



Design Studio for GENESIS

HOW TO MANUAL

STEP BY STEP INSTRUCTIONS

VERSION 11.0

October 2009

© VANDERPLAATS RESEARCH & DEVELOPMENT, INC.
1767 SOUTH 8TH STREET
COLORADO SPRINGS, CO 80905
Phone: (719) 473-4611 Fax: (719) 473-4638
<http://www.vrand.com>
email: genesis.support@vrand.com

COPYRIGHT NOTICE

© Copyright, 1991-2009 by Vanderplaats Research & Development, Inc. All Rights Reserved, Worldwide. No part of this manual may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any human or computer language, in any form or by any means, electronic, mechanical, magnetic, optical, chemical, manual, or otherwise, without the express written permission of Vanderplaats Research & Development, Inc., 1767 South 8th Street, Suite 100, Colorado Springs, CO 80905.

WARNING

This software and manual are both protected by U.S. copyright law (Title 17 United States Code). Unauthorized reproduction and/or sales may result in imprisonment of up to one year and fines of up to \$10,000 (17 USC 506). Copyright infringers may also be subject to civil liability.

DISCLAIMER

Vanderplaats Research & Development, Inc. makes no representations or warranties with respect to the contents hereof and specifically disclaims any implied warranties of merchantability or fitness for any particular purpose. Further, Vanderplaats Research & Development, Inc. reserves the right to revise this publication and to make changes from time to time in the content hereof without obligation of Vanderplaats Research & Development, Inc. to notify any person or organization of such revision or change.

TRADEMARKS MENTIONED IN THIS MANUAL

GENESIS, DOT and DOC are trademarks of Vanderplaats Research & Development, Inc. Other products mentioned in this manual are trademarks of their respective developers or manufacturers.



CHAPTER 1

Introduction

- Design Studio
- Conventions
- About this manual



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 the window along the right side of the screen. The Main Window is the primary site 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.

Additionally, the top left corner of the Messages Window 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 chapters are intended to familiarize the user with: using Design Studio; the methods to create optimization data using Design Studio for Genesis; and the methods to visualize results with Design Studio for Genesis. The methods to create analysis data (grid locations, elements, and loads) are not covered in these exercises. Analysis data may be created in any other preprocessor.

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 About this manual

With this manual, you will learn, step by step, how to use the different options and capabilities of Design Studio.

The first part of this manual teaches you how to use the functions of each tab and also the general functions of Design Studio.

The second part of this document shows how to add or modify Genesis analysis parameters (PARAM) and/or Genesis design optimization parameters (DOPT) using Design Studio.

To use this document, you do not need to read all of it. You can search for the functions you are interested in by searching keyword(s). For example, if you are looking for how to delete an element, search for the keywords “element” and “delete”.

Useful tip on how to navigate this document:

A set of steps (“How to...”) can refer to another set of steps. In this case you can directly click on the other set of steps (typically in bold) to access it. To come back to your original set of steps you can use the acrobat reader shortcut for “previous view”: **Alt+ left arrow**. You can also use the **Alt+ right arrow** to navigate forward.



CHAPTER 2

How to use the main functions of Design Studio

- How to use the general functions of Design Studio
- How to use the functions of the Display tab
- How to use the functions of the Analysis tab
- How to use the functions of the Topology tab
- How to use the functions of the Design tab
- How to use the functions of the Post tab
- How to use the 'Find' Filter



2.1 How to use the general functions of Design Studio

How to Start and Load an Existing Genesis Input Data

1. Start Design Studio
Design Studio will open with an empty database
2. 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 (*.dat).
3. Go to the directory where the input data is stored and select it
4. Push the **Open** button
Alternatively, one can drag a Input data file (*.dat) or a Design Studio database file (*.dsg) and drop it on the Design Studio icon to import or open the file.

How to Save the Design Studio Database File

1. From the main menu bar, select **File** → **Save As...**
2. Enter a name as filename
3. Push the **Save** button (as a Design Studio File)

How to Import Analysis Post-Processing Files

1. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...** (or Ctrl-R)
2. Select the filename_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.
You can also import *.DVG or *.SHP files.
Results can be filtered by type, loadcase and/or eigenvalue mode.
3. Push the **Open** button
4. Alternatively, if you have the **Genesis Console Output** window open (after an Analysis Run or Optimization Run), push the **Import Post...** button located at the bottom of the **Genesis Console Output** window.
5. Select the filename_dsg00.pch file from the New Post-Processing Files window
Multiple files can be selected for import by using the **Shift** or **Ctrl** keys



6. Push the **Import** button located at the bottom right of the New Post-Processing Files window

How to Import an Isodensity Surface Mesh

1. From the main menu bar, select **File** → **Import** → **Input Data...** (or Ctrl-I)
2. Select the `filenameSURF.dat` file
3. Push the **Open** button

How to Import Design Post-Processing Files

1. From the main menu bar, select **File** → **Import** → **Punch/Output2 Results...** (or Ctrl-R)
2. Select the `filename_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.
Results can be filtered by type, loadcase and/or eigenvalue mode.
3. Push the **Open** button
4. Alternatively, if you have the **Genesis Console Output** window open (after an Analysis Run or Optimization Run), push the **Import Post...** button located at the bottom of the **Genesis Console Output** window.
5. Select the `filename_dsgOPOST00.pch` file from the New Post-Processing Files window

Multiple files can be selected for import by using the **Shift** or **Ctrl** keys
6. Push the **Import** button located at the bottom right of the New Post-Processing Files window

How to Import the History File

1. From the main menu bar, select **File** → **Import** → **Design History Results...** (or Ctrl-Shift-R)
2. Select the `filename_dsg.HIS` file
3. Push the **Open** button

