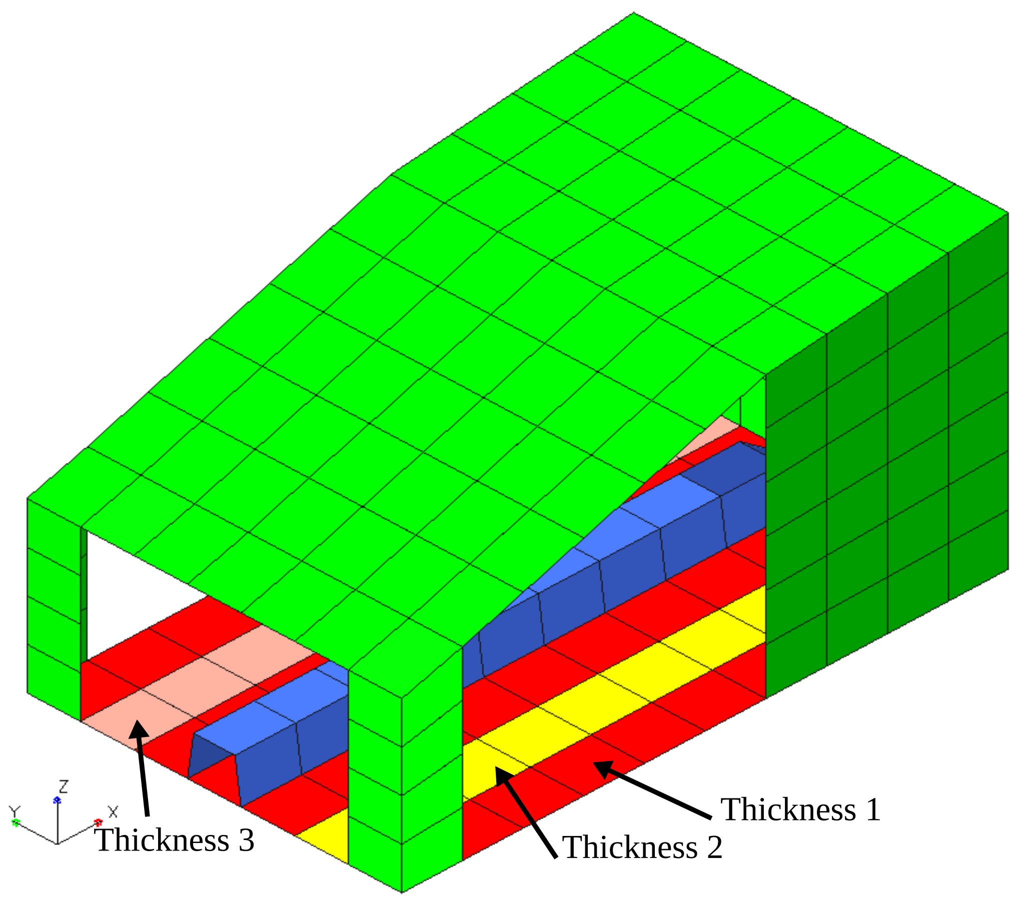
The purpose of this exercise is to setup an optimization problem for normal modes analysis for the entire Finite Element Model (FEM) without using superelements. The results for this full system design will serve as a baseline for subsequent exercises.

Genesis example ID:

D048

Problem Statement:

Maximize the frequency of the first elastic mode (mode 7) of the cabin subject to a mass constraint of 9kg for the bottom parts (MAT1=1). The design variables are the shell thicknesses of the bottom parts shown below.



Design Problem:

Maximize frequency of the first elastic mode.

1. One normal modes load case.
2. Three design variables (thickness of the plates).
3. One mass constraint.

Problem Description:

1. The design objective is to maximize the frequency of the first elastic mode which is mode number 7. The first six modes are rigid body modes.

2. The structure consists of 5 PSHELL groups of plate elements. The top of the cabin (PID=5) is not designed and remains unchanged. The Center tunnel of the floor (PID=7) is also not designed. The remaining 3 floor sections are sized with the following shell thickness design variables:

DVAR 1, PID 3 Initial Thickness=1.0, LB=0.5, UB=2.0

DVAR 2, PID 10 Initial Thickness=1.0, LB=0.5, UB=2.0

DVAR 3, PID 12 Initial Thickness=1.0, LB=0.5, UB=2.0

3. The mass of the entire floor (PID's 3, 7, 10, and 12) is constrained to 9kg. MID=1 is the unique material ID for the floor assembly and is exploited to easily apply the mass constraint on the floor assembly.

Example ID

`SEDSG001`

Files Used in This problem

A list of the key files provided and created is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SEDSG001.dat`	`Input data`	Provided: Contains the finite element mesh of the structure
`SEDSG001_ref.dat`	`Input data`	Provided: Reference input file ready to be optimized
`SEDSG001_dsg.dat`	`Input data`	Created: File written by Design Studio to run genesis. It contains all the data created in this example along with data in `SEDSG001.dat`
`SEDSG001_dsg.out`	`Output file`	Created: file listing output of Genesis Optimization

Start Design Studio

1. Start Design Studio

2. Import the Genesis data file: `SEDSG001.dat`

Modify the Existing Normal Modes Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select `Loadcase 1`
5. Push the **Modify Loadcase** button from the Edit Menu toolbar
6. Enter `Free_Normal_Modes` in the **Name** field
7. Push **Next>**

   Verify that there are no SPC or MPC boundary conditions for this loadcase.
8. Push **Next>**

   Verify that Method 1 is chosen for the Eigenvalue Method.
9. Push **Next>**
10. Push the **Finish** button

Define the Design Objective

11. From the **Design** category chooser, select **Objectives**
12. Push the **New Objective** button from the Edit Menu toolbar
13. Enter `Freq` in the **Name** field
14. Select the **Frequency Mode Number** radio button as Response Type and type in `7` for the mode number
15. Select the **Max** radio button as Objective Definition Switch
16. Push **Next>**
17. Select the `Free_Normal_Modes` loadcase
18. Push the **Finish** button

Define the Constraint

19. From the **Design** category chooser, select **Constraints**
20. Push the **New Constraint** button from the Edit Menu toolbar
21. Enter `Mass` in the **Name** field
22. Select the **Mass** radio button as the Response Type
23. From the **Mass** pull down menu, select **Selected Material**

24. Enter `0.009` as **Upper Bound**
25. Push **Next>**
26. From the **Material** pull down menu, select `MAT1 1`
27. Push the **Finish** button

Create an Independent Design Variable for the thickness

28. From the **Design** category chooser, select **Design Variables**
29. Push the **New Design Variable** button from the Edit Menu toolbar
30. Enter `THCK1` in the **Name** field
31. Select the **Independent Design Variable** radio button
32. Push **Next>**
33. Enter `1.0` as **Initial Value**, `0.5` as **Lower Bound**, and `2.0` as **Upper Bound**
34. Push the **Finish** button

    Notice that there is an asterisk in front of the independent design variable, it indicates that this design variable is not being used. The "**I**" indicates this is an **I**ndependent design variable.

35. Select the design variable `THCK1` and from the Edit menu toolbar, select the **Copy Design Variable** button
36. From the Edit menu toolbar, select the **Paste Design Variable** button twice to create 2 copies of the design variable
37. Select the first `Copy of THCK1`, and push the **Modify Design Variable** button from the Edit Menu toolbar
38. Enter `THCK2` in the **Name** field
39. Push the **Finish** button
40. Select the second `Copy of THCK1`, and push the **Modify Design Variable** button from the Edit Menu toolbar
41. Enter `THCK3` in the **Name** field
42. Push the **Finish** button

Define the Sizing Data

43. From the **Design** category chooser, select **Sizing**
44. Select PSHELL 3 group
45. Push the **Modify Sizing Design** button from the Edit Menu toolbar

46. From the **Thickness** category chooser, select `1 THCK1`
47. Push the **Finish** button
48. Select PSHELL 10 group
49. Push the **Modify Sizing Design** button from the Edit Menu toolbar
50. From the **Thickness** category chooser, select `2 THCK2`
51. Push the **Finish** button
52. Select PSHELL 12 group
53. Push the **Modify Sizing Design** button from the Edit Menu toolbar
54. From the **Thickness** category chooser, select `3 THCK3`
55. Push the **Finish** button

Set the Maximum Number of Design Cycles to 15

56. From the **Genesis** menu, select **Options...**
57. Select the **Design Control** tab
58. Check the **Maximum Design Cycles** check box, and enter `15`
59. Push the **Apply** button

Save the Design Studio file

60. From the main menu bar, select **File** → **Save As...**
61. Enter `SEDSG001` as the Filename and push **Save (**as a Design Studio File)

Optimize the structure using Genesis

62. From the main menu bar, select **Genesis** →**Optimize**
63. Study the **Genesis Console Output**; notice the design objective has increased from 6.5061 Hz. to 9.2857 Hz.
64. Push the **Close** button

Study the Output file

65. Start any text editor
66. In the text editor load the Genesis data file: `SEDSG001_dsg.out`

67. Navigate to the bottom of the file and study the three design variable histories

    For the final design cycle:
    THCK1=1.7139mm, THCK2=1.1930mm and THCK3=1.1930mm

68. Close the file and exit the editor

Quit Design Studio

69. From the main menu bar, select **File** →**Quit**

70. Push the **Don’t Save** button

12.2 Truck Cabin - Superelement Simulation using Component Mode Synthesis

Introduction

The purpose of this exercise is to show how the top of the truck cabin, which is not being designed, can be reduced to a superelement using Design Studio.

Genesis example ID:

D049

Problem Statement:

Six Craig-Bampton modes will be calculated relative to an A-set interface of the truck cabin. The C-B modes along with statically condensed Constraint modes (i.e. Guyan reduction) will be assembled into Matrices [KAA] and [MAA] for the top of the truck cabin. This superelement data will be used in subsequent exercises to optimize the truck in a substructured optimization set up.

Data File:

This problem starts with the normal modes data for just the top of the truck cabin in file `SEDSG002.dat`. Six S-points have been defined in the bulk data that will be used as placeholders for the six C-B modes.

Example ID

`SEDSG002`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SEDSG002.dat	Input data	Provided: Contains the finite element mesh of the structure
SEDSG002_r.dat	Input data	Created: Contains the finite element mesh of the structure and superelement set up data
SEDSG002_ref.dat	Input data	Provided: Reference input file ready to be analyzed
SEDSG002_r_dsg0001.MAA	Output data	Created: Superelement mass data

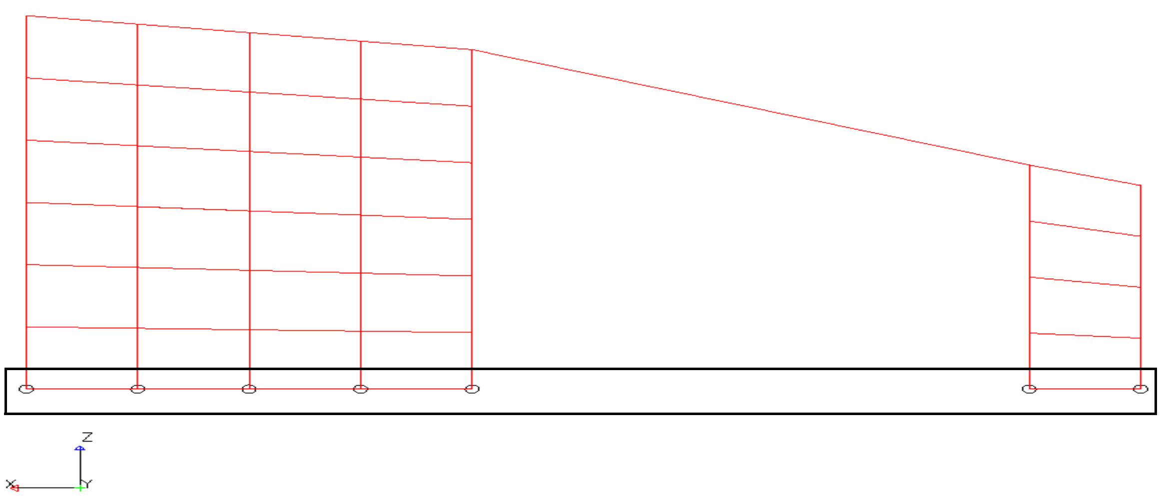
`SEDSG002_r_dsg0001.KAA`	`Output data`	Created: Superelement stiffness data
`SEDSG002_r_dsg.out`	`Output file`	Created: file listing output of Genesis analysis

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SEDSG002.dat`

Create the A-set at the Truck Top/Floor Interface

3. From the **Analysis** category chooser, select **Grid-Component Sets**
4. Push the **New Grid-Component Set** button from the Edit Menu toolbar
5. Enter `ASET` in the **Name** field
6. Check the **ASET** radio button
7. Push **Next>**
8. View the model in the X-Z plane and select the grids at the bottom by click-and-hold the left mouse button and dragging a selection box as shown below:



This operation will display “22 grids selected” below the Grid Selection Options box in the right viewport.

9. Enter `123456` for Components, and push the **Set Components** button.

This will display “132 total dof on 22 grids” at the bottom of the window.

10. Push the **Finish** button

Assign the S-Points to the Q-set for the Craig-Bampton Modes

11. From the **Analysis** category chooser, select **Grid-Component Sets**
12. Push the **New Grid-Component Set** button from the Edit Menu toolbar
13. Enter `QSET` in the **Name** field
14. Check the **QSET** radio button
15. Push **Next>**
16. Push the **Select None** button
17. Enter `501-506` in the Select by Grid ID: field an push the **Add** button

    501-506 are the S-Points defined in the bulk data file.
18. Enter `0` in the Components: field and push the **Set Components** button

    This will display “6 total dof on 6 grids” at the bottom of the window.
    Here 0 corresponds to a S-Points.
19. Push the **Finish** button

Create an Eigenvalue Method for the Craig-Bampton Modes

20. From the **Analysis** category chooser, select **Eigenvalue Methods**
21. Push the **New Eigenvalue Method** button from the Edit Menu toolbar
22. Enter `CB_Method` in the **Name** field
23. Check the **Lanczos** radio button
24. Leave the **V1** field blank, and insert a blank for the **V2** field.
25. Enter `30` for the **Number of Modes** field

    Even though Genesis will calculate 30 C-B modes for the roof we have only set aside space for the first 6 component modes in the q-set. The remaining modes (7-30) will not be part of the superelement.
26. Push the **Finish** button

Modify the Frequency Loadcase for CMS

27. From the **Analysis** category chooser, select **Loadcases**
28. Select `Normal Modes`, and push the **Modify Loadcase** button from the Edit Menu toolbar
29. Enter `Superelement_Assembly` in the **Name** field
30. Push **Next>**

31. Push **Next>**
32. From the **ASET** category chooser, select `1 ASET`
33. From the **QSET** category chooser, select `2 QSET`
34. From the **Craig-Bampton Method** category chooser, select `2 CB_Method`
35. Push the **Finish** button

Export the Bulk Data File

36. From the main menu bar, select **File** →**Export** → **Input Data**
37. Enter `SEDSG002_r.dat` in the File Name field
38. Push the **Save** button

Quit Design Studio

39. From the main menu bar, select **File** →**Quit**

Edit the Genesis Input File just Created

40. Start any text editor
41. In the text editor load the Genesis data file: `SEDSG002_r.dat`
42. Modify the subcase so that it includes the MAA and KAA commands as shown below:

```
SUBCASE 1
   LABEL = Superelement_Assembly
   METHOD = 1
   ASET = 1
   CBMETHOD = 2
   QSET = 2
   MODTRK=NO
   MAA = POST
   KAA = POST
BEGIN BULK
```

43. Save and close the file.
44. Exit the editor

Start Design Studio and Run Genesis Analysis

45. Start Design Studio
46. Import the Genesis data file: `SEDSG002_r.dat`
47. From the main menu bar, select **Genesis** →**Single Analysis**
48. Push the **Close** button to close the **Genesis Console Output** window
49. Make sure two files have been created in the work directory:

```
SEDSG002_r_dsg0001.KAA
SEDSG002_r_dsg0001.MAA
```

Quit Design Studio

50. From the main menu bar, select **File** →**Quit**
51. Push the **Don’t Save** button

Study the Output file

52. Start any text editor
53. In the text editor load the Genesis data file: `SEDSG002_r_dsg.out`
54. Study the file. In particular compare the eigenvalue tables for the unrestrained system modes and the fixed Craig-Bampton modes.
55. Close the file and exit the editor

12.3 Truck Cabin - Design Using Imported External Superelement

Introduction

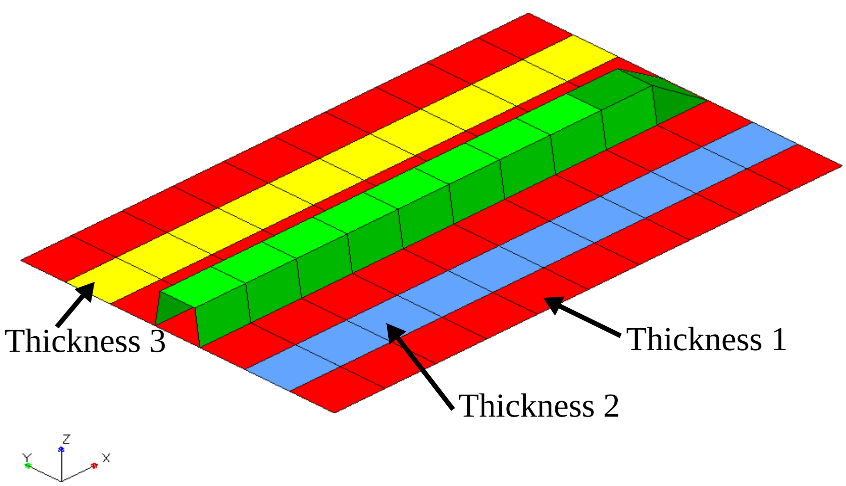
The purpose of this exercise is to setup an optimization problem for normal modes analysis of the substructured truck cabin. The FEM bulk data consists of the floor, which is designed during optimization, and the center tunnel. The top section which isn't designed will be brought into the analysis as a superelement. The results for this substructured design optimization analysis should match those of the full system design.

Genesis example ID:

D050

Problem Statement:

Maximize the frequency of the first elastic mode (mode 7) of the cabin subject to a mass constraint of 9kg for the bottom parts (MAT1=1). The design variables are the shell thicknesses of the bottom parts shown below. The top of the cabin that was reduced to a superelement in the previous exercise will be represented in this Genesis problem as external boundary matrices.



Design Problem:

Maximize frequency of the first elastic mode.

1. One normal modes load case.
2. Three design variables (thickness of the plates).
3. One mass constraints.

Problem Description:

1. The design objective is to maximize the frequency of the first elastic mode which is mode number 7. The first six modes are rigid body modes.

2. The structure consists of 4 PSHELL groups of plate elements. The top of the cabin is not designed and is brought into the analysis as a superelement. The Center tunnel of the floor (PID=7) is also not designed. The remaining 3 floor sections are sized with the following shell thickness design variables:

DVAR 1, PID 3 Initial Thickness=1.0, LB=0.5, UB=2.0

DVAR 2, PID 10 Initial Thickness=1.0, LB=0.5, UB=2.0

DVAR 3, PID 12 Initial Thickness=1.0, LB=0.5, UB=2.0

3. The mass of the entire floor (PID's 3, 7, 10, and 12) is constrained to 9kg. MID=1 is the unique material ID for the floor assembly and is exploited to easily apply the mass constraint on the floor assembly.

Example ID

SEDSG003

Files Used in This problem

A list of the key files provided and created is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SEDSG003.dat	Input data	Provided: Contains the finite element mesh of the structure. The executive control section already contains M2UU, and K2UU entries for the cabin top superelement.
SEDSG003_ref.dat	Input data	Provided: Reference input file ready to be optimized
SEDSG003_dsg0001.MAA	Output data	Provided: Superelement mass data for truck top. Same as the file SEDSG002_r_dsg0001.MAA created in the previous example
SEDSG003_dsg0001.KAA	Output data	Provided: Superelement stiffness data for truck top. Same as the file SEDSG002_r_dsg0001.KAA created in the previous example
SEDSG003_dsg.out	Output file	Created: file listing output of Genesis Optimization

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SEDSG003.dat`

Check the Loadcases

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the loadcase and push the **Modify Loadcase** button from the Edit Menu toolbar
5. Push **Next>**
6. Push **Next>**
7. Make sure the Eigenvalue method is correct.
8. Push the **Cancel** button

Define the Design Objective

9. From the **Design** category chooser, select **Objectives**
10. Push the **New Objective** button from the Edit Menu toolbar
11. Enter `Freq` in the **Name** field
12. Select the **Frequency Mode Number** radio button as Response Type and type in `7` for the mode number
13. Select the **Max** radio button as Objective Definition Switch
14. Push **Next>**
15. Select the **Free Normal Modes** loadcase
16. Push the **Finish** button

Define the Constraint

17. From the **Design** category chooser, select **Constraints**
18. Push the **New Constraint** button from the Edit Menu toolbar
19. Enter `Mass` in the **Name** field
20. Select the **Mass** radio button as the Response Type
21. From the **Mass** pull down menu, select **Selected Material**
22. Enter `0.009` as **Upper Bound**

23. Push **Next>**
24. From the **Material** pull down menu, select `MAT1 1`
25. Push the **Finish** button

Create an Independent Design Variable for the thickness

26. From the **Design** category chooser, select **Design Variables**
27. Push the **New Design Variable** button from the Edit Menu toolbar
28. Enter `THCK1` in the **Name** field
29. Select the **Independent Design Variable** radio button
30. Push **Next>**
31. Enter `1.0` as **Initial Value**, `0.5` as **Lower Bound**, and `2.0` as **Upper Bound**
32. Push the **Finish** button

    Notice that there is an asterisk in front of the independent design variable, it indicates that this design variable is not being used. The "**I**" indicates this is an **I**ndependent design variable.