How to Import a Shell-to-Solid (SSOL) File

1. Start Design Studio

2. From the main menu bar, select **File** → **Import** → **Input Data...** (or Ctrl-I)
3. Select the `filename_dsgSSOLxx.dat` file (xx corresponds to the number of the last design cycle)
4. Push the **Open** button

How to Import an Update File

1. Start Design Studio
2. From the main menu bar, select **File** → **Import** → **Input Data...** (or Ctrl-I)
3. Select the Genesis data file: `filename_dsgUPDATExx.dat` (xx corresponds to the number of the design cycle)
4. Push the **Open** button

How to Export the Input File

1. From the main menu bar, select **File** → **Export** → **Input Data...** (or Ctrl-E)
2. Enter a name as Filename
3. Put a checkmark in the **Only Export Visible Groups/Elements** checkbox

Put a checkmark in the **Only Export Visible Groups/Elements** checkbox to export only visible groups/elements or uncheck the **Only Export Visible Groups/Elements** checkbox to export all groups/elements.
4. Push the **Save** button

How to Export a Surface Representation in IGES Data Format

1. From the main menu bar, select **File** → **Export** → **Coarsened Surface...** (or Ctrl+Shift-E)
2. For **Surface File Format**, select **IGES**
3. Move the **Surface Mesh** slider to the intended setting

The slider is used to alter the number of elements used to represent the surface. The number of elements used and the size of the generated file increases as one goes from **Coarse** to **Fine** settings
4. Enter a file name
5. Push the **Save** button

How to Export a Surface Representation in STL Format

1. From the main menu bar, select **File** → **Export** → **Coarsened Surface...** (or Ctrl+Shift-E)
2. For **Surface File Format**, select **STL**
3. Move the **Surface Mesh** slider to the intended setting

The slider is used to alter the number of elements used to represent the surface. The number of elements used and the size of the generated file increases as one goes from **Coarse** to **Fine** settings
4. Enter a file name
5. Push the **Save** button

How to Export a Coarse Surface Representation in Bulk Data Format

1. From the main menu bar, select **File** → **Export** → **Coarsened Surface...** (or Ctrl-Shift-E)
2. For **Surface File Format**, select **Bulk Data**
3. Move the **Surface Mesh** slider to the intended setting

The slider is used to alter the number of elements used to represent the surface. The number of elements used and the size of the generated file increases as one goes from **Coarse** to **Fine** settings
4. Enter a file name
5. Push the **Save** button

How to Create a Picture File

Select a good view of the results.

1. From the main menu bar, select **File** → **Print to Image File...**
2. Enter a file name
3. Push the **Save** button

Check your working directory. A file with the picture should be there. The extension of the file should be “png”.

How to Quit Design Studio

1. From the main menu bar, select **File** → **Quit** (or Alt-F4)
2. Push the **Save** button to save your work or push the **Don't Save** button.

How to Use One Color for the Whole Structure

1. From the main menu bar, select **Color** → **Group Color Style**→**All One Color**

How to Get the Problem Summary

1. From the main menu bar, select **Genesis** → **Model Summary**
2. Push the **Close** button

How to Request the UPDATE File

The UPDATE file is a Genesis input file that contains the updated data entries for a particular optimization cycle. The name of the file will be `filename_dsgUPDATExx.dat` (where `xx` is the design cycle number).

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **File Control** tab
3. For **Updated Input File**, select the option

All Cycles option will create updated files for all the design cycles.

First Cycle option will create the updated file for only the first design cycle.

Last Cycle option will create the updated file for only the last design cycle.

First & Last option will create two updated files, one corresponding to the first design cycle the other corresponding to the last.

Every Other Cycle option will create updated files for every other cycle and the final design cycle.

Every Fifth Cycle option will create updated file in every fifth design cycle and the final design cycle.

Never option will not create any updated files. This is the default option in Genesis.

4. Push the **Apply** button

How to Request the SSOL File (Solid 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. The name of the SSOL file will be: `filename_dsgSSOLxx.dat` (where `xx` corresponds to the last design cycle number).

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **File Control** tab
3. For **Shell-to-Solid File**, select the option

Create option will create the SSOL files.

Create (Fixed Norms) option will align 2-D Element's Orientations and then create the



SSOL file.

Do Not Create option will not create the SSOL file. This is the default option in Genesis.

4. Push the **Apply** button

How to Request the OPOST Post-Processing File

The OPOST file is a Genesis post-processing file that contains the sizing optimization results. The name of the post-processing file will be: `filename_dsgOPOSTxy.pch` (where `xy` is the design cycle number).

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **File Control** tab
3. For **Element Sizing File**, select one of the options

Create option will create the OPOST files based on the Design Output Control (DPRINT) data.

Do Not Create option will not create any OPOST files.

4. Push the **Apply** button

How to Request the Shape Change (SHP) File

The Shape Change file is a Genesis post-processing file that contains the shape changes during the each of the optimization cycles in the specified post-processing format. The name of the post-processing file will be: `filename_dsg.SHP`.

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **File Control** tab
3. For **Shape Change File**, choose an option

Create option will create the Shape Change file.

Do Not Create option will not create the Shape Change file.