33. Select the design variable `THCK1` and from the Edit menu toolbar, select the **Copy Design Variable** button
34. From the Edit menu toolbar, select the **Paste Design Variable** button twice to create 2 copies of the design variable
35. Select the first `Copy of THCK1`, and push the **Modify Design Variable** button from the Edit Menu toolbar
36. Enter `THCK2` in the **Name** field
37. Push the **Finish** button
38. Select the second `Copy of THCK1`, and push the **Modify Design Variable** button from the Edit Menu toolbar
39. Enter `THCK3` in the **Name** field
40. Push the **Finish** button

Define the Sizing Data

41. From the **Design** category chooser, select **Sizing**
42. Select PSHELL 3 group
43. Push the **Modify Sizing Design** button from the Edit Menu toolbar
44. From the **Thickness** category chooser, select `1 THCK1`

45. Push the **Finish** button
46. Select PSHELL 10 group
47. Push the **Modify Sizing Design** button from the Edit Menu toolbar
48. From the **Thickness** category chooser, select `2 THCK2`
49. Push the **Finish** button
50. Select PSHELL 12 group
51. Push the **Modify Sizing Design** button from the Edit Menu toolbar
52. From the **Thickness** category chooser, select `3 THCK3`
53. Push the **Finish** button

Set the Maximum Number of Design Cycles to 15

54. From the **Genesis** menu, select **Options...**
55. Select the **Design Control** tab
56. Check the **Maximum Design Cycles** check box, and enter `15`
57. Push the **Apply** button

Optimize the structure using Genesis

Before launching the optimization, make sure that the files `SEDSG003_dsg0001.MAA` and `SEDSG003_dsg0001.KAA` are present in the current working directory (provided for this exercise).
If you do not have these files, go through the previous example to create these files and make sure to rename the files:
`SEDSG003_dsg0001.MAA` as `SEDSG002_r_dsg0001.MAA`
`SEDSG003_dsg0001.KAA` as `SEDSG002_r_dsg0001.KAA`

58. From the main menu bar, select **Genesis** →**Optimize**
59. Study the **Genesis Console Output**; notice the design objective has increased from 6.5174 Hz. to 9.3141 Hz.
60. Push the **Close** button

Quit Design Studio

61. From the main menu bar, select **File** →**Quit**

Study the Output file

62. Start any text editor

63. In the text editor load the Genesis data file: `SEDSG003_dsg.out`

64. Navigate to the bottom of the file and study the three design variable histories

    For the final design cycle:
    THCK1=1.7134mm
    THCK2=1.1942mm
    THCK3=1.1943mm

    Note: These values for frequency and final thicknesses compare favorably with the results of the full system model optimization.

65. Close the file and exit the editor

Compare The Results for Both Models

The table below shows the results for both the full system model, and the substructured model. The final frequency and the floor thickness values compare favorably for both optimizations.

	Final Design Cycle	
	Full System Model	Sub-Structured Model
Mode 7 Freq (Hz.)	9.2857	9.3141
THCK1 (mm)	1.7139	1.7134
THCK2 (mm)	1.1930	1.1942
THCK3 (mm)	1.1930	1.1943

Notice that the substructured model with the superelement Craig-Bampton (C-B) modes calculated a slightly higher frequency than the full system model. This is normal when a truncated set of C-B modes is used for the superelement and is evident for all design cycles in the output file. If a complete set of component modes is used in the superelement reduction then these results would be the same since no information is discarded and the two simulations are mathematically identical.

Since the frequency difference of mode 7 for the two simulations is only 0.3% these results show the substructured model has been reduced accurately. For large FEM's that run many design cycles CMS can be used to represent portions of the model that are not designed and is a good way to reduce run times without sacrificing accuracy.

12.4 Truck Cabin - Superelement REDUCE Mode

Introduction

The purpose of this exercise is to demonstrate how to statically reduce the mass, stiffness, and loading of the truck roof to its boundary degrees of freedom for later use as a superelement in static analysis problem.

Problem Statement:

Consider the Case Control section of provided file.

```
SOL COMPAT0
CEND
TITLE=DEMONSTRATION SUPERELEMENT REDUCTION PROBLEM
SUBTITLE=PART 1 - REDUCTION OF SUPERELEMENT TO BOUNDARY
BOUNDARY = 1
ALOAD=DMIG
$
LOADCASE        1
   LABEL=Z-DIRECTION ACCELERATION
    KAA = DMIG
    MAA = DMIG
LOADCASE        2
   LABEL=X-DIRECTION ACCELERATION
LOADCASE        3
   LABEL=FRONT LOAD
    LOAD   =       3
LOADCASE        4
   LABEL=TOP LOAD
    LOAD   =       4
LOADCASE        5
   LABEL=GRAVITY LOAD
    LOAD   =      50
BEGIN BULK
```

The BOUNDARY command points to the ASET2/3 entries that contain Grid/Component data for all superelement degrees of freedom.

The ALOAD=DMIG command is inserted above the first loadcase to output the reduced loads for all loadcases in DMIG format.

Because all loadcases use the same boundary conditions (in this case, no SPC/MPC entries apart from the boundary), the reduced mass & stiffness matrices will be the same for all loadcases. The KAA/MAA commands are placed inside the first loadcase instead of above it so that only one copy of the matrices will be generated.

When this superelement model is used in the coupled system model, loadcases 1 & 2 will use inertia relief. During superelement reduction do NOT use a SUPORT entry. Instead, the reduced mass of the roof must be coupled to the residual structure (i.e. the floor+tunnel) to account for the inertia relief of the superelement. With ordinary static/thermal/gravity loads the reduced mass is not needed because the reduced loads will contain all load information for the superelement.

Notice that loadcases 1 & 2 do not have any load activator commands, because all of the direct loading is applied to the residual structure. Here, loadcases 1 & 2 are simply placeholders, so that the reduced load vectors for the remaining loadcases will be in their proper columns in the DMIG file.

Example ID

SEDSG004

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SEDSG004.dat	Input data	Provided: Contains the finite element mesh of the structure, and the case control set up for the REDUCE exercise
SEDSG004_ref.dat	Input data	Provided: Reference input file ready to be analyzed
SEDSG004_dsg00.DMIG	DMIG data	Created: File that contains the reduced mass, stiffness, and loads

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SEDSG004.dat`

Check the Grid-Component Sets

3. From the **Analysis** category chooser, select **Grid-Component Sets**
4. Select the `ASET 1` and study it in the Viewport
5. Select the `SPC Set 1` and study it in the Viewport

Check the Loadcases

6. From the **Analysis** category chooser, select **Loadcases**
7. Select each loadcase and study them in the Viewport

Set the REDUCE Executive Control Command

8. From the **Genesis** menu, select **Options...**
9. Select the **Analysis Control** tab
10. Push the **Advanced...** button
11. Check the **REDUCE Mode** check box
12. Push the **Close** button
13. Push the **Apply** button

Reduce the Substructure using Genesis

14. From the main menu bar, select **Genesis** →**Single Analysis**
15. Study the **Genesis Console Output**
16. Push the **Close** button

Quit Design Studio

17. From the main menu bar, select **File** →**Quit**

    The working directory for this problem should now show that file `SEDSG004_dsg00.DMIG` has been created. This file contains the reduced mass, stiffness, and load matrices for the superelement and will be used in the design for the next problem.

12.5 Truck Cabin - Design Using Statically Reduced Superelement

Introduction

The purpose of this exercise is to show how to use the statically reduced superelement processed in the previous example to design the truck cabin. The truck top superelement for this example contains reduced mass, stiffness, and loading data.

Problem Statement:

Minimize the mass of the truck floor and center tunnel subject to stress constraints for 5 loadcases. The design variables are the thicknesses of the floor parts, and the shape of the center tunnel. Consider the Solution Control section of provided file.

```
TITLE=DEMONSTRATION SUPERELEMENT REDUCTION PROBLEM
SUBTITLE=PART 2 - RESIDUAL MODEL
K2GG = K0000001
M2GG = M0000001
P2G = PASET
$
LOADCASE        1
   LABEL=Z-DIRECTION ACCELERATION
    SUPORT =    1
    LOAD =      1
LOADCASE        2
   LABEL=X-DIRECTION ACCELERATION
    SUPORT =    1
    LOAD =      2
LOADCASE        3
   LABEL=FRONT LOAD
    SPC    =      1
LOADCASE        4
   LABEL=TOP LOAD
    SPC    =      1
LOADCASE        5
   LABEL=GRAVITY LOAD
    SPC    =      1
    LOAD   =     50
BEGIN BULK
```

The K2GG/M2GG/P2G commands activate the superelement reduction matrices in the DMIG file `SEDSG005_dsg00.DMIG`.

Loadcases 3, & 4 do not contain any load activator commands because all of the loading for these cases is supplied by the superelement.

WARNING
No data recovery is performed for the superelement. That is to say, no stresses or displacements are calculated for the portion of the model eliminated. If the superelement reduction technique is used in an optimization problem (like this example), only FE responses (e.g. stresses, displacements, etc.) for elements and grids defined in the residual structure (i.e. defined in this file) can be used. Satisfied stress constraints for the residual does NOT necessarily guarantee that the reduced portion will have acceptable stresses. A final analysis of the full model should be used to insure all stresses/displacements are acceptable.

Example ID

`SEDSG005`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this exercise, is presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`SEDSG005.dat`	`Input data`	Provided: Contains the finite element mesh of the structure, and the case control set up for reading DMIG data
`SEDSG005_ref.dat`	`Input data`	Provided: Reference input file ready to be analyzed
`SEDSG005_dsg00.DMIG`	`DMIG data`	Provided: File that contains the reduced mass, stiffness, and loads. Same as the file `SEDSG004_dsg00.DMIG` created in the previous example

Start Design Studio

Before importing the start file, make sure that the file `SEDSG005_dsg00.DMIG` is present in the current working directory (provided with this example).
If you do not have this file, go through the previous example and make sure to rename the files:
`SEDSG004_dsg00.DMIG` as `SEDSG005_dsg00.DMIG`

1. Start Design Studio
2. Import the Genesis data file: `SEDSG005.dat`

Check the Loadcases

3. From the **Analysis** category chooser, select **Loadcases**
4. Study the loadcases in the Viewport

Define the Design Objective

5. From the **Design** category chooser, select **Objectives**
6. Push the **New Objective** button from the Edit Menu toolbar
7. Enter `Mass` in the **Name** field
8. Select **Mass** as the Response Type and keep **Entire Model** in the pull down menu
9. Select the **Min** radio button as Objective Definition Switch
10. Push the **Finish** button

Define the Constraint for Groups 3, 10, & 12

When specifying constraints in Genesis those elements whose properties are designed with DVPROP3 (as PID’s 3, 10, & 12 are) must be kept separate from the elements that do not use DVPROP3. If the constraint is defined and these elements are not separated on the DRESP1 entry then the following error message will occur:

```
ERROR MESSAGE FROM SUBROUTINE "GN16CT".
ERROR CODE = 160032
DVPROP3 ELEMENTS ARE MIXED WITH NON-DVPROP3 ELEMENTS IN DRESP1
ENTRY        2.
MOVE DVPROP3 ELEMENTS TO A SEPARATE DRESP1 ENTRY.
```

This is why group 7 is separated from groups 3, 10, & 12 even though their stress constraint definition is the same.

11. From the **Design** category chooser, select **Constraints**
12. Push the **New Constraint** button from the Edit Menu toolbar
13. Enter `VonMises-3,10,12` in the **Name** field
14. Select the **Stress** radio button as the Response Type
15. From the **Stress** pull down menu, select **Selected Groups**
16. Enter `30.0` as **Upper Bound**
17. Push **Next>**
18. Select the 3 groups that are thickness designed: PSHELL 3, 10, & 12
19. Push **Next>**
20. From the **PSHELL Stress** pull down menu, select **von Mises Top & Bottom**
21. Push **Next>**
22. Select all the loadcases

23. Push the **Finish** button

Define the Constraint for Group 7

24. From the **Design** category chooser, select **Constraints**
25. Push the **New Constraint** button from the Edit Menu toolbar
26. Enter `VonMises-7` in the **Name** field
27. Select the **Stress** radio button as the Response Type
28. From the **Stress** pull down menu, select **Selected Groups**
29. Enter `30.0` as **Upper Bound**
30. Push **Next>**
31. Select the PSHELL 7 group that is shape designed
32. Push **Next>**
33. From the PSHELL Stress pull down menu, select **von Mises Top & Bottom**
34. Push **Next>**
35. Select all the loadcases
36. Push the **Finish** button

Create Independent Design Variables for the Thickness

37. From the **Design** category chooser, select **Design Variables**
38. Push the **New Design Variable** button from the Edit Menu toolbar
39. Enter `THCK1` in the **Name** field
40. Select the **Independent Design Variable** radio button
41. Push **Next>**
42. Enter `1.0` as **Initial Value**, `0.5` as **Lower Bound**, and `2.0` as **Upper Bound**
43. Push the **Finish** button

    Notice that there is an asterisk in front of the independent design variable, it indicates that this design variable is not being used. The "**I**" indicates this is an **I**ndependent design variable.

44. Select the design variable `THCK1` and from the Edit menu toolbar, select the **Copy Design Variable** button
45. From the Edit menu toolbar, select the **Paste Design Variable** button twice to create 2 copies of the design variable

46. Select the first `Copy of THCK1`, and push the **Modify Design Variable** button from the Edit Menu toolbar
47. Enter `THCK2` in the **Name** field
48. Push the **Finish** button
49. Select the second `Copy of THCK1`, and push the **Modify Design Variable** button from the Edit Menu toolbar
50. Enter `THCK3` in the **Name** field
51. Push the **Finish** button

Define the Sizing Data

52. From the **Design** category chooser, select **Sizing**
53. Select PSHELL 3 group
54. Push the **Modify Sizing Design** button from the Edit Menu toolbar
55. From the **Thickness** category chooser, select `1 THCK1`
56. Push the **Finish** button
57. Select PSHELL 10 group
58. Push the **Modify Sizing Design** button from the Edit Menu toolbar
59. From the **Thickness** category chooser, select `2 THCK2`
60. Push the **Finish** button
61. Select PSHELL 12 group
62. Push the **Modify Sizing Design** button from the Edit Menu toolbar
63. From the **Thickness** category chooser, select `3 THCK3`
64. Push the **Finish** button

Create an Independent Design Variable for the Shape Perturbation

65. From the **Design** category chooser, select **Design Variables**
66. Push the **New Design Variable** button from the Edit Menu toolbar
67. Enter `SHAP1` in the **Name** field
68. Select the **Independent Design Variable** radio button
69. Push **Next>**

70. Enter `0.0` as **Initial Value**, `-1.0` as **Lower Bound**, and `1.0` as **Upper Bound**
71. Push the **Finish** button

    Notice that there is an asterisk in front of the independent design variable, it indicates that this design variable is not being used. The "**I**" indicates this is an **I**ndependent design variable.

Create a Quad Shape Domain

72. From the **Design** category chooser, select **Shape Domains**
73. Push the **New Domain** button from the Edit Menu toolbar
74. Check the **Define A New Original Domain** radio button
75. Push **Next**>
76. Select `Domain 13` as the **Domain Group**
77. Push **Next>**
78. Select the **Quad** radio button
79. Push **Next>**
80. Enter `34,45,376,377` in the **Select by Grid ID** field
81. Push the **Add** button
82. Push **Next>**
83. Push the **Select Interior Grids** button

    "42 grids selected" should be displayed at the bottom of the window
84. Push the **Finish** button

Create a Perturbation Associated to the Domain

85. From the **Design** category chooser, select **Shape Morphing Sets**
86. Push the **New Shape Set** button from the Edit Menu toolbar
87. Enter `TUNNEL-Z` in the **Name** field
88. Select the **Domain Morphing Set** radio button
89. Select `SHAP1` in the Design Variable category chooser
90. Push **Next>**
91. Select `Domain 1` from the **Select Act Upon** list.
92. Push **Next>**
93. Push the **Select None** button

94. Enter `377` in the **Select by Grid ID** field

    This grid could also be picked by the mouse in the main viewport. Verify that there is 1 grid selected to apply the perturbation to.

95. Push the **Add** button
96. Push the **By 2 Grids...** button
97. Enter `377,34` in the **Select by Grid ID** field, and hit enter

    Verify that there are 2 grids chosen. These grids define the direction of the perturbation from grid 377 to 34, and could also have been picked by the mouse in the main viewport.

98. Push **Next>**
99. Enter `15.0` for the **Magnitude**
100. Push the **Add Perturbation** button

    Verify that there is “**1 perturbations on 1 grids”** at the bottom of the window

101. Push the **Select None** button
102. Enter `376` in the **Select by Grid ID** field
103. Push the **Add** button

    Verify that there is 1 grids selected.

104. Push the **By 2 Grids...** button
105. Enter `376,45` in the **Select by Grid ID** field, and hit enter

    This defines the direction of the perturbation from grid 376 to grid 45

106. Push **Next>**
107. Enter `15.0` for the **Magnitude**
108. Push the **Add Perturbation** button

    Verify that there is “**2 perturbations on 2 grids”** at the bottom of the window

109. Push the **Finish** button
110. Right click on the Viewport window and select **Clear** → **All**

Preview the Shape Morphing Set

111. Select the **Post** tab
112. Push the **Deform Mesh/Color Mesh** button
113. Select **DVAR 4 Shape Morphing Set Preview**
114. Select the **Oscillate** radio button

    This animates the shape morphing set so that the user can verify the shape is changing as intended, and is not radically deforming the mesh.

Set the Analysis Print (APRINT) to Output only the First and Last Design Cycle Responses

115. From the **Genesis** menu, select **Options...**
116. Select the **Output Control** tab
117. Check the **Analysis Output** check box, and select First & Last from the pull down menu
118. Push the **Apply** button

Optimize the structure using Genesis and View the Results

119. From the main menu bar, select **Genesis** →**Optimize**
120. Study the **Genesis Console Output**
121. Push the **Import Post..** button
122. Select `SEDSG005_dsg.SHP` from the file browser
123. Select all `SEDSG005_dsgxx.pch` punch result files where `xx` corresponds to the design cycle number
124. Push the **Import** button
125. Push **Deform Mesh/Color Mesh** in the **Post** tab
126. Study the results

Quit Design Studio

127. From the main menu bar, select **File** →**Quit**

Study the Output file

128. Start any text editor
129. In the text editor load the Genesis data file: `SEDSG005_dsg.out`
130. Navigate to the bottom of the file and study the three design variable histories

    For the final design cycle:
    THCK1=0.8701mm
    THCK2=2.0000mm
    THCK3=0.5000mm
    SHAP1=1.0 => 15mm shorter in height for the tunnel
    The final mass is 6.0298kg

131. Close the file and exit the editor

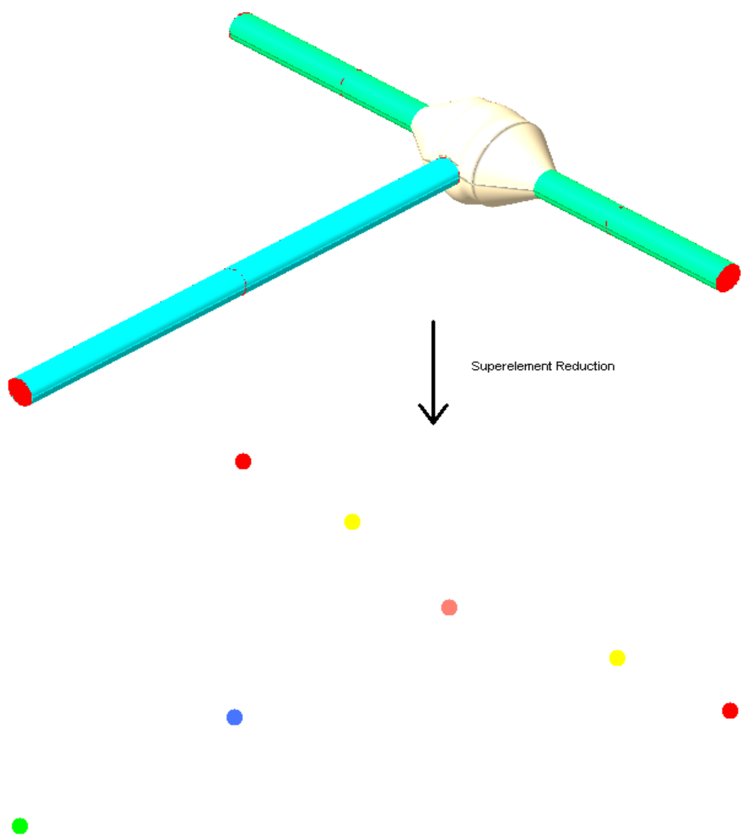
12.6 Driveline - Superelement reduction using the Craig-Bampton modes

Introduction

The purpose of this example is to learn how to perform a superelement reduction using the Craig-Bampton modes. The example is divided in two parts:

- Set up the superelement reduction

- Use the superelement reduction files provided from the previous step



In the part 1, the tube and the box will be reduced in order to keep only the CBUSH and the CELAS elements.