4. Push the **Apply** button

How to Change the Title or the Subtitle of a Project

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Output Control** tab
3. Enter a name in the **Title** or **Subtitle** field
4. Push the **Apply** button

How to Increase the Calculating Memory in Genesis

1. From the main menu bar, select **Genesis** → **Options...**

2. Select the **Analysis Control** tab
3. Check **Memory Limit in MW (LENVEC)**
4. Enter a value (e.g 250) in the **Memory Limit in MW** field
e.g this will cause genesis to use 1024MB
5. Push the **Apply** button

How to Increase the Number of Threads in Genesis

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Select the **Processors (THREADS)** checkbox
4. Enter a value (e.g 2) in the **Processors** field
Care should be taken to input a value lesser than the number of available processors. If the number is greater than the number of processors available, performance will be degraded
5. Push the **Apply** button

How to Analyze the Structure Using Genesis

1. From the main menu bar, select **Genesis → Single Analysis**
2. Study the **Genesis Console Output**;
3. Push the **Close** button

How to Optimize the Structure Using Genesis

1. From the main menu bar, select **Genesis → Optimize**
2. Study the **Genesis Console Output**;
3. Push the **Import Post...** button located at the bottom of the **Genesis Console Output window**.
Similar results for ALL Design Cycles cannot be imported in a single step by this method.
4. Select the files to be imported from the New Post-Processing Files window
Multiple files can be selected for import by using the **Shift** or **Ctrl** keys
5. Push the **Import** button located at the bottom right of the New Post-Processing Files window
6. Push the **Close** button
7. Study the **Design History**; when done, push the **Close** button



How to Access Design Studio Online Help

1. From the main menu bar, select **Help** → **Design Studio Help**
2. Navigate
3. Exit by closing the Help window

How to Create a New View in the View Catalog

1. Select an appropriate view using your mouse and the view icons in the Viewport
2. Push the **View Catalog** icon in the Viewport window
3. Push the **New** button
4. Enter a name
5. Push the **Close** button

How to Clear All Selections

1. Right-click on the Viewport, select **Clear**→ **All**

To clear a Grid Selection, select **Clear**→ **Grid Selection**

To clear a Group Selection, select **Clear**→ **Group Selection**

To clear an Element Selection, select **Clear**→ **Element Selection**

To clear a Coordinate System Selection, select **Clear**→ **Coordinate System Selection**

To clear a Domain Selection, select **Clear**→ **Domain Selection**

To clear Labels, select **Clear**→ **Labels**

To clear Arrows, select **Clear**→ **Arrows**

To clear Special Hilites, select **Clear**→ **Special Hilities**

2.2 How to use the functions of the Display tab

How to create Model Cutaway View

1. Select the **Display** tab
2. If necessary, set the View Coordinate system using **How to Set a Coordinate System as a View Reference**
3. For the **Model Cutaway**, select an option
 - Cut V.C.S X axis, hide - side** will create cutaway perpendicular to the X-axis and hide the model on the negative side of the axis.
 - Cut V.C.S X axis, hide + side** will create cutaway perpendicular to the X-axis and hide the model on the positive side of the axis.
 - Cut V.C.S Y axis, hide - side** will create cutaway perpendicular to the Y-axis and hide the model on the negative side of the axis.
 - Cut V.C.S Y axis, hide + side** will create cutaway perpendicular to the Y-axis and hide the model on the positive side of the axis.
 - Cut V.C.S Z axis, hide - side** will create cutaway perpendicular to the Z-axis and hide the model on the negative side of the axis.
 - Cut V.C.S Z axis, hide + side** will create cutaway perpendicular to the Z-axis and hide the model on the positive side of the axis.
 - None** will display the entire model.
4. Use the Slider for moving the cutting plane along the axis selected

How to Display an Assembly

1. Select the **Display** tab
2. From the category chooser, select the assembly you want to display

How to Show/Hide Groups

1. Select the **Display** tab
2. Push the **Show/Hide Groups** button
3. Pick in the group list the group(s) to hide
 - 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.
4. Push the **Up** button

How to Show/Hide Elements

1. From the **Display** tab, push the **Show/Hide Elements** button

2. In the Viewport, select an appropriate view icon
3. Clear any selection using **How to Clear All Selections**
4. Select the elements you want to Display using **How to Select Element(s) by picking in the Viewport** steps
5. Push the **View Only Selected** button
You can also select the elements you want to Hide and push the **Hide Selected** button
You can display all the elements by pushing the **All On** button
6. Push the **Up** button

How to Identify Grids

1. From the **Display** tab, push the **Identify Grids** button
2. In the Viewport, select an appropriate view icon
3. Select the grid(s) you want to identify using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
4. In the Message window, you can read Grid ID, Grid Coordinate
You can also calculate the distance between 2 grids: select the 2 grids, push the **Calculate Distance** button, read the value in the Messages window.
You can also calculate an angle: select 3 grids, push the **Calculate Angle** button, read the value in the Messages window.
You can also calculate an arc center: select 3 grids, push the **Calculate Arc Center** button, read the coordinate of the arc center in the Messages window.
5. Push the **Up** button

How to Identify Elements

1. From the **Display** tab, push the **Identify Elements** button
2. In the Viewport, select an appropriate view icon
3. Clear any selection using **How to Clear All Selections**
4. Select the elements you want to identify using **How to Select Element(s) by picking in the Viewport** steps
5. In the Message window, you can read Element ID, Element Group (e.g PSOLID), Element Type (e.g Hexa), the Grid ID Corners of the Element
6. Push the **Clear** button to clear the selection
7. Push the **Up** button

How to Create a New Group

1. From the **Display** tab, push the **Manage Groups** button
2. Push the **New Group** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select a Type
5. Select a Color
6. Push the **Finish** button
7. Push the **Up** button

By default, Design Studio assigns some material properties. You can check, change or complete the material properties by selecting Group Properties in the Analysis category chooser and picking the corresponding material.