In the part 2, you will learn how to use the product of the reduction in order to perform a frequency response analysis.

Analysis Problem:

SMS method

Example ID

SEDSG006

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
SEDSG006_1.dat	Input data	Provided: Contains the finite element mesh of the driveline
SEDSG006_2.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus a part of the data in SEDSG006_1.dat
SEDSG006_2_ref.dat	Input data	Provided as a back up: This file contains all the data generated in this example plus the data in SEDSG006_1.dat
SEDSG006_2.dsg	DSG file	Generated by Design Studio. This file is the Design Studio database file.
SEDSG006_200.DMIG	DMIG file	Generated using Genesis within Design Studio. This file contains the superelement reduction.
SEDSG006_200_ref.DMIG	DMIG file	Provided as a back up: These file contains the superelement reduction and is the same file as SEDSG006_200.DMIG
SEDSG006_3.dat	Input data	Generated by Design Studio to run Genesis. This file contains all the data generated in this example plus a part of the data in SEDSG006_1.dat
SEDSG006_3_ref.dat	Input data	Provided as a back up: This file contains all the data generated in this example plus a part of the data in SEDSG006_1.dat
SEDSG006_3_dsg.dat	DSG file	Generated by Design Studio to run Design Studio. This file contains all the data generated in this example plus a part of the data in SEDSG006_1.dat
SEDSG006_3_dsg.out	DSG file	Generated using Genesis within Design Studio. This file contains FE post-processing results.
SEDSG006_3_dsg.pch	punch file	File generated using Genesis within Design Studio. This file contains the element force for post-processing.

12.6.1 Part 1. Superelement reduction using the Craig-Bampton modes

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `SEDSG006_1.dat`

Create new grids to perform the reduction

To capture the structure behavior is good to have as many degrees of freedom as number of modes * 6. So, you will create extra degrees of freedom.

3. Select the **Analysis** tab
4. From the category chooser, select **Grids**
5. Push the **New Grids** button from the Edit Menu toolbar
6. Enter `44` for the Number of New Grids

   Accept the default **Along a line** option. 44 grids for 44 modes as found in the Example 1.
7. Push **Next>**
8. Enter `200.0` for **Base X1**
9. Enter `1.0` for **Offset U1**
10. Enter `0.001` for **Offset Scale Factor**

    Note that since the offset scale factor is a small value compared to the size of the model, the 44 grids appear in the viewport as one grid.
11. Push the **Finish** button

Division of the model in two files

You will divide the model in two parts: the part to reduce and the part to keep.

12. Select the **Display** tab
13. Push the **Show / Hide Groups** button
14. Turn off (by selecting the properties ID or the parts on the Viewport window) the parts `Tube` (`PSHELL 3` and `PSHELL 4`), the Box (`PSHELL 6`) and the `RBE RBE2` elements.
15. From the main menu bar, select **File** → **Export** → **Input data...**
16. Name the file `SEDSG006_3`

17. Check the option Only Export Visible Groups/Elements
18. Push the **Save** button
19. In a text editor, open both files `SEDSG006_1.dat` and `SEDSG006_3.dat` and compare them.
20. Select the **Analysis** tab
21. From the category chooser, select **Elements**
22. Push the **Select All** button
23. From the Edit menu toolbar, select the **Delete Elements** button
24. Select the **Display** tab
25. Push the **Invert All** button
26. Push **Up**

Create a set of Independent degrees of freedoms (ASET)

You will define the interface of the superelement reduction.

27. Select the **Analysis** tab
28. From the category chooser, select **Grid-Component Sets**
29. Push the **New Grid-Component Set** button from the Edit Menu toolbar
30. Enter `ASET` for the **Name**
31. Select the Type **ASET**
32. Push **Next>**
33. Type in Select by Grid ID: `10000001-10000007`
34. Push the **Add** button

    Verify that there is the message "7 grids selected".
35. Push the **Set Components** button checking the Components are `123456`

    Verify that there is the message "42 total dof on 7 grids"
36. Push the **Finish** button
37. Right click the Viewport, Select **Clear** → **All**

Create a set of generalized degrees of freedoms (QSET)

You will define the interface of the superelement reduction.

38. Push the **New Grid-Component Set** button from the Edit Menu toolbar

39. Enter `QSET` for the **Name**
40. Select the Type **QSET**
41. Push **Next>**
42. In the Viewport, select the grids you have created earlier

    Verify that there is the message "44 grids selected".
43. Push the **Set Components** button checking the Components are `123456`

    Verify that there is the message "264 total dof on 44 grids".
44. Push the **Finish** button
45. Right click the Viewport, Select **Clear** → **All**

Create the Loadcase to perform the reduction

46. From the **Analysis** category chooser, select **Loadcases**
47. Select `Loadcase 1`
48. From the Edit menu toolbar, select the **Delete Loadcase** button
49. Push the **New Loadcase** button from the Edit Menu toolbar
50. Enter `Reduction` for the **Name**
51. Select the **Normal Modes** radio button
52. Push **Next>**
53. Push **Next>**
54. From the **Eigenvalue Method** category chooser, select `50 Method 50`
55. From the **ASET** category chooser, select `1 ASET`
56. From the **QSET** category chooser, select `2 QSET`
57. From the **Craig-Bampton Method** chooser, select `50 Method 50`
58. Push the **Finish** button

Setup the Genesis Options

59. From the main menu bar, select **Genesis** → **Options...**
60. In the **Output Control** tab, select **Print Module Times: Both**
61. In the **Output Control** tab, select **Analysis Output: First & Last**
62. In the **Analysis Control** tab, select **Automatic SPC (AUTOSPC): YES**
63. Push the **Advanced** button

64. Check the **Reduce mode**
65. Push the **Close** button
66. Push the **Apply** button

Export the Input file

67. From the main menu bar, select **File** → **Export** → **Input Data...**
68. Name the file `SEDSG006_2`
69. Push the **Save** button

Quit Design Studio

70. From the main menu bar, select **File** → **Quit**
71. Push the **Don't Save** button

Modify the reduction file

72. In a text editor, open the file `SEDSG006_2.dat`
73. Type `BOUNDARY = 1` just before `SUBCASE 2`
74. Type `MAA = DMIG` in the `SUBCASE 2`
75. Type `KAA = DMIG` in the `SUBCASE 2`
76. Save the file
77. Close the file

Perform the reduction

78. Run the file `SEDSG006_2.dat` directly from Genesis

Result of the reduction

79. In a text editor, Open the file `SEDSG006_200.DMIG`

    This file contains the product of the reduction (Mass and Stiffness).
    Verify that the name of matrix Mass and Stiffness are respectively `M0000002` and `K0000002`

12.6.2 Part 2. Driveline - Analysis Using Imported Superelement

Modify the condensed file

80. In a text editor, open the file `SEDSG006_3.dat`
81. Type `M2GG = M0000002` as an Solution Control command
82. Type `K2GG = K0000002` as an Solution Control command
83. Type `INCLUDE 'SEDSG006_200.DMIG'` after `BEGIN BULK`

    If you don't have this file, rename the provided file:
    `SEDSG006_200_ref.DMIG` to `SEDSG006_200.DMIG`
84. Save the file
85. Close the file

Start Design Studio

86. Start Design Studio
87. Import the Genesis data file: `SEDSG006_3.dat`

Create an element set

To plot the force in an element, you need to create an element set which contains this element.

88. Select the **Analysis** tab
89. From the category chooser, select **Element Sets**
90. Push the **New Element Set** button from the Edit Menu toolbar
91. Enter name `Set1`
92. Enter `11000004` in the **Select by Element ID** field
93. Push the **Add** button

    Verify that there is the message "1 element selected".
94. Push the **Finish** button

Create the Loadcase to plot the Element Force

You will learn how to set up a file in order to plot the Force in an element.

95. From the **Analysis** category chooser, select **Loadcases**
96. Select `Loadcase 1`

97. From the Edit menu toolbar, select the **Delete Loadcase** button
98. Push the **New Loadcase** button from the Edit Menu toolbar
99. Enter `Element_Force` for the **Name**
100. Select the **Modal Frequency Response** radio button
101. Push **Next>**

    In this case, there are nor SPC neither MPC. In your model, you may have one of them or both. Don't forget to select them.

102. Push **Next>**
103. From the **Eigenvalue Method** category chooser, select `50 Method 50`
104. Push **Next>**
105. From the **Dynamic Load Set** category chooser, select `1 DLoad Set`
106. From the **Loading Frequency Set** category chooser, select `2 Frequency Set`
107. From the **Modal Damping** category chooser, select `3 Modal Damping`
108. Push **Next>**
109. From the first **Element Force** category chooser, select **Post**
110. From the second **Element Force** category chooser, select `Set 1`
111. Push the **Finish** button

Setup the Genesis Options

112. From the main menu bar, select **Genesis** → **Options...**
113. In the Output Control tab, select **Print Module Times: Both**
114. In the Output Control tab, select **Complex Output: Polar**
115. In the Analysis Control tab, select **Automatic SPC (AUTOSPC): YES**
116. Push the **Apply** button

Export the input file

117. From the main menu bar, select **File** → **Export** → **Input data...**
118. Name the file `SEDSG006_3.dat`
119. Push the **Save** button

Analyze the structure using Genesis

120. From the main menu bar, select **Genesis** → **Single Analysis**
121. Study the **Genesis Console Output**; when done, push the **Close** button

Import the Post-Processing Files

122. From the main menu bar, select **File** → **Import** → **Punch Results...**
123. Select the `SEDSG006_3_dsg00.pch` file
124. Push the **Open** button

Post-Processing the Results (Element Force)

125. Select the **Post** tab
126. Push the **Freq. Resp. Plot** button
127. Push the **New Freq. Resp. Plot** button from the Edit Menu toolbar
128. Push the **Magnitude + Phase** radio button
129. Push **Next>**
130. Push the + button
131. Choose `Cycle 0 Loadcase 2 Elas Force`
132. Push **Next>**
133. Choose `Element 11000004`
134. Push **Next>**
135. Find the maximum value of the 3 highest peaks by highlighting around the maximum frequencies.

    Maximum and minimum responses between the highlighted loading frequency range along with loading frequency values are shown in the bottom left corner in the plot. Write down the Maximum responses values

Frequency (Hz)	Magnitude Reference Solution (1)	Magnitude Solution (2)
340	6.966	
440	18.23	
910	9.446	

(1) Result from plotting `SEDSG006_3_ref_dsg00.pch`
(2) Result from plotting `SEDSG006_3_dsg00.pch`

136. Push the **+** button
137. Right click on the graph
138. Push the **Save Image...** button
139. Enter the name `SEDSG006_3`
140. Push the **Save>** button
141. Push the **Finish** button
142. Push the **Close** button
143. Push the **Up** button

Quit Design Studio

144. From the main menu bar, select **File** → **Quit**
145. Push the **Don’t Save** button

CHAPTER 13

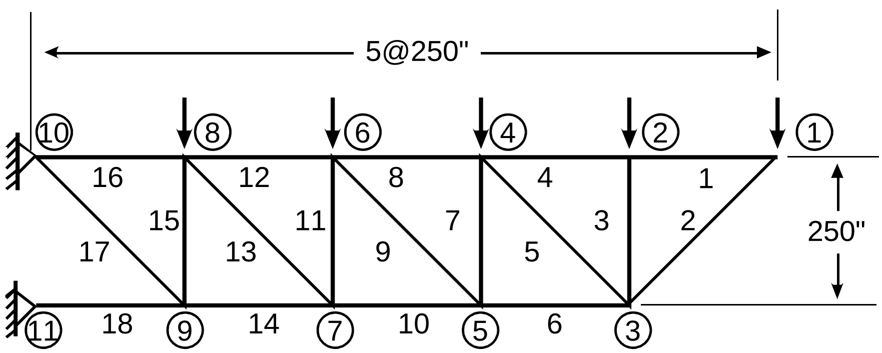
User Response Examples

13.1 Incorporating User-defined Subroutines (DRESP3)

Introduction

The purpose of this example is to develop user-defined subroutines and use them within Genesis for optimization. User-defined subroutines are helpful when users require performing an analysis or computation that is not intrinsically available within Genesis but utilizes information calculated during each Genesis optimization run. In this example, the design of a 2-D truss is used as an example. The truss members need to be designed to reduce overall weight while satisfying stress and Euler-Buckling constraints. For this example, the buckling constraints are calculated in a user-defined subroutine.

A 18-member truss is shown in the figure below. It is clamped at grids 10 and 11. The truss is subject to 5 point loads of 20000.0 as shown.



The numbers in the circle represent grid ID’s while the numbers next to the individual rods represent element IDs.

Background on Euler-Buckling Constraints:

To avoid Euler buckling of the rod elements, the force in each element must be constrained to be less in magnitude than the element’s Euler buckling load. This constraint can be expressed as:

$$F \geq F_E = -\frac{\pi^2 EI}{L^2}$$

The example problem uses rod elements with circular cross sections for the individual truss members. For a circular cross section,

$$I = \frac{A^2}{4\pi}$$

Therefore, the constraint can be rearranged as:

$$B = \frac{FL^2}{A^2} \geq \frac{-\pi E}{4} = -7.854 \times 10^6$$

where the rod length is calculated as:

$$\mathrm{L} = \sqrt{(\mathrm{X1}-\mathrm{X2})^2 + (\mathrm{Z1}-\mathrm{Z2})^2}$$

Thus, the formula for Euler Buckling value is:

$$B = \frac{F[(\mathrm{X1}-\mathrm{X2})^2 + (\mathrm{Z1}-\mathrm{Z2})^2]}{\mathrm{A}^2}$$

The above expression is calculated in a user-defined subroutine, where F, A, X1, Z1, X2, and Z2 are obtained from the Genesis output and input to the subroutine. The subroutine outputs the value for B.

The following optimization problem will be created, solved and post-processed:

Minimize Mass

Subject to:

-20,000.00 ≤ Axial stress at end B ≤ 20,000 psi (all elements)

$$\left(B = \frac{[(\mathrm{X}i-\mathrm{X}j)^2 + (\mathrm{Z}i-\mathrm{Z}j)^2]}{(\mathrm{A}p)^2}\mathrm{F}\right) \geq -7.854 \times 10^6$$

The above constraint applies for all elements, *i*,*j* are the two grid ID's for each element, *p* represents the associated element's property which provides the cross sectional area.

Designable region:

Cross-sectional areas of all truss members (sizing optimization)

Coordinate locations of Grids 3, 5, 7, 9 (shape optimization)

Solution Methodology

For this example, solution of the optimization problem using a user defined subroutine requires completion of 2 main stages:

1. Prepare the user subroutine, compile and create shared libraries (Example 1a)

2. Configure the problem in Design Studio and run Genesis (Example 1b)

The required steps for carrying out the above stages are presented in individual examples (Example 1a and 1b). Post processing of the optimization results is also discussed in Example 1b.

Note on example input file `URDSG001.dat`: The provided input data file used in this example already has all sizing variables defined previously. It also has most shape change perturbations already defined. In this example you will create perturbations on only one grid.

Finally, DRESP3 statements are used to define responses obtained from user-supplied subroutines. For each truss element, a DRESP3 statement is required to define the euler-buckling constraint. In order to reduce the time for completing this example, the input data file includes DRESP3 statements for 15 of the 18 elements. You will create the remaining DRESP3 data using Design Studio in Example 1b.

Example ID

URDSG001

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
URDSG001.c	Source Code	Provided: C Source code for user defined subroutine
URDSG001.f	Source Code	Provided: Fortran Source code for user defined subroutine
URDSG001.dll	Shared Library	Created: Dynamic Linked Library created for user defined subroutine in Windows 32 bit architecture
URDSG001.so	Shared Library	Created: Shared Library created for user defined subroutine in Linux 64 bit architecture
URDSG001.dat	Input data	Provided: The Genesis input data file for the truss example problem
URDSG001_dsg.SHP	Output file	Generated using Genesis within Design Studio. This file contains the results of the shape optimization

13.1.1 Part 1. User-defined Subroutine

Introduction

The purpose of this example is for you, the user, to define your own subroutine, compile it and create a shared library to use in conjunction with Genesis. The example includes instructions to compile C and Fortran version of user defined subroutines on the Linux and Windows platforms. For compiling instructions on other platform types, contact your system administrator. (It is assumed that the user has some experience creating and compiling Fortran/C codes).

Create the Subroutines

1. Using a text editor, open either the `URDSG001.f` or `URDSG001.c` depending on your compiler
2. Study the Fortran or C subroutine provided. Special instructions to prepare these subroutines are provided below.

Fortran Subroutine:

The primary subroutine must have the following declaration. This declaration is independent of the operating system.

```
SUBROUTINE DRESP3 (IWHICH, VAR, N, VAL, IERR)
```

C Subroutine:

Depending on your operating system, the primary subroutine must have the following format declaration.

For Windows:

```
__declspec( dllexport ) void DRESP3(int *iwhich, double var[], int
*n, double *val, int *ierr)
```

For Solaris, Linux, HP-UX:

```
void dresp3_(int *iwhich, double var[], int *n, double *val, int
*ierr)
```

For AIX:

```
void dresp3(int *iwhich, double var[], int *n, double *val, int
*ierr)
```

Important: The `URDSG001.c` file provided has the Linux format declaration. Modify this declaration depending on you operating system

Notes on subroutine declarations:

- The variable "iwhich" stores the subroutine ID. You can use any integer between 1 and 9 to be pointed by iwhich. However, it is important to remember the choice of iwhich since it corresponds to the "Subroutine ID". This will be important when defining the design data in Design Studio.

- The "var" array stores the inputs to the subroutine. The order of inputs to this array always follows the same order used for DRESP3 data. Refer to Genesis manual for the default DRESP3 order of inputs. For this example, the order is: Cross section area of element (DVAR), X and Z coordinates of the element's two grids (DGRID) and the force on each member calculated during each run of Genesis (DRESP1).

- "n" is the number of entries in the var array. Corresponds to the total number of inputs being provided to the user defined subroutine.

- "val" is the output variable that is returned after each call to the user defined subroutine by Genesis.

- The provided subroutine has all calculations performed within the DRESP3 subroutine. However, this is not necessary. The primary subroutine has to be DRESP3, which can then call other subroutines. In order to use user defined subroutines with Genesis, it is necessary to have the primary DRESP3 subroutine in your code.

Compile Subroutines and Create Shared Libraries

3. Compile the code to create shared libraries.

The following compiler instructions are compiler and architecture specific. Please contact your system administrator for your specific compiler and architecture type.

Windows 32-bit machine:

Fortran Code (Intel Fortran 10.0 Compiler):

```
C:\Temp>ifort /dll /out:URDSG001.dll URDSG001.f
```

`URDSG001.f` is the source code, /dll creates the .dll and /out flag is used to name the .dll file.

C Code:

```
C:\Temp>cl /LD URDSG001.c /FeURDSG001.dll
```

`URDSG001.c` is the source code. The /LD flag creates the .dll, the /Fe flag is used to name the dll file.

At the end of this step, a `URDSG001.dll` shared library should be created. This will be used in Example 1b next.

Linux 64-bit machine:

Fortran Code:

```
f90 -G -m64 -PIC -dalign -o URDSG001.so URDSG001.f
```

C Code:

```
cc -m64 -fPIC -shared -o URDSG001.so URDSG001.c
```

At the end of this step, a `URDSG001.so` or `URDSG001.dll` shared library should be created. This will be used in Part 2 next.