How to Rename a Group

1. From the **Display** tab, push the **Manage Groups** button
2. Select the Group you want to rename
3. Push the **Modify Group** button in the Edit Menu Tool bar
4. Enter a name in the name field
5. Push the **Finish** button
6. Push the **Up** button

How to Change the Color of a Group

1. From the **Display** tab, push the **Manage Groups** button
2. Select the Group you want the color to be changed
3. Push the **Modify Group** button in the Edit Menu Tool bar
4. Select a Color
5. Push the **Finish** button
6. Push the **Up** button

How to Create a Copy of a Group

1. From the **Display** tab, push the **Manage Groups** button
2. Select the Group you want to copy
3. Push the **Copy Group** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+C



4. Push the **Paste Group** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+V

5. Push the **Up** button

How to Delete a Group

1. From the **Display** category chooser, push the **Manage Groups** button
2. Select the Group(s) to be deleted
3. Push the **Delete Group** button in the Edit Menu Tool bar
4. Push the **Up** button

How to Create a Fixed Assembly

1. Select the **Display** tab
2. Push the **Manage Assemblies** button
3. Push the **New Assembly** button in the Edit Menu Toolbar
4. Enter a name in the name field
5. Select the **Fixed Assembly** radio button
6. Push **Next>**
7. Select Group(s) for Creating Assembly
8. Push the **Finish** button
9. Push the **Up** button

How to Create a Smart Assembly with a Set of Elements

1. Select the **Display** tab
2. Push the **Manage Assemblies** button
3. Push the **New Assembly** button in the Edit Menu Toolbar
4. Enter a name in the name field
5. Select the **Smart Assembly** radio button
6. Push **Next>**
7. Select conditions for Creating Assembly

Multiple conditions can be created by using the + button

8. Push the **Finish** button

9. Push the **Up** button

How to Create a New Color

1. From the **Display** tab, push the **Manage Color Palette** button
2. Push the **New Color** button in the Edit Menu Toolbar
3. Select the **Swatches** tab
4. Pick a color

You can also select the **HSB** tab, choose the color you want.
Or you can also select the **RGB** tab and define the level of each primitive color
5. Push the **Ok** button
6. Push the **Up** button

How to Modify a Color

1. From the **Display** tab, push the **Manage Color Palette** button
2. Select the color to be modified
3. Push the **Modify Color** button in the Edit Menu Tool bar
4. Select the **Swatches** tab
5. Pick a color

You can also select the **HSB** tab, choose the color you want.
Or you can also select the **RGB** tab and define the level of each primitive color
6. Push the **Ok** button
7. Push the **Up** button

How to Set a Coordinate System as a View Reference

1. Create a new coordinate system using **How to Create a Coordinate System**
2. From the **Display** tab, push the **Set View Coord. Sys.** button
3. Select the coordinate system you have created
4. Push the **Set** button

Now, all the view icons from the Viewport refers to the coordinate system you have created.
5. Push the **Up** button

How to Show/Hide Free Grids

1. Select the **Display** tab
2. Select the **Show** (or **Hide**) radio button, which is to the right of **Free Grids**

How to Study a Symmetric Design Space

1. Request the update file using **How to Request the UPDATE File**
2. Run Single Analysis using **How to Analyze the Structure Using Genesis**
3. Import an Update file using **How to Import an Update File**
4. Select the **Display** tab
5. 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 tooltip will reveal the names/function of the buttons.

6. Push the **Show/Hide Groups** button

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).

2.3 How to use the functions of the Analysis tab

How to Select Grid(s) by Grid ID

In many cases, you need to select grids. For example to define a Displacement constraint or to define a symmetric axis, etc... For all these cases you can select the grids by their Grid ID as the following steps show you.

1. Select the **Analysis** tab
2. From the **Analysis** category chooser, select **Grids**
3. Enter a Grid ID in the **Select by Grid ID** field
4. Push the **Add** button

You need to repeat these last two steps to select each Grid

How to Select Grid(s) by picking in the Viewport

In many cases, you need to select grids. For example to define a Displacement constraint or to define a symmetric axis, etc. For all these cases you can select the grids by picking them in the Viewport window as the following steps show you.

1. Select the **Analysis** tab
2. From the **Analysis** category chooser, select **Grids**
3. Select an appropriate View
4. Pick the Grid(s) in the Viewport

To deselect a grid, pick by holding the Ctrl key

How to Select Element(s) by Element ID

In many cases, you need to select elements. For example to define a Stress constraint or to define a group, etc... For all these cases you can select the elements by their Element ID as the following steps show you.

1. Select the **Analysis** tab
2. From the **Analysis** category chooser, select **Elements**
3. Enter an Element ID in the **Select by Element ID** field
4. Push the **Add** button

You need to repeat these last two steps to select each Element

How to Select Element(s) by picking in the Viewport

In many cases, you need to select elements. For example to define a Displacement constraint

or to define a symmetric axis, etc... For all these cases you can select the elements by picking them in the Viewport window as the following steps show you.

1. Select the **Analysis** tab
2. From the **Analysis** category chooser, select **Elements**
3. Select an appropriate View
4. Pick the Element(s) in the Viewport

To deselect an element, pick by holding the Ctrl key

How to Create Boundary Conditions (Single Point constraint SPC)

1. From the **Analysis** category chooser, select **Grid-Component Sets**
2. Push the **New Grid-Component Set** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Check the **Single-Point Constraints** radio button
5. Push **Next>**
6. Select the grid(s) to be constrained using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport**
7. For **Components**, enter direction number(s) and rotation number(s) you want 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
8. Push the **Set Components** button
9. Push the **Finish** button

How to Modify Existing Boundary Conditions (Single Point constraint SPC)

1. From the **Analysis** category chooser, select **Grid-Component Sets**
2. Select the Grid-Component Set you want to modify
3. Push the **Modify Grid-Component Set** button in the Edit Menu Toolbar
4. Change the name if necessary
5. Push **Next>**
6. To remove boundary conditions, select the grids using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport**

7. For **Components**, enter direction number(s) and rotation number(s) you want to remove the constraint
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
8. Push the **Clear Components** button
9. To add boundary conditions, select the grid(s) to be constrained using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport**
10. For **Components**, enter direction number(s) and rotation number(s) you want to restrain
11. Push the **Set Components** button
12. Push the **Finish** button

How to Create a Copy of Boundary Conditions (Grid-Component Sets)