13.1.2 Part 2. Configure Design Studio and run Genesis

Introduction

In this example, the design data and optimization problem is set up in Design Studio. The shared library created in Part 1 (*URDSG001.so* in Linux, *URDSG001.dll* in Windows) is incorporated into the design data to evaluate Euler-Buckling constraints. It is a good idea to create a new folder for this example on your computer. Save the input data file (`URDSG001.dat`) and the shared library in this folder.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `URDSG001.dat`

Check the Sizing Design Variables

3. Select the **Design** tab
4. From the category chooser, select **Sizing**
5. Notice that all four properties have a little hammer next to them. This implies that all these properties are designable.
6. Select PROD1
7. Push the **Modify Sizing Design** button from the Edit Menu toolbar

   You will see Area assigned a design variable A1.
8. Push the **Finish** button.

   The design variables for the cross sectional areas are called A1, A2, A3 and A4. You can verify this by selecting the different properties and clicking **Modify Sizing Design** button from the Edit Menu toolbar. The initial values and bounds for all design variables are shown below.

DV Name	Initial Value	Lower Bound	Upper Bound
A1	10.0	5.0	40.0
A2	21.65	10.0	40.0
A3	12.5	0.5	40.0
A4	7.07	1.0	40.0

Table 13-1 Design Variables for Sizing

Creating a Perturbation with Shape Morphing Sets

Note: You will only provide shape perturbations on Grid 3. The input file includes the remaining shape perturbations.

9. From the **Design** category chooser, select **Shape Morphing Sets**

   Notice ShapeTwoX, ShapeTwoZ etc shape morphing sets already existing in the input data. Click on these perturbations to see where and how they act on the truss. Similar perturbations in the X and Z directions are created for Grid 3.

10. Push the **New Shape Set** button from the Edit Menu toolbar
11. Enter `ShapeOneX` in the Name field
12. Check the **Raw Morphing Set** radio button

    You will create a Design Variable and then come back where you are now. Do not push the **Cancel** button at this point

13. From the **Design** category chooser, select **Design Variables**

    Notice that the 10 design variables listed here already exist in the input data: A1, A2, A3, A4, X5, Z5, X7, Z7, X9 and Z9.

14. Push the **New Design Variable** button from the Edit Menu toolbar
15. Enter `X3` as Name
16. Check the default **Independent Design Variables** radio button
17. Push **Next>**
18. Enter `0.0` for **Initial Value**, `-200.0` for **Lower Bound** and `200.0` for **Upper Bound**
19. Push the **Advanced Parameters** button
20. Enter `2.0` as **Minimum Move Limit**
21. Push **Next>**
22. Push the **Finish** button

    Verify that there are a total of 11 design variables in the list with the last one having an * next to it.

23. From the **Design** category chooser, select **Shape Morphing Sets**

    You will be brought back to where you had left

24. Select the design variable `X3`
25. Push **Next>**
26. In the **Select by Grid ID** field, enter `3`

27. Click **Add**.

    Verify that “1 grid selected” is displayed

28. Enter `1.0` in the **X** field, `0.0` in **Y** field and `0.0` in the **Z** field.
29. Enter `50.0` in the **Magnitude** field
30. Push the **Add Perturbation** button

    Verify that “1 perturbations on 1 grids” appears at the bottom of the window.

31. Push the **Finish** button

    Repeat Steps 11-32 to create a perturbation using the data from the following table

Name	Applied on grid	Perturbation Direction	Magnitude	Associated Design Variable
`ShapeOneZ`	3	+Z	50.0	Z3

You need to create the Z3 design variable. For the Initial Value, the Lower Bound and the Upper Bound, keep the same value as X3

Verify there are a total of 8 perturbations (Number of **Shape Morphing Sets**) (4 in the X direction and 4 in the Z direction)

Defining the Objective

32. From the category chooser, select **Objectives**
33. Push the **New Objective** button from the Edit Menu toolbar
34. Enter `Mass` as the name
35. Check the **Mass** radio button as Response Type and accept the default of **Entire Model** in the category chooser
36. For the Objective Definition Switch, accept the default selection of **Min**
37. Push the **Finish** button

Defining the Constraints

There are a total of 19 constraints - 1 overall stress constraint and 18 Euler Buckling constraints on each truss member. 15 of the 18 Euler Buckling constraints have already been defined in the input data; you will define the remaining 3. The overall stress constraint is defined first.

Overall Stress Constraint Definition:

38. From the category chooser, select **Constraints**

    Notice there already are 15 constraints defined. There are the existing Euler Buckling constraints.

39. Push the **New Constraint** button from the Edit Menu toolbar
40. Enter `Stress1` as the name
41. Check the **Stress** radio button as Response Type

    Keep the default "Selected Groups".
42. Enter `-20000.0` as the **Lower Bound** and `20000.0` as the **Upper Bound**
43. Push **Next>**
44. Select all the 4 properties

    Hold the Ctrl button and click on all 4 properties
45. Push **Next>**
46. Select **Stress End B** as the option in the PROD Stress field
47. Push **Next>**
48. Select the existing Loadcase.
49. Push the **Finish** button

    Verify there is a new constraint called Stress1.

Euler Buckling Stress Constraints Definition:

50. Push the **New Constraint** button from the Edit Menu toolbar
51. Select **Synthetic Response** as the Response Type
52. Enter `-7854000.0` as the **Lower Bound**
53. Push **Next>**

    There are no Synthetic Responses currently available for selection. We will create synthetic responses for euler buckling of each element next. Do not push the **Cancel** button at this point.
54. From the **Design** category chooser, select **Synthetic Responses**

    Notice there are several synthetic responses already defined. There have been used for the pre-defined 15 Euler Buckling constraints.
55. Push the **New Synthetic Response** button from the Edit Menu toolbar
56. Enter `B01` as the name of the synthetic response
57. Select the **User Library Subroutine (DRESP3)** option
58. Push **Next>**

    Recall the Euler Buckling constraint requires 4 inputs - the axial element force, the cross sectional area of each truss member, and the end coordinates of each element. These inputs will be entered next and combined with the user defined subroutine to determine the Euler Buckling constraints.

59. Push the + button
60. Check the **Fundamental Response...** radio button
61. Push **Next>**
62. Check the **More Response Types...** radio button
63. Push **Next**>
64. Select the **Force** radio button
65. From the **Force** category chooser, select **Selected Elements**
66. Push **Next>**
67. Enter `1` in the **Select by Element ID** field
68. Push the **Add** button

    Verify that there is 1 element selected

69. Push **Next>**
70. From the **PROD Force** category chooser, select default of **Force End A**
71. Push **Next>**

    You do not select the existing loadcase

72. Push **Next>**

    You will be brought back to the original Synthetic Response editing screen. The newly created argument to the user defined subroutine can be seen in the panel. This argument is the "Force" in the Euler Buckling equation.

73. Push the + button

    Next, we will define the cross-sectional area to be used in the Euler Buckling equation

74. Select the **Design Variable** option
75. From the category chooser, select A1

    It represents the cross-sectional area of all elements associated with PROD 1

76. Push **Next>**
77. Push the + button

    Next the two grids at the end points of element 1 are going to be defined. This requires definition of 2 X coordinates and 2 Z coordinates.

78. Check the **Grid X Coordinate** radio button
79. Enter `1` in the **Select by Grid ID** field

    Refer to the figure of the truss to determine grid ID 1. Verify that the Select by Grid ID: field has the number 1 appear in it.

80. Push the Enter key from your keyboard

81. Push **Next>**
82. Push the + button
83. Check the **Grid Z Coordinate** radio button

    Ensure that Grid ID 1 is still selected

84. Push **Next>**
85. Push the + button
86. Check the **Grid X Coordinate** radio button
87. Enter 2 in the **Select by Grid ID** field
88. Push the Enter key from your keyboard
89. Push **Next>**
90. Push the + button
91. Check the **Grid Z Coordinate** radio button

    Ensure that Grid ID 2 is still selected

92. Push **Next>**
93. Set the **Subroutine ID** drop down menu to 3
94. Push the **Finish** button

    Verify that newly created User subroutine (User SUB3) named B01 appears in the list of synthetic responses.

95. From the category chooser, select **Constraints**

    You will be back where you left off in the constraint definition.

96. Select the newly created synthetic response B01
97. Push **Next>**
98. Select the existing loadcase
99. Push the **Finish** button

    Verify that the list of constraints now includes a new constraint B01.

100. Repeat steps 53-102 for all elements 2 and 3 in the truss by referring to the following table

Selected Response Name	Cross Sectional Area	Element ID	Grid ID
B02	A2	2	1 3
B03	A3	3	2 3

After completion of all steps, there should be a total of 19 constraints - 18 Euler Buckling constraints (labeled B01 through B18) and 1 stress constraint (labeled Stress1).

Input Shared Library in Genesis Executive Control

101. From the main menu bar, select **Genesis** → **Options**
102. Select the **Design Control** tab
103. Push the **Advanced** button.
104. Push the **Browse** button next to the DRESP3 Shared Library field
105. Scroll to select the shared library (*`URDSG001.so`* for Linux, *`URDSG001.dll`* for Windows) created in Part 1.
106. Push the **Open** button
107. Push the **Close** button
108. Push the **Apply** button

Optimizing using Genesis

109. From the main menu bar, select **Genesis** → **Optimize**

    Study the results in the console output and the design history windows. Verify that the problem has converged and the final solution is feasible.

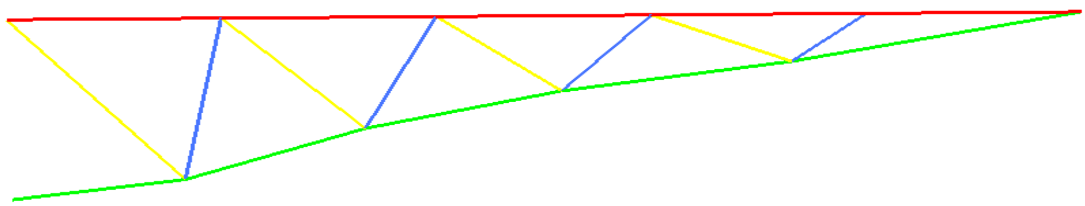
110. Push the **Close** button

Post-Processing the Results

111. From the main menu, select **File** → **Import** → **Punch/Output2 Results...**
112. Select the file `URDSG001_dsg.SHP`
113. Select the **Post** tab
114. Push the **Deform Mesh/Color Mesh** button

115. Select "Cycle 10 Shape Change"

You should now see the final shape of the truss in the viewport. The final shape is also shown in the figure below as a reference.



Quit Design Studio

116. From the main menu bar, select **File** → **Quit**

117. Push the **Don't Save** button

Study the Output File

118. On your computer, find the `URDSG001_dsg.out` file and open it in a text editor

119. Fill out the following table

	Initial Value	**Reference Final Solution**	**Final Value (Your Result)**
A1	10.0	12.295	
A2	21.65	39.98	
A3	12.5	12.101	
A4	7.07	4.1098	
Mass	6430.3	7809.9	
Maximum Constraint Violation %	409.3%	0.0%	

Note: Your final result might be slightly different due to precision differences.

13.2 Using Synthetic Responses (DRESP2)

Introduction

The purpose of this example is to learn how to create synthetic responses and use them in the objective and constraints. With this example you will learn that is possible to simultaneously improve multiple natural frequencies and separate them from each other.

The following Objective and Constraints will be created:

Maximize (3*Freq 7 + Freq 8 + Freq 9 + Freq 10)/6

Subject to:

Freq 7 $\geq$ 6.2 Hz (standard constraint)
Freq 8 $\geq$ 9.0 Hz (standard constraint)
Freq 9 $\geq$ 15.0 Hz (standard constraint)
Freq 10 $\geq$ 19.0 Hz (standard constraint)
Freq 8 - Freq 7 $\geq$ 2.0 Hz (synthetic constraint, to separate frequencies)
Freq 9 - Freq 8 $\geq$ 2.0 Hz (synthetic constraint, to separate frequencies)
Freq 10 - Freq 9 $\geq$ 2.0 Hz (synthetic constraint, to separate frequencies)

Example ID

`URDSG002`

Files Used in This problem

A list, of the key files provided and the ones that you will create during this example, is presented next. These files will be introduced during the example, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`URDSG002.dat`	`Input data`	Provided: The Genesis input data file for the example problem
`URDSG002_dsg.dat`	`Input file`	Generated using Genesis within Design Studio. This file contains the data in `URDSG002.dat` along with the data created in this example
`URDSG002_dsg.SHP`	`Output file`	Generated using Genesis within Design Studio. This file contains the results of the shape optimization
`URDSG002_ref.dat`	`Output file`	Provided for Reference. This file contains the same data as `URDSG002_dsg.dat`

Start Design Studio

1. Import the Genesis data file: `URDSG002.dat`

   This file has shape optimization data already created.

Preview the applied shape changes

2. From the **Post** tab, select the **Deform Mesh/Color Mesh** button
3. Select the **Oscillate** radio button
4. Under the **Color Mesh**, select **Filled Contours** radio button
5. Under the **Color Mesh**, select each **Shape Morphing Set Preview**

   Notice how each of the morphing sets is used to define the shape/location of the members in the frame.
6. Push the **Up** button

Define the Objective

7. Select the **Design** tab
8. From the **Design** category chooser, select **Objectives**
9. Push the **New** button to create a new Objective
10. Enter `FreqComb` for the **Name**
11. Select **Max** as Objective Definition Switch
12. Select **Synthetic Response** as Response Type
13. Push the **Next>** button

    You will need to create a synthetic response with a new trail and then come back to where you are now. There is no need to push **Cancel** now. (This is due to the ability of Design Studio to work simultaneously with multiple trails).

Create a Synthetic function

14. From the Design category chooser, select **Synthetic Responses**
15. Push the **New** button to create a new synthetic response
16. Enter `FreqComb` for the **Name**
17. Leave **User Function (DRESP2)** as the type
18. Push **Next>**
19. Push the **+** button to create the first argument of the function

20. Push **Next>** (as there will no grids in our synthetic response)
21. Select **Frequency Mode Number** as the response type
22. Enter `7` in the box next to **Frequency Mode Number**
23. Push **Next>**
24. Select the exiting loadcase
25. Push **Next>**
26. Repeat steps 14 through 20 to create three more arguments.

    For argument 2 use: Mode = 8
    For argument 3 use: Mode = 9
    For argument 4 use: Mode = 10

27. Write the following equation:
28. **F** = `(3*Arg1 + Arg2 + Arg3+ Arg4)/6`
29. Push the **Finish** button, to finish the creation of the synthetic response

Return to the Objective Trail

30. From the Design category chooser, select **Objectives**

    Now you returned to the same point you where before going to the synthetic response creation trail. This time though, the list of synthetic responses contain the response you just created.

31. Select the `FreqComb` synthetic response
32. Push the **Finish** button

Set up the Direct Constraints

33. From the Design category chooser, select **Constraints**
34. Push the **New** button, to create a new constraint
35. Enter `F7` for the **Name**
36. Select **Frequency Mode number** as response type
37. Enter `7` in the box next to **Frequency Mode Number**
38. For the **Lower Bound** enter `6.2`

    Note that before entering a bound the **Next>** button is disabled

39. Push the **Next>** button
40. Select the existing loadcase
41. Push the **Finish** button

42. Repeat steps 29 through 36 to create three more direct constraints

    For constraint 2:
    change name from `F7` to `F8`, mode `7` to `8`. Use lower bound of `9.0`
    For constraint 3
    change name from `F8` to `F9`, mode `8` to `9`. Use lower bound of `15.0`
    For constraint 3:
    change name from `F9` to `F10`, mode `9` to `10`. Use lower bound of `19.0`

Set up the Synthetic Constraints

43. Push the **New** button to create a new constraint
44. Enter `F8-F7` for the **Name**
45. Select **Synthetic Response** as the response type
46. For the **Lower Bound** enter `2.0`
47. Push the **Next>** button

    You will need to create a synthetic response with a new trail and then come back to where you are now.

Set up a Synthetic function for a Constraint

48. From the Design category chooser, select **Synthetic Respones**
49. Push the **New** button to create a new synthetic response
50. Enter `F8-F7` for the **Name**
51. Leave **User Function (DRESP2)** as the type
52. Push the **Next>** button
53. Push the **+** button to create the first argument
54. Push the **Next>** button (as there will be no grids in this synthetic response)
55. Select **Frequency Mode Number** as the response type
56. Enter `8` in the box next to **Frequency Mode Number**
57. Push the **Next>** button
58. Select the existing loadcase
59. Push the **Next>** button
60. Repeats the last eight steps to create a second arguments.

    For the second argument use Mode number = 7.

61. Write the equation:

    F = `Arg1 - Arg2`

62. Push the **Finish** button, to finish the creation of the synthetic response

Return the Constraint Creation Trail

63. From the Design category chooser, select **Constraints**

    Now you returned to the same point you where before going to the synthetic response creation. This time though the list of synthetics responses contain the response you just created.

64. Select the newly created `F8-F7` synthetic response

65. Push the **Finish** button

66. Repeat steps 38 through 60 to create two more synthetic response constraints:

    For constraint 6:
    Where it says `8` use `9` and where it says `7` use `8` in both the name and mode number inputs.
    For constraint 7
    Where it says `8` use `10` and where it says `7` use `9` in both the name and mode number inputs.

Optimize the Structure Using Genesis

67. From the main menu bar, select **Genesis** → **Optimize**

68. Study the **Design History**, when done push the **Close** button

69. Study the **Genesis Console Output**, when done push the **Close** button

70. Open the output file. Study the values of the objective and constraints and complete the following table:

Type	Function definition	Initial Value	Final Value	Gain%
Objective	(3*Freq7+Freq8+Freq9+Freq10)/6			
Constraint 1	Frequency 7			
Constraint 2	Frequency 8			
Constraint 3	Frequency 9			
Constraint 4	Frequency 10			
Constraint 5	Freq 8 - Freq 7			
Constraint 6	Freq 9 - Freq 8			
Constraint 7	Freq 10 - Freq 9			

Import the Shape Changes File

71. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
72. Select the `URDSG002_dsg.SHP` file
73. Push the **Open** button

Post-Processing the Results (Shape Changes)

74. Select the **Post** tab
75. Push the **Deform/Mesh Color Mesh** button
76. Push the **Filled Contours** radio button
77. Select a Shape Change for the last design cycle in the **Color Mesh** frame
78. Push the **Up** button

Quit Design Studio

79. From the main menu bar, select **File** → **Quit**
80. Push the **Don't Save** button

CHAPTER 14

Analysis - Meshing Examples

- Ten Rod Truss Model
- Cantilever Beam with Solid Elements
- Simply Supported Plate with Shell Elements
- Changing Element Groups
- Composite Plate
- Hollow Tube with Shell Elements
- Generating a Shell Skin on a Solid Part
- Automatic Creation of Ribs - Bar Elements
- Automatic Creation of Ribs-QUAD Elements
- Mesh Refinement
- Three Story Facade with Panel

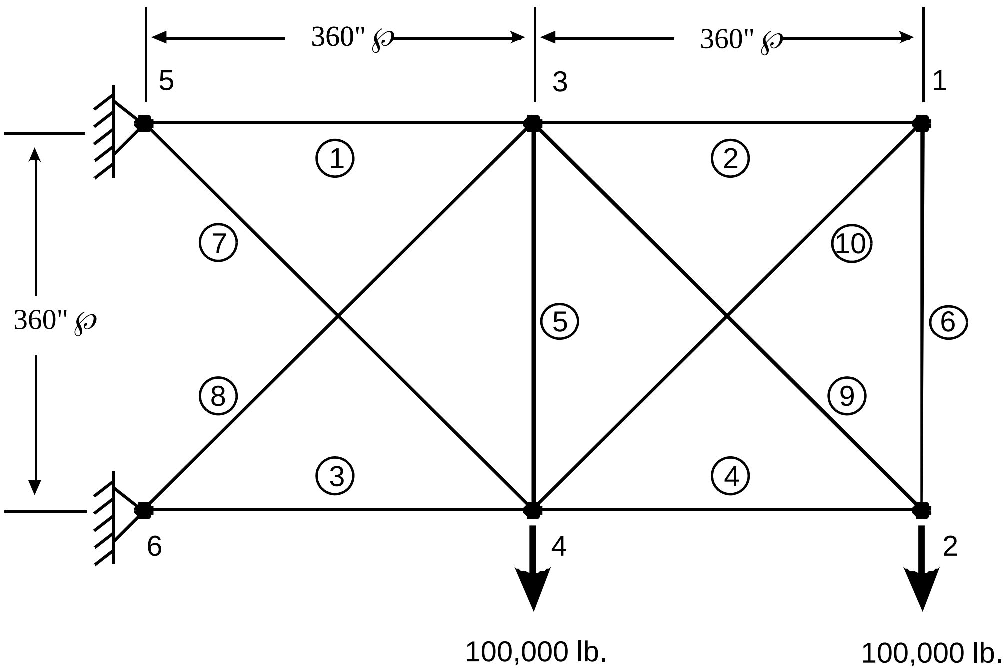
14.1 Ten Rod Truss Model

Introduction

The purpose of this exercise is to create a finite element model of a ten rod truss shown in the figure below and analyze it for a static load case.

Problem Statement:

Find the nodal displacements and element stresses in the ten rod truss as shown in the figure below.



Problem Description:

1. Planar truss structure modeled with 10 CROD elements.
2. Section properties: Area=5.0 in^2
3. Material: E=1.0E7 psi.
4. Loading: Two 100,000 lb. concentrated loads applied simultaneously to GRIDS 2 and 4.

Example ID

AMDSG001

Files Used in This problem

A list, of the key files provided or created are presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`AMDSG001.dat`	`Input data`	Created: Input file exported at the end of the exercise
`AMDSG001_ref.dat`	`Input data`	Provided: Reference input file ready to be analyzed
`AMDSG001_dsg00.pch`	`DSG file`	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio

Create a New Material for the rods

2. From the **Analysis** category chooser, select **Materials**
3. Push the **New Material** button from the Edit Menu toolbar
4. Enter `Rod_Material` in the name field
5. Check the **Isotropic (MAT1)** radio button
6. Push **Next**>
7. For the Young modulus **E**, enter `1E7`
8. For the Poisson coefficient **Nu**, enter `0.3`
9. Push the **Finish** button

Create a New Group

10. From the **Analysis** category chooser, select **Group Properties**
11. Push the **New Group Property** button from the Edit Menu toolbar
12. Enter `Rods` in the name field
13. Select **PROD** in the **Type** category chooser
14. Choose the Color you want the group to be displayed in
15. Push **Next**>
16. Select the material created from the drop-down list box for **Material**
17. Enter `5.0` for the **Area**

18. Push the **Finish** button

Create grid points for the model

19. From the **Analysis** category chooser, select **Grids**
20. Push the **New Grids** button from the Edit Menu toolbar
21. Enter `1` as **Number of New Grids**
22. Accept the default option **Along a Line**

    Design Studio is capable of creating multiple grids at the same time. The above two options can be used for creating multiple grids.

23. Push **Next>**
24. Enter **Base X1** = `720.0`, **Base X2** = `360.0` and **Base X3** = `0.0`

    Base X1, **Base X2** and **Base X3** correspond to the x, y and z coordinates of the grid point when a single grid point is created.

25. Push the **Finish** button
26. Repeat steps 20-25 using the values in the following table:

Grid	Base X1	Base X2	Base X3
2	720.0	0.0	0.0
3	360.0	360.0	0.0
4	360.0	0.0	0.0
5	0.0	360.0	0.0
6	0.0	0.0	0.0

To see all the grids create, select the **Fill** icon button in the Viewport

Create Rod elements

27. From the **Analysis** category chooser, select **Elements**
28. Push the **New Elements** button from the Edit Menu toolbar
29. Check the **Define New Original Elements** radio button
30. Push **Next>**
31. Select the **PROD** group created earlier
32. Push **Next>**

33. Select the **Point by Point** radio button

    Point by Point can be used to create the domain by entering the points.
    You can also select the **Drag out size** radio button. It is used to create the region by clicking in the Viewport.

34. Select the Lines icon in the **Region Definition options** for creating rod elements
35. Select the **Pick existing grids** radio button
36. Type in 5 in the adjacent textbox for the first grid ID and press the enter key on the keyboard
37. Replace 5 with 3 in the textbox for the second grid ID and press the enter key

    You can alternatively, pick the grid points on the viewport to make the selection.

38. Repeat steps 36-38 using the following table to define the all the rod elements.

Element ID	First Grid ID	Second Grid ID
2	3	1
3	6	4
4	4	2
5	3	4
6	1	2
7	5	4
8	6	3
9	3	2
10	4	1

39. Push the **Finish** button

Define the static loads

40. From the **Analysis** category chooser, select **Static Loads**
41. Push the **New Load Set** button from the Edit Menu toolbar
42. Check the **Force, Moment, Pressure, SPCD** radio button
43. Push **Next>**
44. Push the **Select None** button to make sure none of the grids are selected

45. Select Grids `2` and `4` to define the force by picking them in the viewport or entering in the text box
46. Define the direction of the loading by entering the **X**, **Y** and **Z** components of the force. In this case the force is acts in the direction of the negative Y-axis. So enter `0.0,-1.0,0.0` for the components
47. Enter `100000.0` as **Magnitude** value
48. Push the **Add Force** button
49. Push the **Finish** button

Define the boundary conditions (Single point constraints)

50. From the **Analysis** category chooser, select **Grid-Component Sets**
51. Push the **New Grid-Component Set** button from the Edit Menu toolbar
52. Enter `BoundaryConditions` in the name field
53. Check the **Single-Point Constraints** radio button
54. Push **Next>**
55. Push the **Select None** button to clear the previous grid selection
56. Select grids `5` and `6` to be constrained by picking them in the viewport or entering in the textbox.
57. For **Components**, enter `123456` to restrain

    1 for X direction, 2 for Y direction, 3 for Z direction, 4 for the rotation around X axis, 5 for the rotation around Y axis, 6 for the rotation around Z axis
58. Push the **Set Components** button

    Make sure 12 dofs are constrained on 2 grids
59. Push the **Finish** button

Create a Static Loadcase

60. From the **Analysis** category chooser, select **Loadcases**
61. Push the **New Loadcase** button from the Edit Menu toolbar
62. Enter `Loads` in the name field
63. Make sure the **Static** radio button is selected
64. Push **Next>**
65. From the **SPC** category chooser, select the Grid-Component-Set created

66. Push **Next>**
67. From the **Load Set** category chooser, select the Static Load created
68. Push **Next>**
69. Select **Post** and **All** for the Displacement options
70. Select **Post** and **All** for the Element Stresses options

    This is done to post the results of the analysis into a punch file that can be read into Design Studio for post processing.

71. Push the **Finish** button

Define AUTOSPC parameter

72. From the main menu bar, select **Genesis** → **Options**
73. Select the **Analysis Control** tab
74. Check **Automatic SPC (AUTOSPC)**
75. From the category chooser select **Yes** to automatically constrain degrees of freedom without stiffness

    To have AUTOSPC=NO, from the category chooser, select **No**

76. Push the **Apply** button

Export the Input File

77. From the main menu bar select **File** → **Export** → **Input Data...**
78. Enter `AMDSG001`
79. Push the Save button

Save the Design Studio file

80. From the main menu bar, select **File** → **Save As...**
81. Enter `AMDSG001` as the Filename and push **Save (**as a Design Studio File)

Analyze the structure using Genesis

82. From the main menu bar, select **Genesis** → **Single Analysis**

Import the Post-Processing Files

83. In the **Genesis Console Output** window, push the **Import Post...** button

    Alternatively, one can import post-processing files by selecting **File** → **Import** → **Punch/Output2 Results...** from the main menu bar

84. Select the `AMDSG001_dsg00.pch` file
85. Push the **Import** button
86. In the **Genesis Console Output** window, push the **Close** button

Post-Process Displacements Results

87. Select the **Post** tab
88. Push the **Deform Mesh/Color Mesh** button
89. Select the `Cycle 0 Loadcase 1 Displacement` Result
90. Push the **Ramp** button
91. Push the **Filled Contours** radio button in the **Color Mesh**
92. In the **Color Mesh** list, select the same displacement Result
93. Right-click in the viewport, select **List Top Ten**
94. Right-click in the viewport, select **List Bottom Ten**

    List Top Ten and **List Bottom Ten** options can be used to display the largest ten and smallest ten values of the displacements. The listing of the results is displayed in the Messages window

Visualize the Stresses Results

95. Select `Cycle 0 Loadcase 1 Rod Stress` Result
96. Push the **Filled Elements** button in the **Color Mesh**
97. Push the **Options...** button. Slide the bar to hide elements with low values
98. Select any of the elements to display its stress value in the messages window
99. Push the **Close** button
100. Push the **Up** button

Quit Design Studio

101. From the main menu bar, select **File** → **Quit**

14.2 Cantilever Beam with Solid Elements

Introduction

The purpose of this exercise is to demonstrate the procedure for creating a solid finite element mesh in Design Studio.

Problem Statement:

This example demonstrates the creation of a finite element mesh easily by creating the grids and elements at the same time.

A cantilever beam is modeled using solid HEXA elements. This model is subject to point loads at the end of the beam. The displacements of the grids and the element stresses are obtained as results.

Note: The same technique used here to generate the 3-D mesh can be used for creating 1-D as well as 2-D meshes.

Example ID

`AMDSG002`

Files Used in This problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`AMDSG002.dat`	`Input data`	Created: Input file exported at the end of the exercise
`AMDSG002_ref.dat`	`Input data`	Provided: Reference file ready to be analyzed
`AMDSG002_dsg00.pch`	`DSG file`	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio

Create a Material for the beam

2. Select the **Analysis** tab
3. From the category chooser, select **Materials**

4. Push the **New Material** button from the Edit Menu toolbar
5. Enter `Beam_Material` in the name field
6. Check the **Isotropic (MAT1)** radio button
7. Push **Next**>
8. For the Young modulus **E**, enter `210000`
9. For the Poisson coefficient **Nu**, enter `0.28`
10. For the density **Rho**, enter `7.8E-9`
11. Push the **Finish** button

Create a Group

12. From the **Analysis** category chooser, select **Group Properties**
13. Push the **New Group Property** button from the Edit Menu toolbar
14. From the **Type** category chooser, select PSOLID

    You can choose PSHELL, etc. to create 2-D meshes, or PROD, etc. to create 1-D meshes
15. For Name, enter `Beam`
16. Push **Next**>
17. Select the Material created in the previous section
18. Push the **Finish** button

Create a finite element mesh

Now you will create a mesh of the beam with the following 10*6*80 elements

19. From the category chooser, select **Elements**
20. Push the **New Elements** button from the Edit Menu toolbar
21. Check the **Define New Original Elements** radio button
22. Push **Next**>
23. Select the PSOLID group from the list
24. Push **Next**>
25. Select the **Point by Point** radio button
26. Select the **Hexas** icon as **Region Definition options**
27. Enter `0.0` for **X**, `0.0` for **Y** and `0.0` for **Z**
28. Push the `Enter` key on your keyboard

Repeat steps 28 and 29 to define the other corners of the Hexas domain in order of the following table

Corner	X	Y	Z
2	0.0	0.0	10.0
3	0.0	4.0	10.0
4	0.0	4.0	0.0
5	80.0	0.0	0.0
6	80.0	0.0	10.0
7	80.0	4.0	10.0
8	80.0	4.0	0.0

29. Select the **Iso Right-Front-Top** view icon in the Viewport to view the domain created
30. In the list, select the `Hexa` you just created
31. Push the **Subdivide** button
32. Enter `10` as Elements in 1st Dimension
33. Enter `4` as Elements in 2nd Dimension
34. Enter `80` as Elements in 3rd Dimension

    The first dimension is displayed by a red line on the shape domain in the Viewport window.
    The second dimension is displayed by a green line.
    The third dimension is displayed by a blue line.
35. Push **Next>**
36. Push the **Finish** button

Create a Grid-Component-Sets (Boundary Conditions)

37. From the **Analysis** category chooser, select **Grid-Component-Sets**
38. Push the **New Grid-Component Set** button from the Edit Menu toolbar
39. Check the **Single-Point Constraints (SPC)** radio button
40. Push **Next>**
41. Select the **Right** view icon in the Viewport window to change the view
42. Select the first left vertical column of grids

    You should have 55 grids selected

43. For **Components**, enter `123`
44. Push the **Set Components** button

    You should have 165 total dof on 55 grids
45. Push the **Finish** button

Create a Static Load

46. From the **Analysis** category chooser, select **Static Loads**
47. Push the **New Load Set** button from the Edit Menu toolbar
48. Check the **Force, Moment, Pressure, SPCD** radio button
49. Select the **Iso Right-Front-Top** view icon in the Viewport window to change the view
50. Push **Next>**
51. Push the **Select None** button
52. Enter `33` in the **Select by Grid ID** field
53. Push the **Add** button
54. Enter `0.0` as **X** direction, `0.0` as **Y** direction and `-1.0` as **Z** direction.

    This defines the direction of the applied force (the negative Z direction)
55. Enter `20` for the **Magnitude**
56. Push the **Add Force** button
57. Push the **Finish** button

Create a Loadcase

58. From the **Analysis** category chooser, select **Loadcases**
59. Push the **New Loadcase** button from the Edit Menu toolbar
60. Enter `Loads` for the name
61. Check the **Static** radio button
62. Push **Next>**
63. From the **SPC** category chooser, select the Grid-Component-Set you have just created
64. Push **Next>**
65. From the **Load Set** category chooser, select the Static Load you have just created

66. Push **Next>**
67. Select **Post** and **All** for the Displacement options
68. Select **Post** and **All** for the Element Stress options
69. Push the **Finish** button

Export the Input File

70. From the main menu bar select **File** → **Export** → **Input Data...**
71. Enter `AMDSG002`
72. Push the **Save** button

Save the Design Studio file

73. From the main menu bar, select **File** → **Save As...**
74. Enter `AMDSG002` as the Filename and push **Save (**as a Design Studio File)

Analyze the structure using Genesis

75. From the main menu bar, select **Genesis** → **Single Analysis**

Import the Post-Processing Files

76. In the **Genesis Console Output** window, push the **Import Post...** button

    Alternatively, one can import post-processing files by selecting **File** → **Import** → **Punch/Output2 Results...** from the main menu bar
77. Select the `AMDSG002_dsg00.pch` file
78. Push the **Import** button
79. In the **Genesis Console Output** window, push the **Close** button

Post-Process Displacements Results

80. Select the **Post** tab
81. Push the **Deform Mesh/Color Mesh** button
82. Select the `Cycle 0 Loadcase 1 Displacement` Result
83. Push the **Ramp** button
84. Push the **Filled Contour** button in the **Color Mesh**

85. In the **Color Mesh** list, select the same Displacement Result
86. Right-click in the viewport, select **List Top Ten**
87. Right-click in the viewport, select **List Bottom Ten**

Visualize the Stresses Results

88. Select `Cycle 0 Loadcase 1 Solid Stress` Result
89. Push the **Filled Elements** button in the **Color Mesh**
90. Push the **Options...** button. Slide the bar to hide elements with low values
91. Select any of the elements to display its stress value in the messages window
92. Push the **Close** button
93. Push the **Up** button

Quit Design Studio

94. From the main menu bar, select **File** → **Quit**

14.3 Simply Supported Plate with Shell Elements

Introduction

The purpose of this exercise is to demonstrate the creation of a finite element mesh with shell elements in Design Studio.

Problem Statement:

A simply supported plate is modeled using quadrilateral shell (PSHELL) elements. This model is subject to point load at the center of the plate. The results requested are the displacements of the grids and the element stresses.

Note: The same technique used here to generate the 2-D mesh can be used for creating 1-D as well as 3-D meshes.

Example ID

AMDSG003

Files Used in This problem

A list of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
AMDSG003.dat	Input data	Created: Input file exported at the end of the exercise
AMDSG003_ref.dat	Input data	Provided: Reference file ready to be analyzed
AMDSG003_dsg00.pch	DSG file	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio

Create a Group

2. From the **Analysis** category chooser, select **Group Properties**

   Alternatively, one can also create a group by going into the **Manage Groups** button in the **Display** tab and pushing the **New Group** button from the Edit Menu toolbar. While creating a group using this feature, Design Studio would assign default values for the group properties which can be modified by going into the **Group Properties** in the **Analysis** tab

3. Push the **New Group Property** button from the Edit Menu toolbar
4. Enter `Plate` in the **Name** field
5. From the **Type** category chooser, select PSHELL

   You can choose PSOLID, etc. to create 3-D meshes, or PROD, etc. to create 1-D meshes
6. Push **Next**>
7. Check for `1 MAT1 (fix me)` for the material.

   If no material is created prior to creating a group, Design Studio automatically creates an isotropic material with some default properties with the above name. The user can modify these properties by choosing **Materials** in the **Analysis** tab
8. Enter `1.0` for the thickness
9. Push the **Finish** button

Modify the Material Properties

10. Select the **Analysis** tab
11. From the category chooser, select **Materials**
12. Select `1 MAT1 (fix me)` from the list
13. Push the **Modify Material** button from the Edit Menu toolbar
14. Enter `Plate_Material` in the name field
15. Check the **Isotropic (MAT1)** radio button
16. Push **Next**>
17. For the Young's modulus **E**, enter `207000`
18. For the Poisson coefficient **Nu**, enter `0.3`
19. For the density **Rho**, enter `7.8E-9`
20. Push the **Finish** button

Create a finite element mesh

Now you will create a finite element mesh with 18*18 quadrilateral elements. This step in Design Studio would automatically create all the necessary grid points and elements

21. From the category chooser, select **Elements**
22. Push the **New Elements** button from the Edit Menu toolbar
23. Check the **Define New Original Elements** radio button
24. Push **Next**>

25. Select the PSHELL group created earlier from the list
26. Push **Next>**
27. Select the **Point by Point** radio button
28. Select the **Quads** icon as **Region Definition options**
29. Enter `0.0` for **X**, `2.0` for **Y** and `0.0` for **Z**
30. Push the `Enter` key on your keyboard

    Repeat steps 29 and 30 to define the other corners of the Quads domain in order of the following table

Corner	X	Y	Z
2	18.0	2.0	0.0
3	18.0	20.0	0.0
4	0.0	20.0	0.0

31. Select the **Top** view icon in the Viewport to view the element created
32. In the list, select the `Quad` you just created
33. Push the **Subdivide** button
34. Enter `18` as Elements in 1st Dimension
35. Enter `18` as Elements in 2nd Dimension

    The first dimension is displayed by a red line on the shape domain in the Viewport window. The second dimension is displayed by a green line.

36. Push **Next>**
37. Push the **Finish** button

Create a Grid-Component-Set (Boundary Conditions)

38. From the **Analysis** category chooser, select **Grid-Component-Sets**
39. Push the **New Grid-Component Set** button from the Edit Menu toolbar
40. Enter `Simply_Supported` for the **Name**
41. Check the **Single-Point Constraints (SPC)** radio button
42. Push **Next>**
43. Select the four corner grids of the plate in the Viewport window

    You should have 4 grids selected

44. For **Components**, enter `123`

45. Push the **Set Components** button

    You should have 12 total dof on 4 grids

46. Push the **Finish** button

Create a Static Load

47. From the **Analysis** category chooser, select **Static Loads**
48. Push the **New Load Set** button from the Edit Menu toolbar
49. Enter `Center_Point_Load` for the **Name**
50. Check the **Force, Moment, Pressure, SPCD** radio button
51. Select the **Top** view icon in the Viewport window to change the view
52. Push **Next>**
53. Push the **Select None** button
54. Select the center grid point of the plate

    Alternatively, you can also enter 181 in the **Select by Grid ID** field and push the **Add** button

55. Enter `0.0` as **X** direction, `0.0` as **Y** direction and `1.0` as **Z** direction.

    This defines the direction of the perturbation (the positive Z direction)

56. Enter `100` for the **Magnitude**
57. Push the **Add Force** button
58. Push the **Finish** button

Create a Loadcase

59. From the **Analysis** category chooser, select **Loadcases**
60. Push the **New Loadcase** button from the Edit Menu toolbar
61. Enter `Loading` for the name
62. Check the **Static** radio button
63. Push **Next>**
64. For the **SPC**, select the Grid-Component-Set `Simply_Supported` you have just created
65. Push **Next>**
66. For the **Load Set**, select the Static Load `Center_Point_Load` you have just created
67. Push **Next>**

68. Select **Post** and **All** for the Displacement options
69. Select **Post** and **All** for the Element Stress options
70. Push the **Finish** button

Export the Input File

71. From the main menu bar select **File** → **Export** → **Input Data...**
72. Enter `AMDSG003`
73. Push the **Save** button

Save the Design Studio file

74. From the main menu bar, select **File** → **Save As...**
75. Enter `AMDSG003` as the Filename and push **Save (**as a Design Studio File)

Analyze the structure using Genesis

76. From the main menu bar, select **Genesis** → **Single Analysis**
77. Push the **Close** button

Import the Post-Processing Files

78. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**

    Alternatively, one can select the **Import Post...** button in the Genesis Console Output window before closing it in the earlier step.

79. Select the `AMDSG003_dsg00.pch` file
80. Push the **Open** button

View Displacements Results

81. Select the **Post** tab
82. Select any ISO view icon in the viewport
83. Push the **Deform Mesh/Color Mesh** button
84. Select the `Cycle 0 Loadcase 1 Displacement` Result
85. Push the **Oscillate** button
86. Push the **Filled Contour** button in the **Color Mesh**

87. In the Color Mesh list, select the same Displacement Result
88. Right-click in the viewport, select **List Top Ten**
89. Right-click in the viewport, select **List Bottom Ten**

Visualize the Stresses Results

90. Select `Cycle 0 Loadcase 1 Shell Stress` Result
91. Push the **Filled Elements** button in the **Color Mesh**
92. Select an element to display its stress value in the messages window
93. Push the **Close** button
94. Push the **Up** button

Quit Design Studio

95. From the main menu bar, select **File** → **Quit**

14.4 Changing Element Groups

Introduction

This example is aimed at demonstrating the methodology for changing the elements from one group to another using Design Studio.