1. From the **Analysis** category chooser, select **Grid-Component Sets**
2. Select the Grid-Component Set you want to copy
3. Push the **Copy Grid-Component Set** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+C
4. Push the **Paste Grid-Component Set** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+V

How to Delete a Boundary Conditions (Grid-Component Sets)

1. From the **Analysis** category chooser, select **Grid-Component Sets**
2. Select the Grid-Component Set(s) to be deleted
3. Push the **Delete Grid-Component Set** button in the Edit Menu Tool bar

How to Create an Element Set

1. From the **Analysis** category chooser, select **Element Sets**
2. Push the **New Element Set** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select Element(s) you want to be in your set using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps
5. Push the **Finish** button



How to Modify an Element Set

1. From the **Analysis** category chooser, select **Element Sets**
2. Select the Element Set to modify
3. Push the **Modify Element Set** button in the Edit Menu Toolbar
4. Change the name if necessary
5. Modify the selected elements

One can clear the selected elements by right-clicking on the Viewport and selecting **Clear→Element Selection**. New selection of element(s) can be done using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps

If elements need to be added to the existing selection, one can add them by selecting them on the Viewport or select them based on the Element ID

Alternatively, if elements are to be removed from the existing selection, one can remove them by using the **Ctrl** + click on the element in the Viewport or unselect them based on the Element ID

6. Push the **Finish** button

How to Create a Copy of an Element Set

1. From the **Analysis** category chooser, select **Element Sets**
2. Select the Element Set to copy
3. Push the **Copy Element Set** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+C

4. Push the **Paste Element Set** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+V

How to Delete an Element Set

1. From the **Analysis** category chooser, select **Element Sets**
2. Select the Element Set(s) to be deleted
3. Push the **Delete Element Set** button in the Edit Menu Tool bar

You delete a set, not the elements. If you want to delete elements, please refer to **How to Delete Elements**

How to Create a Grid Set

1. From the **Analysis** category chooser, select **Grid Sets**
2. Push the **New Grid Set** button in the Edit Menu Toolbar

3. Enter a name in the name field
4. Select Grid(s) you want to be in your set using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
5. Push the **Finish** button

How to Modify a Grid Set

1. From the **Analysis** category chooser, select **Grid Sets**
2. Select the Grid Set to modify
3. Push the **Modify Grid Set** button in the Edit Menu Toolbar
4. Change the name if necessary
5. Modify the selected grids

One can clear the selected grids by right-clicking on the Viewport and selecting **Clear→ Grid Selection**. New selection of grid(s) can be done using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps

If grids need to be added to the existing selection, one can add them by selecting them on the Viewport or select them based on the Grid ID

Alternatively, if grids are to be removed from the existing selection, one can remove them by using the **Ctrl** + click on the grid in the Viewport or unselect them based on the Grid ID

6. Push the **Finish** button

How to Create a Copy of a Grid Set

1. From the **Analysis** category chooser, select **Grid Sets**
2. Select the Grid Set to copy
3. Push the **Copy Grid Set** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+C

4. Push the **Paste Grid Set** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+V

How to Delete a Grid Set

1. From the **Analysis** category chooser, select **Grid Sets**
2. Select the Grid Set to be deleted
3. Push the **Delete Grid Set** button in the Edit Menu Tool bar

You delete a set, not the grids. If you want to delete grids, please refer to **How to Delete Grid(s)**

How to Create a Static Load (Force, Moment)

1. From the **Analysis** category chooser, select **Static Loads**
2. Push the **New Load Set** button in the Edit Menu Toolbar
3. Check the **Force, Moment, Pressure,SPCD** radio button
4. Push **Next>**
5. Push the **Select None** button
6. Select the Grid(s) where the Force (Moment) is applied using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
7. Define the direction of the Load
by entering numbers in the **X, Y** and **Z** direction fields
or
by pushing the **By 2 Grids...** button, then by selecting 2 grids in the Viewport and by pushing **Next>**
or
by pushing the **By 3 Grids...** button, then by selecting 3 grids in the Viewport and by pushing **Next>**
8. Enter a **Magnitude** value
9. Push the **Add Force** button
and/or Push the **Add Moment** button to add a moment
10. Push the **Finish** button

How to Modify a Static Load (Force, Moment)

1. From the **Analysis** category chooser, select **Static Loads**
2. Select the Static Load to modify
3. Push the **Modify Load Set** button in the Edit Menu Toolbar
4. Push **Next>**
5. Push the **Select None** button
6. To remove loads, select the grids using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport**
7. Push the **Clear Loads/SPCDs** button
8. To add loads, select the Grid(s) where the Force (Moment) is applied using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps

9. Define the direction of the Load
by entering numbers in the **X**, **Y** and **Z** direction fields
or
by pushing the **By 2 Grids...** button, then by selecting 2 grids in the Viewport and by pushing **Next>**
or
by pushing the **By 3 Grids...** button, then by selecting 3 grids in the Viewport and by pushing **Next>**
10. Enter a **Magnitude** value
11. Push the **Add Force** button
and/or Push the **Add Moment** button to add a moment
12. Push the **Finish** button

How to Create a Copy of a Static Load

1. From the **Analysis** category chooser, select **Static Loads**
2. Select the Static Load you want to copy
3. Push the **Copy Load Set** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+C
4. Push the **Paste Load Set** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+V

How to Delete a Static Load

1. From the **Analysis** category chooser, select **Static Loads**
2. Select the Static Load(s) to be deleted
3. Push the **Delete Load Set** button in the Edit Menu Tool bar

How to Define an Eigenvalue Method

1. From the **Analysis** category chooser, select **Eigenvalue Methods**
2. Push the **New Eigenvalue Method** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. For Method push either the SMS, Lanczos, or Subspace Iteration button
5. Complete the appropriate fields