Problem Statement:

A ten-rod truss model is used in this example. The initial model has all the ten rod elements assigned with the same properties. In this example, design studio is used to divide the ten rod elements into three different groups. The loading and boundary conditions are already applied on the model.

Note: The same technique used here for 1-D elements can be used to change the groups of 2-D or 3-D elements.

Example ID

AMDSG004

Files Used in This problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
AMDSG004.dat	Input data	Provided: File containing the finite element mesh along with boundary and loading conditions
AMDSG004_dsg.dat	Input data	Created: Input file used for running Genesis
AMDSG004_ref.dat	Input data	Provided: Reference file ready to be analyzed
AMDSG004_dsg00.pch	DSG file	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: AMDSG004.dat

Create three new Groups

In this example, the ten rod elements are divided into three groups. One group for the four horizontal elements, one for the two vertical elements and one of the four cross members

3. In the **Display** tab, push the **Manage Groups** button
4. Push the **New Group** button from the Edit Menu toolbar
5. Enter `Vertical` for the **Name**
6. From the **Type** chooser, select **PROD**
7. Choose a color you want
8. Push the **Finish** button

   One can repeat steps 4-7 to create another new group. Alternatively, Design Studio has the option of copying and pasting the group.

9. Select the group (`Vertical`) that was just created.
10. From the Edit Menu toolbar, select the **Copy Group** button
11. From the Edit Menu toolbar, select the **Paste Group** button
12. From the Edit Menu toolbar, select the **Paste Group** button
13. Select one of the new **PROD** group (`Copy of Vertical`)
14. Push the **Modify Group** button from the Edit Menu toolbar
15. For Name enter `Horizontal`
16. Push the **Finish** button
17. Select the other new **PROD** group (`Copy of Vertical`)
18. Push the **Modify Group** button from the Edit Menu toolbar
19. For Name enter `Cross Members`
20. Push the **Finish** button

    Design Studio automatically assigns the existing material and a default cross-sectional area.

Modify properties of the new groups

21. In the **Analysis** category chooser, select **Group Properties**
22. Select the `Vertical` group and push the **Modify Group Property** button from the Edit Menu toolbar
23. Enter `10.0` for the **Area**
24. Push the **Finish** button
25. Select the `Horizontal` group and push the **Modify Group Property** button from the Edit Menu toolbar
26. Enter `7.0` for the **Area**

27. Push the **Finish** button
28. Select the `Cross Members` group and push the **Modify Group Property** button from the Edit Menu toolbar
29. Enter `5.0` for the **Area**
30. Push the **Finish** button

Change Element Groups

31. In the **Analysis** category chooser, select **Elements**
32. Select the two vertical rod elements from the Viewport
33. Push the **Modify Elements** button from the Edit Menu toolbar
34. Check the **Change Elements' Group** option
35. Push the **Next>** button
36. Select the `Vertical` group to be the new group for the elements
37. Push the **Finish** button

    Notice the change in color of the modified elements in the Viewport to the color of the new group.

38. Select the four horizontal elements from the Viewport
39. Push the **Modify Elements** button from the Edit Menu toolbar
40. Check the **Change Elements' Group** option
41. Push the **Next>** button
42. Select the `Horizontal` group to be the new group for the elements
43. Push the **Finish** button
44. Select the four rod elements defining the cross members
45. Push the **Modify Elements** button from the Edit Menu toolbar
46. Check the **Change Elements' Group** option
47. Push the **Next>** button
48. Select the `Cross Members` group to be the new group for the elements
49. Push the **Finish** button

Delete empty group

50. In the **Display** tab, push the **Manage Groups** button

51. Select the empty group `Rods`
52. From the Edit Menu toolbar, select the **Delete Group** button

Analyze the structure using Genesis

53. From the main menu bar, select **Genesis** → **Single Analysis**
54. Push the **Close** button

Import the Post-Processing Files

55. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
56. Select the `AMDSG004_dsg00.pch` file
57. Push the **Open** button

Post-Process Displacements Results

58. Select the **Post** tab
59. Push the **Deform Mesh/Color Mesh** button
60. Select the `Cycle 0 Loadcase 1 Displacement` Result
61. Push the **Oscillate** button
62. Push the **Filled Contours** radio button in the **Color Mesh**
63. In the **Color Mesh** list, select the same Displacement Result

Quit Design Studio

64. From the main menu bar, select **File** → **Quit**

14.5 Composite Plate

Introduction

This example demonstrates the creation of a composite laminate using Design Studio.

Problem Statement:

A simply supported plate is modified to define the plate as a composite laminate. A new PCOMP group is created and the laminate is defined. Then the elements in the PSHELL are changed to the new PCOMP group. The loading and boundary conditions are defined in the provided file.

Note: In this example, the elements in the PSHELL are changed to a PCOMP group.

Example ID

`AMDSG005`

Files Used in This problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`AMDSG005.dat`	`Input data`	Provided: File containing the finite element mesh of a simply supported plate along with the loading and boundary conditions
`AMDSG005_dsg.dat`	`Input data`	Created: Input file created to run genesis
`AMDSG005_ref.dat`	`Input data`	Provided: Reference file ready to be analyzed
`AMDSG005_dsg00.pch`	`DSG file`	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `AMDSG005.dat`

Create a New Material

3. From the **Analysis** category chooser, select **Materials**
4. Push the **New Material** button from the Edit Menu toolbar

5. Enter `Unidirectional` for the **Name**
6. Check the **PSHELL/PCOMP Orthotropic (MAT8)** radio button
7. Push **Next>**
8. Enter the property values from the table below
9. Push the **Finish** button
10. Push the **New Material** button from the Edit Menu toolbar
11. Enter `Cloth` for the **Name**
12. Check the **PSHELL/PCOMP Orthotropic (MAT8)** radio button
13. Push **Next>**
14. Enter the property values from the table below.

Property	Unidirectional	Cloth
E_1	185000.0	81000.0
E_2	5000.0	75000.0
Nu_12	0.05	0.1
G_12	1000.0	5000.0
G_1z	1000.0	5000.0
G_2z	1000.0	5000.0
Rho	1.6E-9	1.6E-9
Xt	950.0	400.0
Xc	600.0	340.0
Yt	50.0	400.0
Yc	50.0	350.0
S	50.0	88.0

15. Push the **Finish** button

Create a new PCOMP Group

16. From the **Analysis** category chooser, select **Group Properties**
17. Push the **New Group Property** button from the Edit Menu toolbar

18. Enter `Laminate` for the **Name**
19. Select **PCOMP** in the **Type** category chooser
20. Choose the Color you want the group to be displayed in
21. Push **Next>**
22. Select `Hoffman` for the **Failure Theory**
23. Select `Unidirectional` for the Material
24. Enter `0.5` for the thickness and `0.0` for the angle for Layer 1
25. Push the + button to add another layer.
26. Repeat steps 23-25 to create the layer using the table below

Layer	Material	Thickness	Angle
2	Cloth	0.5	0.0
3	Unidirectional	0.5	45.0
4	Cloth	0.5	45.0
5	Unidirectional	0.5	90.0
6	Cloth	0.5	90.0

27. Push the **Finish** button

Change Element Groups

28. In the **Analysis** category chooser, select **Elements**
29. Push the **Select All** button to select all the elements in the Viewport
30. Push the **Modify Elements** button from the Edit Menu toolbar
31. Check the **Change Elements' Group** option
32. Push the **Next>** button
33. Select the `Laminate` PCOMP group to be the new group for the elements
34. Push the **Finish** button

Delete empty group

35. In the **Display** tab, push the **Manage Groups** button
36. Select the empty group `Plate`

37. From the Edit Menu toolbar, select the **Delete Group** button

Request for Element Forces/Failure Index

38. In the **Analysis** category chooser, select **Loadcases**
39. Select the existing loadcase `Loading`
40. Push the **Modify Loadcase** button from the Edit Menu toolbar
41. Push **Next>**
42. Push **Next>**
43. Push **Next>**
44. Select **Post** and **All** for the Element Stress options
45. Select **Post** and **All** for the Element Force options
46. Push the **Finish** button

Save the Design Studio file

47. From the main menu bar, select **File** → **Save As...**
48. Enter `AMDSG005` as the Filename and push **Save (**as a Design Studio File)

Analyze the structure using Genesis

49. From the main menu bar, select **Genesis** → **Single Analysis**
50. Push the **Close** button

Import the Post-Processing Files

51. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
52. Select the `AMDSG005_dsg00.pch` file
53. Push the **Open** button

Viewing Failure Index Results

54. Select the **Post** tab
55. Push the **Deform Mesh/Color Mesh** button
56. Select the `Cycle 0 Loadcase 1 Composite Failure Index` result in the **Color Mesh**

57. Right-click in the viewport, select **List Top Ten**
58. In the Viewport, click on an element to display its Failure Index value in the Design Studio Messages window

Quit Design Studio

59. From the main menu bar, select **File** → **Quit**

14.6 Hollow Tube with Shell Elements

Introduction

This example is used to demonstrate how a single element can be copied to create the finite element mesh of a hollow tube using Design Studio.

Problem Statement:

A hollow tube is created with 2-D elements. A normal mode analysis is carried out to estimate its natural frequencies and mode shapes.

Note: In this example, the quadrilateral elements are created by translating a single element. The same approach can be used to create 1-D or 3-D meshes as well.

Example ID

AMDSG006

Files Used in This problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
AMDSG006_ref.dat	Input data	Provided: Reference file ready to be analyzed
AMDSG006_dsg00.pch	DSG file	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio

Create a Group

2. From the **Analysis** category chooser, select **Group Properties**

   Alternatively, one can also create a group by going into the **Manage Groups** button in the **Display** tab and pushing the **New Group** button in the Edit Menu toolbar. While creating a group using this feature, Design Studio would assign default values for the group properties which can be modified by going into the **Group Properties** in the **Analysis** tab

3. Push the **New Group Property** button from the Edit Menu toolbar

4. Enter `Tube` in the **Name** field
5. From the **Type** category chooser, select PSHELL

   You can choose PSOLID, etc. to create 3-D meshes, or PROD, etc. to create 1-D meshes
6. Push **Next**>
7. Check for `1 MAT1 (fix me)` for the material.

   If no material is created prior to creating a group, Design Studio automatically creates an isotropic material with some default properties with the above name. The user can modify these properties by choosing **Materials** in the **Analysis** tab
8. Enter `1.0` for the thickness
9. Push the **Finish** button

Modify the Material Properties

10. Select the **Analysis** tab
11. From the category chooser, select **Materials**
12. Select `1 MAT1 (fix me)` from the list
13. Push the **Modify Material** button from the Edit Menu toolbar
14. Enter `Tube_Material` in the name field
15. Check the **Isotropic (MAT1)** radio button
16. Push **Next**>
17. For the Young modulus **E**, enter `210000`
18. For the Poisson coefficient **Nu**, enter `0.3`
19. For the density **Rho**, enter `7.8E-9`
20. Push the **Finish** button

Create a single quadrilateral element

Now you will create a single quadrilateral finite element that will be later used to create the mesh of a hollow tube.First, four grids are created and a quadrilateral element is defined to connect these four grids.

21. From the **Analysis** category chooser, select **Grids**
22. Push the **New Grids** button from the Edit Menu toolbar
23. Enter `2` in the **Number of New Grids** field
24. Check the **Along an arc** radio button
25. Push **Next**>

26. Enter `0.0` for **Center X1**, `0.0` for **Center X2** and `0.0` for **Center X3**

    These values define the center of the arc along which the grids are defined.

27. Enter `10.0` for **Radius**, `0.0` for **Initial Angle** and `15.0` for **Angle Increment**

    Radius defines the radius of the arc along which the grids are created
    Initial Angle is the starting angle where the first grid is created
    Angle Increment is the increment where consecutive grids are created.

28. Push the **Finish** button
29. Repeat steps 22-29 with the data given below to create two more grids

    Center X1 = `0.0`, **Center X2** = `0.0`, **Center X3** = `5.0`
    Radius = `10.0`, **Initial Angle** = `0.0`, **Angle Increment** = `15.0`

30. From the **Analysis** category chooser, select **Elements**
31. Push the **New Elements** button from the Edit Menu toolbar
32. Select the **Define New Original Elements** option
33. Push **Next**>
34. Select the **PSHELL** `Tube` created earlier
35. Push **Next**>
36. Select the **Point by Point** radio button
37. Select the **Quads** icon as **Region Definition options**
38. Select the **Pick Existing Grids** radio button
39. Select the **Iso-Front-Left-Top** button on the Viewport
40. Select the grids created from the Viewport

    The four grids need to be selected in either the clockwise or counter clockwise direction to define the element

41. Push the **Finish** button

Copy elements to create mesh

The single element created in the previous section is first rotated to create a circular layer of shell elements. Then this layer is duplicated in the axial direction to create the mesh of a hollow tube.

42. From the **Analysis** category chooser, select **Elements**
43. Push the **New Elements** button from the Edit Menu toolbar
44. Select the **Duplicate Selected Elements (use new rotated grids)** option
45. Push **Next**>

46. Enter `23` in the **Number of copies** field
47. Push **Next**>
48. Push **Select All** button
49. Push **Next**>
50. Enter `0.0` for **Center X1**, `0.0` for **Center X2** and `0.0` for **Center X3**

    These values define the center along which the element is rotated.
51. Check/Select **Z** as the **Rotational Axis**
52. Enter `15.0` for **Angle Increment**

    Angle Increment is the increment where the elements are created.
53. Push **Next**>
54. Select the **PSHELL** `Tube` created earlier
55. Push the **Finish** button
56. Select the **Iso-Front-Left-Top** button on the Viewport to view all the elements
57. Push the **New Elements** button from the Edit Menu toolbar
58. Select the **Duplicate Selected Elements (use new translated grids)** option
59. Push **Next**>
60. Enter `39` in the **Number of copies** field
61. Push **Next**>
62. Push **Select All** button
63. Push **Next**>
64. Enter `0.0` for **Offset X1**, `0.0` for **Offset X2** and `5.0` for **Offset X3**

    These values define the offset for each element in the x, y and z directions.
65. Push **Next**>
66. Select the **PSHELL** `Tube` created earlier
67. Push the **Finish** button
68. Select the **Iso-Front-Left-Top** button on the Viewport to view all the elements

    Although the model looks like a complete finite element mesh, the elements are disconnected because every time elements are duplicated new grids are created. So two adjacent elements do not share the same grids but have different grids. Design studio has a feature to **Merge Coincident** grids based on a tolerance.
69. From the **Analysis** category chooser, select **Grids**
70. Push the **Select All** button

71. Push the **Merge Coincident** button

    List Merge Candidates button prints out a list of the merging grids in the Design Studio messages window

72. Push the **Finish** button

    One can now notice the free grids in the Viewport represented by “*”

73. From the Edit Menu toolbar, select the **Delete Grids** button

    Even though all the grids are selected, Design studio only deletes the grids that are not connected to any elements.

Define an Eigenvalue Method

74. From the **Analysis** category chooser, select **Eigenvalue Methods**
75. Push the **New Eigenvalue Method** button from the Edit Menu toolbar
76. Enter `Method` in the **Name** field
77. For Method, select/check the **SMS** radio button
78. Enter `10000.0` in the **V2** field
79. Enter `50` in the **Number of Modes** field
80. Push the **Finish** button

Create a Loadcase for Normal Mode Analysis

81. From the **Analysis** category chooser, select **Loadcases**
82. Push the **New Loadcase** button from the Edit Menu toolbar
83. Enter `Normal Modes` in the **Name** field
84. Select the **Normal Modes** radio button
85. Push **Next>**
86. Push **Next>**

    In this example, no boundary conditions are defined. If boundary conditions are to be defined, one needs to define them in the **Grid-Component Sets** and selected from the **SPC/MPC** list boxes

87. Select the `Method` in the **Eigenvalue Method** listbox
88. Push **Next>**
89. Select **Post** and **All** for the **Displacement** options
90. Push the **Finish** button

Save the Design Studio file

91. From the main menu bar, select **File** → **Save As...**
92. Enter `AMDSG006` as the Filename and push **Save (**as a Design Studio File)

Analyze the structure using Genesis

93. From the main menu bar, select **Genesis** → **Single Analysis**
94. Push the **Close** button

Import the Post-Processing Files

95. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
96. Select the `AMDSG006_dsg00.pch` file
97. Push the **Open** button

View Mode Shapes and Natural Frequency Results

98. Select the **Post** tab
99. Push the **Deform Mesh/Color Mesh** button
100. Select the `Cycle 0 Loadcase 1 Mode 1 Eigenvector` Result

    Notice the frequency of the first six modes is close to zero indicating the rigid body modes of the structure.

101. Push the **Oscillate** button
102. Push the **Filled Contours** radio button in the **Color Mesh**
103. In the Color Mesh list, select the different eigenvector Result

    One can save the animation of the mode shape by using the **Save Animation** button. The animation is saved as a GIF file.

Quit Design Studio

104. From the main menu bar, select **File** → **Quit**

14.7 Generating a Shell Skin on a Solid Part

Introduction

The main purpose of this exercise is for you to learn how to generate a shell skin using Design Studio.

Problem Statement:

A solid part is imported into design studio and analyzed for its natural vibrations. The structure is free to move and it has 6 rigid body modes. Then Design Studio is used to generate a skin and the structure is reanalyzed and the natural frequencies are compared.

Example ID

AMDSG007

Files Used in This Problem

A list of the files provided to you and the files that you are expected to create during this exercise is presented next. It is not necessary to study the list in detail at this point.

File Name	Type	Description
AMDSG007.dat	Input data	Provided: Input file imported in design studio to start the exercise. Contains the solid mesh with an eigenvalue loadcase
AMDSG007_dsg00.pch	DSG file	Created: Punch file containing the analysis results
AMDSG007_dsg.dat	Input data	Created: Input file created to run genesis
AMDSG007_ref.dat	Input data	Provided: Reference file ready to be analyzed. Same as AMDSG007.dat

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: AMDSG007.dat

Analyze the Structure Using Genesis

3. From the main menu bar, select **Genesis** → **Single Analysis**

Import the Post-Processing Files

4. In the **Genesis Console Output** window, push the **Import Post...** button
5. Select the `AMDSG007_dsg00.pch` file
6. Push the **Import** button
7. In the **Genesis Console Output** window, push the **Close** button

Post-Processing the Results (Mode Shape)

8. Select the **Post** tab
9. Push the **Deform Mesh/Color Mesh** button
10. Push the **Filled Contours** radio button, this button is in the Color Mesh Window
11. From the **Color Mesh** list, select Mode 1 result for design cycle 0
12. To animate, push the **Oscillate** radio button
13. Verify that the first 6 modes are rigid body modes
14. Inspect modes 7, 8 and some others
15. Push the **Up** button

Study the Output File

16. Start any text editor
17. In the text editor load the Genesis data file: `AMDSG007_dsg.out`
18. Note the natural frequency for the 7th and 8th modes
19. Close the output file

Create a New Group

20. Select the **Display** tab
21. Push the **Manage Groups** button
22. Push the **New Group** button from the Edit Menu toolbar
23. Enter Name: `Reinforcement_Skin`
24. Pick Type: PSHELL
25. Pick Red Color
26. Push the **Finish** button
27. Push the **Up** button

Assign Property Values to the New Group

28. From the **Analysis** category chooser, select **Group Properties**
29. Select the **PSHELL** you just created, `Reinforcement_Skin`
30. Push the **Modify Group Property** button from the Edit Menu toolbar
31. For thickness: Enter `4.0`
32. Push the **Finish** button

Create the Skin

33. From the **Analysis** category chooser, select **Elements**
34. Push the **New Elements** button from the Edit Menu toolbar
35. Select the **Surface elements from Selected Solid Elements** option
36. Push **Next>**
37. Push the **Select ALL** button

    Verify that there are 5442 elements selected.
38. Push **Next>**
39. Select the PSHELL group
40. Push the **Finish** button, to finish the creation of the skin

Visualize the Skin

41. Select the **Display** Tab
42. Push the **Show/Hide Groups** button
43. Hide the PSOLID groups

    Notice that the shell elements forming the skin are shown
44. Push the **Up** button
45. For the **Model Cutaway**, select the **Cut V.C.S, X axis, hide - side** option
46. Use the slider below to change the cutting plane

    Notice that only a layer of shells is created for the surface solid elements.
47. For the **Model Cutaway**, select the **None** option to view the entire model
48. Push the **Show/Hide Groups** button and select the Show All button to display all groups

Save the Design Studio file

49. From the main menu bar, select **File** → **Save As...**
50. Enter `AMDSG007` as the Filename and push **Save (**as a Design Studio File)

Analyze the structure using Genesis

51. From the main menu bar, select **Genesis** → **Single Analysis**
52. Push the **Close** button

Import the Post-Processing Files

53. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
54. Select the `AMDSG007_dsg00.pch` file
55. Push the **Open** button

View Mode Shapes and Natural Frequency Results

56. Select the **Post** tab
57. Push the **Deform Mesh/Color Mesh** button
58. Push the **Oscillate** button
59. Push the **Filled Contours** radio button in the **Color Mesh**
60. In the Color Mesh list, Select the first `Cycle 0 Mode 7 Eigenvector` Result
61. Note the natural frequency from the Viewport
62. Go down the list and Select the second `Cycle 0 Mode 7 Eigenvector` Result
63. Note the frequency from the Viewport

    Notice the increase in the frequency because of the presence of the reinforced skin.

Quit Design Studio

64. From the main menu bar, select **File** → **Quit**

14.8 Automatic Creation of Ribs - Bar Elements

Introduction

This example demonstrates the automatic creation of ribs to enhance the stiffness using Design Studio.

Problem Statement:

A simply supported plate is used to create ribs that stiffen the plate. A new normal modes analysis is performed without the ribs and the natural frequencies are noted. Ribs are created using CBAR elements and analyzed again to compare the natural frequencies.

Example ID

`AMDSG008`

Files Used in This problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
AMDSG008.dat	Input data	Provided: File containing the finite element mesh of a simply supported beam
AMDSG008_dsg.dat	Input data	Created: Input file created to run genesis
AMDSG008_ref.dat	Input data	Provided: Reference file ready to be analyzed
AMDSG008_dsg00.pch	DSG file	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `AMDSG008.dat`

Delete Existing Static Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase

5. From the Edit Menu toolbar, select the **Delete Loadcase** button (or Delete on your keyboard)

Delete Existing Static Load

6. From the **Analysis** category chooser, select **Static Loads**
7. Select the existing static load
8. From the Edit Menu toolbar, select the **Delete Load Set** button (or Delete on your keyboard)

Define an Eigenvalue Method

9. From the **Analysis** category chooser, select **Eigenvalue Methods**
10. Push the **New Eigenvalue Method** button from the Edit Menu toolbar
11. Enter `Method` in the **Name** field
12. For **Method**, select the **SMS** radio button
13. Enter `30000.0` in the **V2** field
14. Enter `4` in the **Number of Modes** field
15. Push the **Finish** button

Create a Loadcase for Normal Mode Analysis

16. From the **Analysis** category chooser, select **Loadcases**
17. Push the **New Loadcase** button from the Edit Menu toolbar
18. Enter `Frequencies` in the **Name** field
19. Select the **Normal Modes** radio button
20. Push **Next>**
21. Select the boundary conditions `Simply_Supported` for **SPC**
22. Push **Next>**
23. Select the `Method` in the **Eigenvalue Method** listbox
24. Push **Next>**
25. Select **Post** and **All** for the **Displacement** options
26. Push the **Finish** button

Save the Design Studio file

27. From the main menu bar, select **File** → **Save As...**
28. Enter `AMDSG008` as the Filename and push **Save (**as a Design Studio File)

Analyze the Structure Using Genesis

29. From the main menu bar, select **Genesis** → **Single Analysis**
30. When done, push the **Close** button.

Import the Post-Processing Files

31. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
32. Select the `AMDSG008_dsg00.pch` file
33. Push the **Open** button

View Mode Shapes and Natural Frequency Results

34. Select the **Post** tab
35. Push the **Deform Mesh/Color Mesh** button
36. Push the **Oscillate** button
37. Push the **Filled Contours** radio button in the **Color Mesh**
38. In the Color Mesh list, Select the `Cycle 0 Mode 1 Eigenvector` Result
39. Note the natural frequency from the Viewport
40. Select the `Cycle 0 Mode 2 Eigenvector` Result
41. Note the frequency from the Viewport
42. Push the **Up** button

Create Ribs using CBAR elements

43. From the Analysis category chooser, select **Elements**
44. Push the **New Elements** button from the Edit Menu toolbar
45. Select the **Generate Autorib Elements** radio button
46. Push **Next>**
47. Select the PSHELL group `Plate` from the list to create the ribs
48. Push **Next>**
49. Select `Plate_Material` from the **Rib Material** listbox

50. Enter `5.0` for **Rib Thickness** and `5.0` for **Rib Height**
51. Check for **1 CBAR** for the **Elements Through Rib Height**
52. Push **Finish**

Analyze the Structure Using Genesis

53. From the main menu bar, select **Genesis** → **Single Analysis**
54. When done, push the **Close** button.

Import the Post-Processing Files

55. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
56. Select the `AMDSG008_dsg00.pch` file
57. Push the **Open** button

View Mode Shapes and Natural Frequency Results

58. Select the **Post** tab
59. Push the **Deform Mesh/Color Mesh** button
60. In the Color Mesh list, Select the first `Cycle 0 Mode 1 Eigenvector` Result
61. Note the natural frequency from the Viewport
62. Go through the list and select the second `Cycle 0 Mode 1 Eigenvector` Result
63. Note the change in the frequencies due to the addition of the rib elements

Quit Design Studio

64. From the main menu bar, select **File** → **Quit**

14.9 Automatic Creation of Ribs-QUAD Elements

Introduction

This example demonstrates the automatic creation of ribs to enhance the stiffness using Design Studio.

Problem Statement:

A simply supported plate is used to create ribs that stiffen the plate. A new normal mode analysis is performed without the ribs and the natural frequencies are noted. Ribs are created using CQUAD elements and analyzed again to compare the natural frequencies.

Example ID

`AMDSG009`

Files Used in This problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
AMDSG009.dat	Input data	Provided: File containing the finite element mesh of a simply supported beam
AMDSG009_dsg.dat	Input data	Created: Input file created to run genesis
AMDSG009_ref.dat	Input data	Provided: Reference file ready to be analyzed
AMDSG009_dsg00.pch	DSG file	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `AMDSG009.dat`

Delete Existing Static Loadcase

3. From the **Analysis** category chooser, select **Loadcases**
4. Select the existing loadcase

5. From the Edit Menu toolbar, select the **Delete Group** button (or Delete on your keyboard)

Delete Existing Static Load

6. From the **Analysis** category chooser, select **Static Loads**
7. Select the existing static load
8. From the Edit Menu toolbar, select the **Delete Group** button (or Delete on your keyboard)

Define an Eigenvalue Method

9. From the **Analysis** category chooser, select **Eigenvalue Methods**
10. Push the **New Eigenvalue Method** button from the Edit Menu toolbar
11. Enter `Method` in the **Name** field
12. For **Method**, select the **SMS** radio button
13. Enter `30000.0` in the **V2** field
14. Enter `4` in the **Number of Modes** field
15. Push the **Finish** button

Create a Loadcase for Normal Mode Analysis

16. From the **Analysis** category chooser, select **Loadcases**
17. Push the **New Loadcase** button from the Edit Menu toolbar
18. Enter `Frequencies` in the **Name** field
19. Select the **Normal Modes** radio button
20. Push **Next>**
21. Select the boundary conditions `Simply_Supported` for **SPC**
22. Push **Next>**
23. Select the `Method` in the **Eigenvalue Method** listbox
24. Push **Next>**
25. Select **Post** and **All** for the **Displacement** options
26. Push the **Finish** button

Save the Design Studio file

27. From the main menu bar, select **File** → **Save As...**
28. Enter `AMDSG009` as the Filename and push **Save (**as a Design Studio File)

Analyze the Structure Using Genesis

29. From the main menu bar, select **Genesis** → **Single Analysis**
30. When done, push the **Close** button.

Import the Post-Processing Files

31. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
32. Select the `AMDSG009_dsg00.pch` file
33. Push the **Open** button

View Mode Shapes and Natural Frequency Results

34. Select the **Post** tab
35. Push the **Deform Mesh/Color Mesh** button
36. Push the **Oscillate** button
37. Push the **Filled Contours** radio button in the **Color Mesh**
38. In the Color Mesh list, Select the `Cycle 0 Mode 1 Eigenvector` Result
39. Note the natural frequency from the Viewport
40. Select the `Cycle 0 Mode 2 Eigenvector` Result
41. Note the frequency from the Viewport

Create Ribs using CQUAD4 elements

42. From the Analysis category chooser, select **Elements**
43. Push the **New Elements** button from the Edit Menu toolbar
44. Select the **Generate Autorib Elements** radio button
45. Push **Next>**
46. Select the PSHELL group (`Plate`) from the list to create the ribs
47. Push **Next>**
48. Select `Plate_Material` from the **Rib Material** listbox
49. Enter `2.0` for **Rib Thickness** and `1.0` for **Rib Height**

50. Select for **1 CQUAD** for the **Elements Through Rib Height**

    2 CQUADs option would create two QUAD elements through the height of the rib
    3 CQUADs option would create three QUAD elements through the height of the rib

51. Push **Finish**

    Notice the newly created CQUAD elements in the Viewport.

Analyze the Structure Using Genesis

52. From the main menu bar, select **Genesis** → **Single Analysis**
53. When done, push the **Close** button.

Import the Post-Processing Files

54. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
55. Select the `AMDSG009_dsg00.pch` file
56. Push the **Open** button

View Mode Shapes and Natural Frequency Results

57. Select the **Post** tab
58. Push the **Deform Mesh/Color Mesh** button
59. In the Color Mesh list, Select the first `Cycle 0 Mode 1 Eigenvector` Result
60. Note the natural frequency from the Viewport
61. Go through the list and select the second `Cycle 0 Mode 1 Eigenvector` Result
62. Note the change in the frequency due to the addition of the rib elements

Quit Design Studio

63. From the main menu bar, select **File** → **Quit**
64. Push the **Don’t Save** button

14.10 Mesh Refinement

Introduction

This example demonstrates the step involved to refine a finite element mesh using Design Studio.

Problem Statement:

A simply supported plate is used as the starting mesh. Each element in this mesh is subdivided into 4 new elements. The loading and boundary conditions available in the provided file.

Note: The same technique used here to refine the 2-D mesh can be used for refining 1-D as well as 3-D meshes.

Example ID

`AMDSG010`

Files Used in This problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
AMDSG010.dat	Input data	Provided: File containing the FE mesh along with the loading conditions
AMDSG010_ref.dat	Input data	Provided: Reference file ready to be analyzed
AMDSG010_dsg00.pch	DSG file	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `AMDSG010.dat`

Split Elements into Smaller Elements

3. From the **Analysis** category chooser, select **Elements**

4. Push the Select All button

   Notice that there are 324 elements selected.
   Alternatively, one can also select the elements to be refined from the viewport.

5. Push the **Modify Elements** button from the Edit Menu toolbar

6. Select the **Split into Smaller Elements** radio button

7. Push the **Finish** button

   Push the Select All button and notice that there are 4*324 = 1296 elements selected.

Save the Design Studio file

8. From the main menu bar, select **File** → **Save As...**

9. Enter `AMDSG010` as the Filename and push **Save (**as a Design Studio File)

Analyze the Structure Using Genesis

10. From the main menu bar, select **Genesis** → **Single Analysis**

11. When done, push the **Close** button.

Import the Post-Processing Files

12. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**

13. Select the `AMDSG010_dsg00.pch` file and push the **Open** button

View Displacements Results

14. Select the **Post** tab

15. Push the **Deform Mesh/Color Mesh** button

16. Select the `Cycle 0 Loadcase 1 Displacement` Result

17. Push the **Filled Contours** radio button in the **Color Mesh**

18. In the Color Mesh list, select the same Displacement Result

19. Right-click in the viewport, select **List Top Ten**

Quit Design Studio

20. From the main menu bar, select **File** → **Quit**

14.11 Three Story Facade with Panel

Introduction

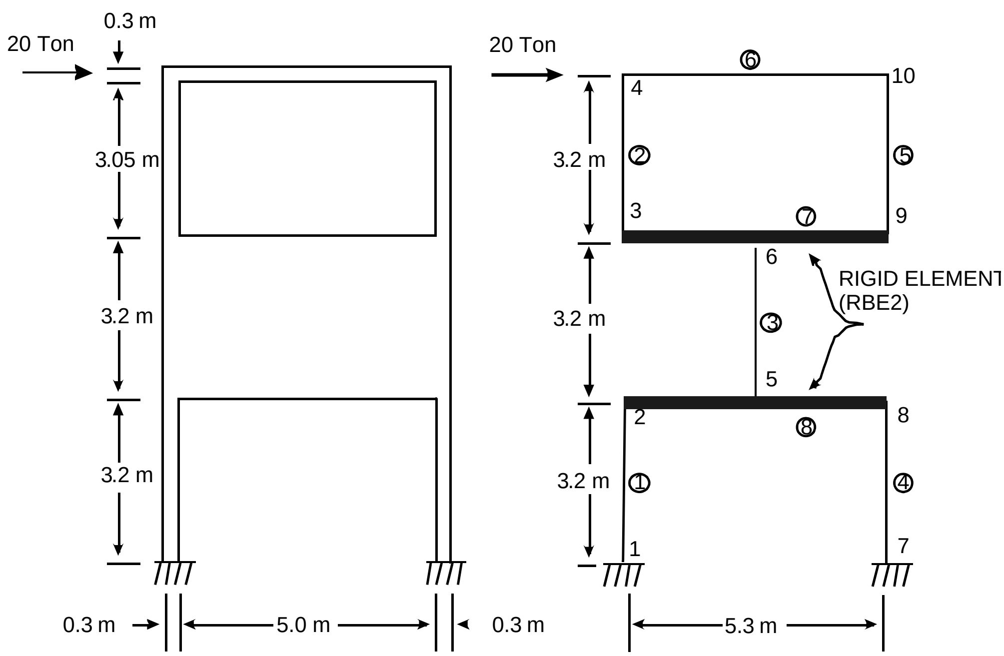
The purpose of this exercise is to demonstrate the creation of Rigid elements using Design Studio.

Genesis example ID:

A008

Problem Statement:

Find the nodal displacements of the facade as shown in the figure below.



Problem Description:

1. Analysis of a two dimensional structure with a simplified model of 6 bar elements (CBAR) and 2 rigid body elements (RBE2).
2. Section properties of flexible beams: Area=0.06 m^2, I2=0.0045 m^4, AS2=0.05 m^2
3. Section properties of modeled beam: Area=1.6 m^2, I2=3.2614 m^4, AS2=1.248 m^2
4. Material: E=3.0E6 T/m^2, G=1.2E6 T/m^2.

5. One loadcase. Loading:20.0 T.

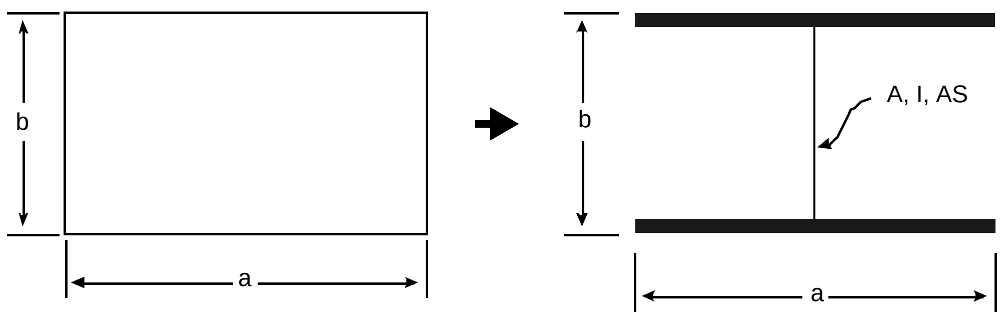
The analysis of the facade is performed using a stick model. Panel zones, as in this problem, can be modeled using two rigid elements and one beam element with modified properties. In the figure below, a detail of the panel model is given. These rigid elements which are parallel to the horizontal force, cause the planes they are located in to remain plane after deformation. Shear deformation is included in all beam elements.
The properties of the equivalent column are:

$$A = \frac{at}{1-\nu}$$

$$I = \frac{a^3 t}{12(1-\nu^2)}\left[1 + \frac{1-\nu}{2}\left(\frac{b}{a}\right)^2\right]$$

$$AS = \frac{A}{(1-\nu)\left[1 + \frac{1-\nu}{2}\left(\frac{b}{a}\right)^2\right]}$$

where t is the thickness of the panel.



Example ID

AMDSG011

Files Used in This problem

A list, of the key files provided or created are presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
AMDSG011.dat	Input data	Provided: Input file imported at the beginning of the exercise. Contains everything except the MPC definitions
AMDSG011_ref.dat	Input data	Provided: Reference input file ready to be analyzed
AMDSG011_dsg00.pch	DSG file	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `AMDSG011.dat`

Create a new group for Rigid elements

3. From the **Display** tab, push the **Manage Groups**
4. Push the **New Group** button from the Edit Menu toolbar
5. Enter `Rigid` in the **Name** field
6. Select **Rigid/Interpolation** from the **Type** listbox
7. Select the blue color
8. Push **Finish**

Create rigid elements

9. From the **Analysis** category chooser, select **Elements**
10. Push the **New Elements** button from the Edit Menu toolbar
11. Select the **Define New Original Elements** radio button
12. Push **Next>**
13. Select the RBE group `Rigid` created earlier
14. Push **Next>**
15. Enter `6` in the Select by Grid ID field and push the Enter on the keyboard

    This first grid defines the master grid where all the independent degrees of freedom are assigned.
16. Enter `9` in the Select by Grid ID field and push the Enter on the keyboard
17. Enter `3` in the Select by Grid ID field and push the Enter on the keyboard

    The grids where the dependent degrees of freedom are assigned are defined from the second grid.
18. Enter `135` for the Dependent Components
19. Push **Finish**
20. Push the **New Elements** button from the Edit Menu toolbar
21. Select the **Define New Original Elements** radio button
22. Push **Next>**

23. Select the RBE group (`Rigid`) created earlier
24. Push **Next>**
25. Enter `5` in the Select by Grid ID field and push the Enter on the keyboard
26. Enter `8` in the Select by Grid ID field and push the Enter on the keyboard
27. Enter `2` in the Select by Grid ID field and push the Enter on the keyboard
28. Enter `135` for the Dependent Components
29. Push **Finish**

Save the Design Studio file

30. From the main menu bar, select **File** → **Save As...**
31. Enter `AMDSG011` as the Filename and push **Save (**as a Design Studio File)

Analyze the Structure Using Genesis

32. From the main menu bar, select **Genesis** → **Single Analysis**
33. When done, push the **Close** button.

Import the Post-Processing Files

34. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
35. Select the `AMDSG011_dsg00.pch` file and push the **Open** button

View Displacements Results

36. Select the **Post** tab
37. Push the **Deform Mesh/Color Mesh** button
38. Select the `Cycle 0 Loadcase 1 Displacement` Result
39. Push the **Filled Contours** radio button in the **Color Mesh**
40. In the Color Mesh list, select the same Displacement Result

Quit Design Studio

41. From the main menu bar, select **File** → **Quit**

CHAPTER 15

Analysis - Solution Control Examples

- Buckling Analysis of a Tripod
- Using Composite Failure Equations
- Two Bar Symmetric Frame
- Dynamic Analysis of Cantilever Beam

15.1 Buckling Analysis of a Tripod

Introduction

The main purpose of this exercise is to learn how to create a buckling loadcase in Design Studio.This exercise also goes through the creation of concentrated mass elements.

Problem Statement:

A finite element mesh of a tripod is imported into design studio and analyzed for buckling. A concentrated mass is modeled at the tip of the tripod. Gravitational force is applied as a static load for the buckling analysis.

Example ID

`ASDSG001`

Files Used in This Problem

A list of the files provided to you and the files that you are expected to create during this exercise is presented next. It is not necessary to study the list in detail at this point.

File Name	Type	Description
`ASDSG001.dat`	`Input data`	Provided: Input file imported in design studio to start the exercise. Contains the finite mesh of a tripod
`ASDSG001_ref.dat`	`Input data`	Provided: Reference file ready to be analyzed.

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `ASDSG001.dat`

Review the finite element mesh

3. From the **Analysis** category chooser, select **Group Properties**
4. Select one of the PBARL properties
5. Push the **Modify Group Property** button from the Edit Menu toolbar
6. Review the cross-section definition

7. Push the **Cancel** button

   The groups are modeled as tubes with varying outside diameters and constant thickness.

Create new group for concentrated mass

8. From the **Analysis** category chooser, select **Group Properties**
9. Push the **New Group Property** button from the Edit Menu toolbar
10. Enter `Concentrated Mass` for the **Name**
11. Select PCONM3 for the **Type**
12. Push **Next>** button
13. Enter `0.01` for the **Mass**
14. Push the **Finish** button

Create a mass element

15. From the **Analysis** category chooser, select **Elements**
16. Push the **New Elements** button from the Edit Menu toolbar
17. Check the **Define New Original Elements** radio button
18. Push **Next**>
19. Select the PCONM3 group created earlier from the list
20. Push **Next**>
21. Select the **Points** icon in **Region Definition options**
22. Select the **Pick existing grids** radio button
23. Select the grid at the tip of the tripod from the Viewport
24. Push the **Finish** button

    Notice a solid circle on the Viewport at the grid point representing the presence of a concentrated mass.