More details about the fields can be obtained by the Volume I:Analysis Manual



6. Push the **Finish** button

How to Modify an Eigenvalue Method

1. From the **Analysis** category chooser, select **Eigenvalue Methods**
2. Select the Eigenvalue Method to modify
3. Push the **Modify Eigenvalue Method** button in the Edit Menu Toolbar
4. Change name if necessary
5. To change the Method select either the SMS, Lanczos, or Subspace Iteration radio button
6. Change/Complete the appropriate fields

More details about the fields can be obtained by the Volume I:Analysis Manual
7. Push the **Finish** button

How to Create a Copy of an Eigenvalue Method

1. From the **Analysis** category chooser, select **Eigenvalue Methods**
2. Select the Eigenvalue Method you want to copy
3. Push the **Copy Eigenvalue Method** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+C
4. Push the **Paste Eigenvalue Method** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+V

How to Delete an Eigenvalue Method

1. From the **Analysis** category chooser, select **Eigenvalue Methods**
2. Select the Eigenvalue Method to be deleted
3. Push the **Delete Eigenvalue Method** button in the Edit Menu Tool bar

How to Create a Loadcase (Static)

1. Create a Load using for example **How to Create a Static Load (Force, Moment)**
2. Create Boundary Conditions using for example **How to Create Boundary Conditions (Single Point constraint SPC)**
3. From the **Analysis** category chooser, select **Loadcases**
4. Push the **New Loadcase** button in the Edit Menu Toolbar

5. Enter a name in the name field
6. Check the **Static** radio button
7. Push **Next>**
8. From the **SPC** category chooser, select the Grid-Component-Set you have just created
9. Push **Next>**
10. From the **Load Set** category chooser, select the Static Load you have just created
11. Push **Next>**
12. Select Output components for the loadcases
13. Push the **Finish** button

How to Create a Buckling Loadcase

1. Define an Eigenvalue Method using **How to Define an Eigenvalue Method**
2. Define a Static Loadcase using **How to Create a Loadcase (Static)**
3. Select the **Analysis** tab
4. From the category chooser, select **Loadcases**
5. Push the **New Loadcase** button in the Edit Menu Toolbar
6. Enter a name in the name field
7. Select **Buckling** as the loadcase type
8. Push **Next>**
9. Select the Eigenvalue Method you have created
10. Push **Next>**
11. Select a Static Loadcase
12. Push **Next>**
13. Select Output components for the loadcases
14. Push the **Finish** button

How to define an Amplitude Set (DAREA)/Time Lag Set (DELAY)/Phase Shift Set (DPHASE)

1. From the **Analysis** category chooser, select **Frequency Response Data**
2. Push the **New Freq. Resp. Data** button in the Edit Menu Toolbar



3. Enter a name in the **Name** field
4. Select the **Amplitude Set (DAREA)/Time Lag Set (DELAY)/Phase Shift Set (DPHASE)** radio button
5. Push the **Next** button
6. Enter the **Grid Id**
7. Enter the **Component** Number
8. Enter the corresponding **Value**
9. Push the '+' located at the bottom left corner of the **Frequency Response Data** Section and repeat the steps 6 - 9 to add data for more grids
10. Push the **Finish** button

How to Modify an Amplitude Set (DAREA)/Time Lag Set (DELAY)/Phase Shift Set (DPHASE)

1. From the **Analysis** category chooser, select **Frequency Response Data**
2. Select the data that needs to be modified
3. Push the **Modify Freq. Resp. Data** button in the Edit Menu Toolbar
4. Modify the name in the **Name** field
5. Push the **Next** button
6. Enter the **Grid Id**
7. Enter the **Component** Number
8. Enter the corresponding **Value**

To add more data for more grids, Push the '+' located at the bottom left corner repeat the steps 6 - 8

9. Push the **Finish** button

How to create a Frequency Function Table (TABLEDi)

1. From the **Analysis** category chooser, select **Frequency Response Data**
2. Push the **New Freq. Resp. Data** button in the Edit Menu Toolbar
3. Enter a name in the **Name** field
4. Select the **Freq.Function Table (TABLEDi)** radio button
5. Push the **Next** button
6. Select the form for the table entry

7. Enter the parameters necessary

More details about the fields can be obtained by the Volume I:Analysis Manual

8. Enter Values in the **X** and **Y** fields
9. Push the '+' button and repeat step 6 and 7 to add more rows of data
10. Push the **Finish** button

How to Modify a Frequency Function Table (TABLEDi)

1. From the **Analysis** category chooser, select **Frequency Response Data**
2. Select the table that needs to be modified
3. Push the **Modify Freq. Resp. Data** button in the Edit Menu Toolbar
4. Modify the name in the **Name** field
5. Push the **Next** button
6. Modify any parameters or data values as needed
 - To add more data, Push the '+' button
7. Push the **Finish** button

How to create a Modal Damping table (TABDMP1)

1. From the **Analysis** category chooser, select **Frequency Response Data**
2. Push the **New Freq. Resp. Data** button in the Edit Menu Toolbar
3. Enter a name in the **Name** field
4. Select the **Modal Damping Table (TABDMP1)** radio button
5. Push the **Next** button
6. Set the damping units by selecting one of the options from the **Form** category chooser
7. Enter Values in the **X** and **Y** fields for the frequency and damping respectively
8. Push the '+' button and repeat step 7 and 8 to add more rows of data
9. Push the **Finish** button

How to Modify a Modal Damping table (TABDMP1)

1. From the **Analysis** category chooser, select **Frequency Response Data**
2. Select the damping table that needs to be modified

3. Push the **Modify Freq. Resp. Data** button in the Edit Menu Toolbar
4. Modify the name in the **Name** field
5. Push the **Next** button
6. Change the damping units by modifying the option for the **Form**
7. Modify the **X** (Frequency) and **Y** (Damping) values if necessary
Additional data can be added by using the '+' button
8. Push the **Finish** button

How to Define a Dynamic Load Set

1. From the **Analysis** category chooser, select **Frequency Response Data**
2. Push the **New Freq. Resp. Data** button in the Edit Menu Toolbar
3. Enter a name in the **Name** field
4. Select the **Dynamic Load Set (RLOADi)** radio button
5. Push the **Next** button
6. Set the **DAREA** by selecting the options from the category chooser
To create new **DAREA** data, use **How to define an Amplitude Set (DAREA)/Time Lag Set (DELAY)/Phase Shift Set (DPHASE)**
7. Set the **DELAY** by selecting the options from the category chooser
To create new **DELAY** data, use **How to define an Amplitude Set (DAREA)/Time Lag Set (DELAY)/Phase Shift Set (DPHASE)**
8. Set the **DPHASE** by selecting the options from the category chooser
To create new **DPHASE** data, use **How to define an Amplitude Set (DAREA)/Time Lag Set (DELAY)/Phase Shift Set (DPHASE)**
9. Set the **TC Table** by selecting the options from the category chooser
To create new **TCTABLE** data, use **How to create a Frequency Function Table (TABLEDi)**
10. Set the **TD Table** by selecting the options from the category chooser
To create new **TDTABLE** data, use **How to create a Frequency Function Table (TABLEDi)**
11. Set the **Static Load** by selecting the options from the category chooser
To create new **Static Load** data, use **How to Create a Static Load (Force, Moment)**
12. Set the **Gravity Load** by selecting the options from the category chooser
13. Push the **Finish** Button

How to Modify a Dynamic Load Set

1. From the **Analysis** category chooser, select **Frequency Response Data**
2. Select the Dynamic Load Set to be modified
3. Push the **Modify Freq. Resp. Data** button in the Edit Menu Toolbar
4. Modify the name in the **Name** field
5. Push the **Next** button
6. Change the **DAREA** by selecting the options from the category chooser
 To create new **DAREA** data, use **How to define an Amplitude Set (DAREA)/Time Lag Set (DELAY)/Phase Shift Set (DPHASE)**
7. Change the **DELAY** by selecting the options from the category chooser
 To create new **DELAY** data, use **How to define an Amplitude Set (DAREA)/Time Lag Set (DELAY)/Phase Shift Set (DPHASE)**
8. Change the **DPHASE** by selecting the options from the category chooser
 To create new **DPHASE** data, use **How to define an Amplitude Set (DAREA)/Time Lag Set (DELAY)/Phase Shift Set (DPHASE)**
9. Change the **TC Table** by selecting the options from the category chooser
 To create new **TCTABLE** data, use **How to create a Frequency Function Table (TABLEDi)**
10. Change the **TD Table** by selecting the options from the category chooser
 To create new **TDTABLE** data, use **How to create a Frequency Function Table (TABLEDi)**
11. Change the **Static Load** by selecting the options from the category chooser
 To create new **Static Load** data, use **How to Create a Static Load (Force, Moment)**
12. Change the **Gravity Load** by selecting the options from the category chooser
13. Push the Finish Button

How To Copy Frequency Response Data

1. From the **Analysis** category chooser, select **Frequency Response Data**
2. Select the Frequency Response Data to be copied
3. Push the **Copy Freq. Resp. Data** button in the Edit Menu Tool bar
 You can also use the shortcut Ctrl+C
4. Push the **Paste Freq. Resp. Data** button in the Edit Menu Tool bar
 You can also use the shortcut Ctrl+V

How to Delete a Frequency Response Data

1. From the **Analysis** category chooser, select **Frequency Response Data**
2. Select the Frequency Response Data to be deleted from the Frequency Response Data list
3. Push the **Delete Freq. Resp. Data** button in the Edit Menu Tool bar

How to Create Frequency Response Loadcase (Modal Frequency Response)

You will learn how to set up a file in order to plot the Force in an element and get the Modal Contribution table.

1. Define an Eigenvalue Method using **How to Define an Eigenvalue Method**
2. Define a Dynamic Load Set using **How to Define a Dynamic Load Set**
3. Define a Loading Frequency Set using **How to create a Frequency Function Table (TABLEDi)**
4. Define a Modal Damping Set using **How to create a Modal Damping table (TABDMP1)**
5. From the **Analysis** category chooser, select **Loadcases**
6. Push the **New Loadcase** button in the Edit Menu Toolbar
7. Enter a name in the name field
8. Select the **Modal Frequency Response** radio button
9. Push **Next>**
10. Select SPC and/or MPC if needed
11. Push **Next>**
12. Select the Eigenvalue Method created from the **Eigenvalue Method** category chooser
13. Push **Next>**
14. From the **Dynamic Load Set** category chooser, select a Dynamic Load Set
15. From the **Loading Frequency Set** category chooser, select a Loading Frequency Set
16. From the **Modal Damping** category chooser, select a Modal Damping Set
17. Push **Next>**
18. Select Output components for the loadcases
19. Push the **Finish** button

How to Create a Frequency Response Loadcase (Direct Frequency Response)

20. From the **Analysis** category chooser, select **Loadcases**
21. Push the **New Loadcase** button in the Edit Menu Toolbar
22. Enter a name in the name field
23. Check the **Direct Frequency Response** radio button
24. Push **Next>**
25. Select SPC and/or MPC if needed
26. Push **Next>**
27. From the Dynamic Load Set category chooser, select a Dynamic Load Set

You can define a Dynamic Load Set with Design Studio using **How to Define a Dynamic Load Set**
28. From the Loading Frequency Set category chooser, select a Loading Frequency Set

You can define a Loading Frequency Set with Design Studio using **How to create a Frequency Function Table (TABLEDi)**
29. Push **Next>**
30. Select Output components for the loadcases
31. Push the **Finish** button