Create boundary conditions

25. From the **Analysis** category chooser, select **Grid-Component Sets**
26. Push the **New Grid-Component Set** button from the Edit Menu toolbar
27. Enter `Boundary Conditions` for the **Name**
28. Check the **Single-Point Constraints** radio button

29. Push **Next>**
30. Enter `1` in the **Select by Grid ID** field and push the **Add** button
31. For **Components**, enter `1236`
32. Push the **Set Components** button
33. Push the **Select None** button
34. Select the remaining two grids on the bottom of the tripod from the Viewport
35. For **Components**, enter `3`
36. Push the **Set Components** button

    You should have 6 total dof on 3 grids

37. Push the **Finish** button

Create a static load with gravitational forces

38. From the **Analysis** category chooser, select **Static Loads**
39. Push the **New Load Set** button from the Edit Menu toolbar
40. Enter `Gravity` for the **Name**
41. Select the **Gravity Load** radio button
42. Push **Next>**
43. Enter `-980.0` for the **Acceleration** and check that `X=0`, `Y=0` and `Z=1`

    X, **Y** and **Z** define the direction of the gravitational force.

44. Push the **Finish** button

Define an Eigenvalue Method

45. From the **Analysis** category chooser, select **Eigenvalue Methods**
46. Push the **New Eigenvalue Method** button from the Edit Menu toolbar
47. Enter `Method` in the **Name** field
48. For Method, select the **Lanczos** radio button
49. Enter `1E-8` in the **V1** field

    Make sure **V2** is left blank

50. Enter `4` in the **Number of Modes** field
51. Push the **Finish** button

Create a Static Loadcase

52. From the **Analysis** category chooser, select **Loadcases**
53. Select the default static loadcase created by Design Studio
54. Push the **Modify Loadcase** button from the Edit Menu toolbar
55. Enter `Static Loading` for the Name
56. Push **Next>**
57. From the **SPC** category chooser, select the Grid-Component-Set `Boundary Conditions` was created earlier
58. Push **Next>**
59. From the **Load Set** category chooser, select the Static Load `Gravity`
60. Push the **Finish** button

Create a Buckling Loadcase

61. From the **Analysis** category chooser, select **Loadcases**
62. Push the **New Loadcase** button from the Edit Menu toolbar
63. Enter `Buckling` in the name field
64. Select **Buckling** as the loadcase type
65. Push **Next>**
66. Select the Eigenvalue Method `Method` created earlier
67. Push **Next>**
68. Select the Static Loadcase `Static Loading` created
69. Push the **Finish** button

Save the Design Studio file

70. From the main menu bar, select **File** → **Save As...**
71. Enter `ASDSG001` as the Filename and push **Save (**as a Design Studio File)

Analyze the Structure Using Genesis

72. From the main menu bar, select **Genesis** → **Single Analysis**
73. When done, push the **View Output File** button.

74. Study the Buckling Load Factors in the output file

Quit Design Studio

75. From the main menu bar, select **File** → **Quit**

15.2 Using Composite Failure Equations

Introduction

This example demonstrates how to create composite failure equations to define the failure index of a composite laminate using Design Studio.

Problem Statement:

A simply supported plate modeled using laminated composite materials is modified so that the failure index is calculated based on an equation that defines the VonMises stresses. A composite failure equation is defined to represent the VonMises stress and used as the failure criteria on the PCOMP data entry.

Example ID

`ASDSG002`

Files Used in This problem

A list, of the key files provided are presented here. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`ASDSG002.dat`	`Input data`	Provided: File containing the FE mesh of a simply supported plate
`ASDSG002_dsg.dat`	`Input data`	Created: Input file created to run genesis
`ASDSG002_ref.dat`	`Input data`	Provided: Reference file ready to be analyzed
`ASDSG002_dsg00.pch`	`DSG file`	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `ASDSG002.dat`

Create a New Composite Failure Equation

3. From the **Analysis** category chooser, select **Composite Failure Equations**
4. Push the **New Findex** button from the Edit Menu toolbar
5. Enter `VonMises` for the **Name**

6. Check the **Stress-based (FINDEX)** radio button

   One can also define a composite failure equation based on strains by selecting the **Strain-based (FINDEXN)** radio button. In this case, the quantities used in the equation correspond to the strains/strain-limits.

7. Enter `F=SQRT(S1**2+S2**2-S1*S2+3*S12**2)+0.0*(XT+XC+YT+YC+S+F12)` in the **Equation** field

   While defining a composite failure equation, all the quantities specified need to be used in the equation definition. That is the reason for multiplying the unnecessary quantities in the equation with a `0.0`

8. Push **Finish**

Defining the failure equation on the PCOMP

9. From the **Analysis** category chooser, select **Group Properties**
10. Select the PCOMP property from the list
11. Push the **Modify Group Property** button from the Edit Menu toolbar
12. Select the created failure equation (VonMises) from the drop-down listbox for the **Failure Theory**
13. Push **Finish**

Save the Design Studio file

14. From the main menu bar, select **File** → **Save As...**
15. Enter `ASDSG002` as the Filename and push **Save (**as a Design Studio File)

Analyze the Structure Using Genesis

16. From the main menu bar, select **Genesis** → **Single Analysis**
17. When done, push the **Close** button.

Import the Post-Processing Files

18. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
19. Select the `ASDSG002_dsg00.pch` file
20. Push the **Open** button

Viewing Failure Index Results

21. Select the **Post** tab
22. Push the **Deform Mesh/Color Mesh** button
23. Select the `Cycle 0 Loadcase 1 Composite Failure Index` Result in the **Color Mesh**
24. In the Viewport, click on an element to display its Failure Index value in the Design Studio Messages window

Quit Design Studio

25. From the main menu bar, select **File** → **Quit**

15.3 Two Bar Symmetric Frame

Introduction

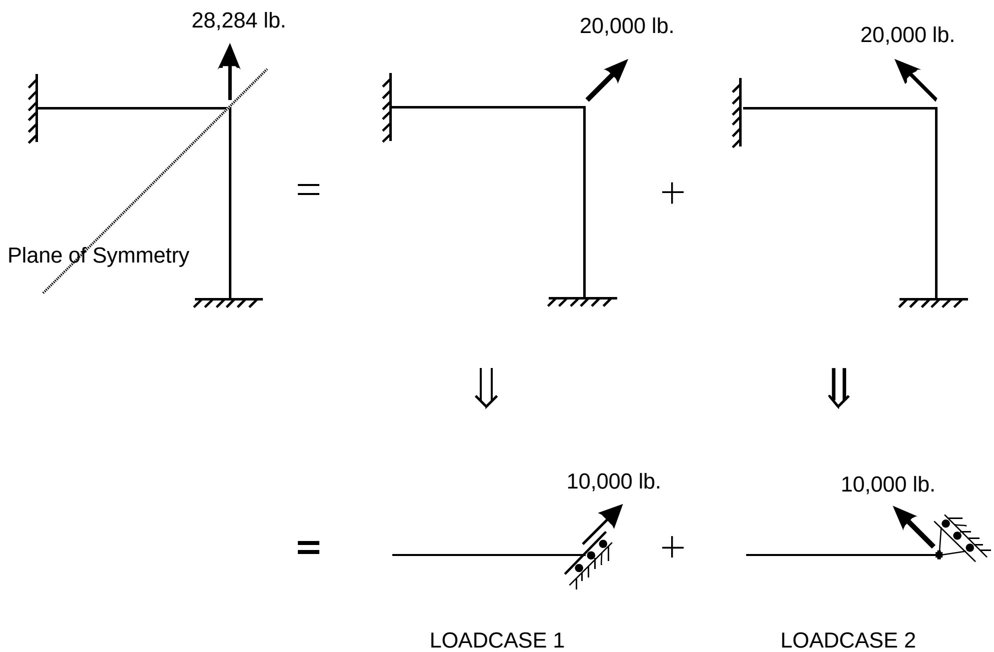
The purpose of this exercise is to demonstrate the use of multi-point constraints (MPC) to break a symmetric structure into a half model under two load conditions.

Genesis example ID:

A007

Problem Statement:

Find the nodal displacements of a symmetric two bar frame as shown in the figure below.



Problem Description:

1. One beam structure in the X-Z plane (half model, CBAR element)
2. Section properties: Area=50 in^2, I2=416.7 in^4
3. Material E=1.0E7 psi.
4. Two load cases: each of them has different boundary conditions

B.C. 1: Ux-Uz=0 (`MPC Set 1`), Uy=0, $\theta_x = \theta_y = \theta_z = 0$.

B.C. 2: Ux+Uz=0 (`MPC Set 2`), Uy=0, $\theta_x = \theta_z = 0$.

5. One load combination that represent the solution of the left part of the structure.

   Utilization of Symmetry: First the load is divided in the sum of its symmetric and antisymmetric components. Then, each of the forces is applied to the structure as a separate load case. Third, the structure in each load case is broken in half by considering suitable boundary conditions and load. Finally, the solution of the left part of the structure is recovered by adding the solutions of the symmetric and antisymmetric load case.

Example ID

`ASDSG003`

Files Used in This problem

A list, of the key files provided or created are presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
`ASDSG003.dat`	`Input data`	Provided: Input file imported at the beginning of the exercise. Contains everything except the MPC definitions
`ASDSG003_ref.dat`	`Input data`	Provided: Reference input file ready to be analyzed
`ASDSG003_dsg00.pch`	`DSG file`	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `ASDSG003.dat`

Create Multi-Point Constraints (MPC)

In this section, the two MPC's are created. `MPC Set 1` corresponds to the boundary conditions for loadcase 1 and `MPC Set 2` corresponds to Loadcase 2

3. From the **Analysis** category chooser, select **Grid-Component Sets**
4. Push the **New Grid-Component Set** button from the Edit Menu toolbar
5. Enter `MPC Set 1` in the **Name** field
6. Select **Multi-Point Constraints (MPC)** radio button
7. Push **Next>**
8. Push the + button to define a new MPC entry
9. Enter `2` for the **Grid ID**, `1` for the **Component** and `1.0` for the **Coefficient**

10. Push the **+** button

    The + button allows for the addition of other degrees of freedom to the MPC equation.

11. Enter `2` for the **Grid ID**, `3` for the **Component** and `-1.0` for the **Coefficient**
12. Push **Next>**
13. Push **Finish**
14. Push the **New Grid-Component Set** button from the Edit Menu toolbar
15. Enter `MPC Set 2` in the **Name** field
16. Select **Multi-Point Constraints (MPC)** radio button
17. Push **Next>**
18. Push the + button to define a new MPC entry
19. Enter `2` for the **Grid ID**, `1` for the **Component** and `1.0` for the **Coefficient**
20. Push the **+** button
21. Enter `2` for the **Grid ID**, `3` for the **Component** and `1.0` for the **Coefficient**
22. Push **Next>**
23. Push **Finish**

Assigning MPC to the loadcase

24. From the **Analysis** category chooser, select **Loadcases**
25. Select the first Loadcase `Loadcase 1`
26. Push the **Modify Loadcase** button from the Edit Menu toolbar
27. Push **Next>**
28. Select `MPC Set 1` from the MPC listbox
29. Push **Finish**
30. Select the second Loadcase `Loadcase 2`
31. Push the **Modify Loadcase** button from the Edit Menu toolbar
32. Push **Next>**
33. Select `MPC Set 2` from the MPC listbox
34. Push **Finish**

    Currently Design Studio cannot create load combination. It can read and write a static combination loadcase. For you convenience, the static combination loadcase is provided in the input file.

Save the Design Studio file

35. From the main menu bar, select **File** → **Save As...**
36. Enter `ASDSG003` as the Filename and push **Save (**as a Design Studio File)

Analyze the Structure Using Genesis

37. From the main menu bar, select **Genesis** → **Single Analysis**
38. When done, push the **Close** button.

Import the Post-Processing Files

39. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
40. Select the `ASDSG003_dsg00.pch` file and push the **Open** button

View Displacements Results

41. Select the **Post** tab
42. Push the **Deform Mesh/Color Mesh** button
43. Select the `Cycle 0 Loadcase 3 Displacement` Result
44. Push the **Filled Contour** button in the **Color Mesh**
45. In the Color Mesh list, select the same Displacement Result

Quit Design Studio

46. From the main menu bar, select **File** → **Quit**

15.4 Dynamic Analysis of Cantilever Beam

Introduction

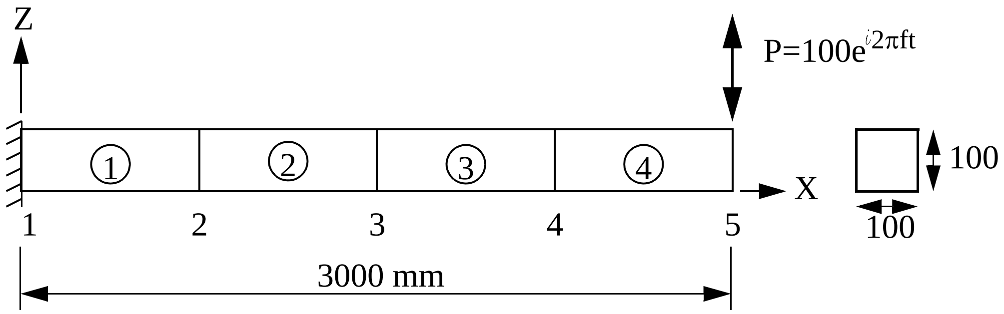
The purpose of this exercise is to demonstrate how to perfrom frequency response analysis and also creating a frequency response plot using Design Studio

Genesis example ID:

A025

Problem Statement:

Find the nodal displacements of the cantilever beam shown in the figure below. The beam is loaded with a tip point load of 100.0 N and it is applied at three different frequencies. The first frequency is close to zero to simulate a static loading (f=1.0e-8 hz), the second frequency is f=8.8 hz and the third is f=57.1 hz. The last two frequencies are very close to the first and second natural modes to simulate loadings close to resonance. Use direct dynamic analysis and modal analysis. For modal analysis the first three modes are considered. Damping is not considered.



Problem Description:

1. Four BAR elements.
2. The section properties are: area=10000 mm^2. I=8.3E6 mm^4
3. Material: E=2.07E+5 MPa. density=8.0-9 N-sec^2/mm^4
4. One frequency load case, necessary for modal dynamic analysis.
5. One direct dynamic response load case. P=100 N
6. One modal dynamic response load case. P=100 N
7. Excitation frequencies for the problem are: f1=1.0e-8 hz, f2=8.8 hz and f3=57.1 hz

Example ID

`ASDSG004`

Files Used in This problem

A list, of the key files provided or created are presented next. These files will be introduced during the exercise, so it is not necessary to study the list in detail at this point.

File Name	Type	Description
ASDSG004.dat	Input data	Provided: Input file imported at the beginning of the exercise.
ASDSG004_ref.dat	Input data	Provided: Input file created at the end of the exercise.
ASDSG004_dsg00.pch	DSG file	Created: Punch file containing the analysis results

Start Design Studio

1. Start Design Studio
2. Import the Genesis data file: `ASDSG004.dat`

Create an Eigenvalue Method

3. From the **Analysis** category chooser, select **Eigenvalue Methods**
4. Push the **New Eigenvalue Method** button from the Edit menu toolbar
5. Enter `SMS-100Hz` for the **Name**
6. Make sure the **SMS** radio button is selected for the **Method**
7. Enter `100.0` for **V2**
8. Push the **Finish** button

Create the Normal Modes Loadcase

9. From the **Analysis** category chooser, select **Loadcases**
10. Select `Loadcase 1`
11. Push the **Delete Loadcase** button
12. Push the **New Loadcase** button
13. Enter `Modes` for the **Name**
14. Select the **Normal Modes** radio button

15. Push the **Next>** button
16. Push **Next>**
17. For the Eigenvalue method, select `SMS-100Hz`
18. Push **Next>**
19. For the Displacement, select **Post**

    Verify that All appears in the 2nd category chooser.

20. Push the **Finish** button

Create the Frequency Response Data

21. From the **Analysis** category chooser, select **Frequency Response Data**
22. Push the **New Freq.Resp.Data** button from the Edit menu toolbar
23. Enter `Table` for the **Name**
24. Select the **Freq. Function Table (TABLEDi)** radio button
25. Push the **Next>** button
26. For the **Form**, select **TABLED1** option
27. Keep the first X-Y pair of X=1.0, Y=1.0 and push the + button to add one more X-Y pairs of X=2.0, Y=1.0
28. Push the **Finish** button

    Verify that the TABLED data is listed.

29. Push the **New Freq.Resp.Data** button
30. Enter `Delay` for the **Name**
31. Push the **Next>** button
32. For the **Grid ID**, enter `5`
33. For the **Component**, select **3**
34. For the **Value**, enter `60.0`
35. Push the **Finish** button

    Verify that the DELAY data is listed.

36. Push the **New Freq.Resp.Data** button
37. Enter `Darea` for the **Name**
38. Push the **Next>** button
39. For the **Grid ID**, enter `5`

40. For the **Component**, select **3**
41. For the **Value**, enter `100.0`
42. Push the **Finish** button

    Verify that the DAREA data is listed.

43. Push the **New Freq.Resp.Data** button
44. Enter `DynamicLoading` for the **Name**
45. Select the **Dynamic Load Set (RLOADi)** radio button
46. Push the **Next>** button
47. For the **DAREA**, select `Darea`
48. For the **DELAY**, select `Delay`
49. For the **TC Table**, select `Table`
50. Push the **Finish** button

    Verify that the DLOAD data is listed.

51. Push the **New Freq.Resp.Data** button
52. Enter `LoadingFrequencies` for the **Name**
53. Select the **Loading Frequency Set (FREQi)** radio button
54. Push the **Next>** button
55. From the category chooser, select **FREQ**
56. Enter `1E-8` for **F**
57. Push the **+** button
58. Enter `8.8` for **F**
59. Push the **+** button
60. Enter `57.1` for **F**
61. Push the **Finish** button

    Verify that the FREQ data is listed.

Create the Direct Frequency Response Loadcase

62. From the **Analysis** category chooser, select **Loadcases**
63. Push the **New Loadcase** button
64. Enter `Direct_Freq_Resp` for the **Name**
65. Select the **Direct Frequency Response** radio button

66. Push the **Next>** button
67. Push the **Next>** button
68. For the **Dynamic Load Set**, select `DynamicLoading`
69. For the **Loading Frequency Set**, select `LoadingFrequencies`
70. Push the **Next>** button
71. For Displacement, select **Post** and **All** in the two pull down menu's respectively
72. Push the **Finish** button

Create the Frequency Response Loadcase

73. From the **Analysis** category chooser, select **Loadcases**
74. Push the **New Loadcase** button
75. Enter `Modal_Freq_Resp` for the **Name**
76. Select the **Modal Frequency Response** radio button
77. Push the **Next>** button
78. Push the **Next>** button

    Do not choose an eigenvalue method. We will use the normal modes from the first loadcase for the frequency response.

79. Push the **Next>** button
80. For the **Dynamic Load Set**, select `DynamicLoading`
81. For the **Loading Frequency Set**, select `LoadingFrequencies`
82. For the **Modes Loadcase**, select `Modes`
83. Push the **Next>** button
84. For Displacement, select **Post** and **All** in the two pull down menu's respectively
85. Push the **Finish** button

Save the Design Studio file

86. From the main menu bar, select **File** → **Save As...**
87. Enter `ASDSG004` as the Filename and push **Save (**as a Design Studio File)

Analyze the Structure Using Genesis

88. From the main menu bar, select **Genesis** → **Single Analysis**

89. When done, push the **Close** button.

Import the Post-Processing Files

90. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...**
91. Select the `ASDSG004_dsg00.pch` file and push the **Open** button

Create a Frequency Response Plot

92. Select the **Post** tab
93. Push the **Freq. Resp. Plot** button
94. Push the **New Freq. Resp. Plot** button from the Edit Menu toolbar
95. Push the **Magnitude + Phase** radio button
96. Push **Next>**
97. Push the **+** button
98. Choose the `Cycle 0 Loadcase 3 Displacement` dataset
99. Push **Next>**
100. Select Grid 5 for the grid
101. Select the Translation 3 radio button to plot the z-component
102. Push **Next>**
103. Push the **+** button

    The + button can be used to plot multiple responses on the same plot
104. Choose the `Cycle 0 Loadcase 4 Displacement` dataset
105. Push **Next>**
106. Select Grid 4 for the grid
107. Select the Translation 3 radio button to plot the z-component
108. Push **Next>**
109. Push the **Finish** button

Save the Frequency Responses Plot

110. Right click on the graph
111. Push the **Save Image...** button
112. Enter `ASDSG004` as the file name

113. Push the **Save** button
114. Push the **Close** button

Quit Design Studio

115. From the main menu bar, select **File** → **Quit**
116. Push the **Don't Save** button