How to Create a Heat Transfer Loadcase

1. Define a Heat Transfer Load Set

You can't define a Heat Transfer Set with Design Studio. You need to complete the Input file with a text editor.
2. From the **Analysis** category chooser, select **Loadcases**
3. Push the **New Loadcase** button in the Edit Menu Toolbar
4. Enter a name in the name field
5. Check the **Heat Transfer** radio button
6. Push **Next>**
7. Select SPC and/or MPC if needed
8. Push **Next>**
9. From the **Heat Transfer Load Set** category chooser, select a Heat Transfer Load Set
10. Push **Next>**



11. Select Output components for the loadcases
12. Push the **Finish** button

How to Modify a Loadcase

1. From the **Analysis** category chooser, select **Loadcases**
2. Select the Loadcase to be modified in the loadcases list
3. Push the **Modify Loadcase** button in the Edit Menu Tool bar
4. Modify the Loadcase as needed
5. Push the **Finish** button

How to Create a Copy of a Loadcase

1. From the **Analysis** category chooser, select **Loadcases**
2. Select the Loadcase you want to copy
3. Push the **Copy Loadcase** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+C
4. Push the **Paste Loadcase** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+V

How to Delete a Loadcase

1. From the **Analysis** category chooser, select **Loadcases**
2. Select the Loadcase to be deleted in the loadcases list
3. Push the **Delete Loadcase** button in the Edit Menu Tool bar

How to Request the Mode Shape/Displacement to be Output

1. From the **Analysis** category chooser, select **Loadcases**
2. Select a Loadcase
3. Push the **Modify Loadcase** button in the Edit Menu Tool bar
4. Push **Next>**
5. Push **Next>**
6. Push **Next>**
7. Select the **Post** and **All** options from the **Displacement** category choosers

8. Push the **Finish** button

How to Request Element Stresses to be Output

1. From the **Analysis** category chooser, select **Loadcases**
2. Select a loadcase
3. Push the **Modify Loadcase** button in the Edit Menu Tool bar
4. Push **Next>**
5. Push **Next>**
6. Push **Next>**
7. Select the **Post** option from the **Element Stress** category chooser
8. Push the **Finish** button

How to Request Element Forces to be Output

1. From the **Analysis** category chooser, select **Loadcases**
2. Select a loadcase
3. Push the **Modify Loadcases** button in the Edit Menu Tool bar
4. Push **Next>**
5. Push **Next>**
6. Push **Next>**
7. Select the **Post** option from the **Element Force** category chooser
8. Push the **Finish** button

How to Create a Coordinate System

1. From the **Analysis** category chooser, select **Coordinate Systems**
2. Push the **New Coordinate System** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Enter **Origin X**
5. Enter **Origin Y**
6. Enter **Origin Z**
7. Alternatively, an existing grid can be selected
8. Select type of Coordinate system

9. Push **Next>**
10. Select the direction of the axes
Direction of the axes can be defined by entering values or by two grids using the **Choose Two Grids..** button
11. Push the **Finish** button

How to Modify a Coordinate System

1. From the **Analysis** category chooser, select **Coordinate Systems**
2. Select the Coordinate system to modify
3. Push the **Modify Coordinate System** button in the Edit Menu Toolbar
4. Change the name if necessary
5. Change the origin of the coordinate system, by selecting a grid or changing the **Origin X**, **Origin Y** and **Origin Z** values
6. Change type of Coordinate system if needed
7. Push **Next>**
8. Change the direction of the axes if necessary
Direction of the axes can be defined by entering values or by two grids using the **Choose Two Grids..** button
9. Push the **Finish** button

How to Create a Copy of a Coordinate System

1. From the **Analysis** category chooser, select **Coordinate Systems**
2. Select the Coordinate system you want to copy
3. Push the **Copy Coordinate System** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+C
4. Push the **Paste Coordinate System** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+V

How to Delete a Coordinate System

1. From the **Analysis** category chooser, select **Coordinate Systems**
2. Select the Coordinate System to be deleted
3. Push the **Delete Coordinate System** button in the Edit Menu Tool bar

How to Create a New Material

1. From the **Analysis** category chooser, select **Materials**
2. Push the **New Material** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the material type to be created
5. Push **Next>**
6. Enter the appropriate material properties

Make sure that the units used are appropriate and consistent with the model. You can also refer to the Volume I: Overview guide

7. Push the **Finish** button

How to Modify a Material

1. From the **Analysis** category chooser, select **Materials**
2. Select the Material to modify
3. Push the **Modify Material** button in the Edit Menu Toolbar
4. Change name if necessary
5. Change the material type if needed
6. Push **Next>**
7. Change the appropriate material properties

Make sure that the units used are appropriate and consistent with the model. You can also refer to the Volume I: Overview guide

8. Push the **Finish** button

How to Create a Copy of a Material

1. From the **Analysis** category chooser, select **Materials**
2. Select the Material you want to copy
3. Push the **Copy Material** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+C

4. Push the **Paste Material** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+V

How to Delete a Material



1. From the **Analysis** category chooser, select **Materials**
2. Select the Material to be deleted
3. Push the **Delete Material** button in the Edit Menu Tool bar

How to Define Group Properties

1. From the **Analysis** category chooser, select **Group Properties**
2. Push the **New Group Property** button in the Edit Menu Toolbar
3. Select a Type From the **Type** category chooser
4. Enter a name in the name field
5. Select an intended color
6. Push **Next>**
7. Select a Material
8. Enter the properties of the Group
9. Push the **Finish** button

How to Modify Group Properties

1. From the **Analysis** category chooser, select **Group Properties**
2. Select the group property to modify
3. Push the **Modify Group Property** button in the Edit Menu Toolbar
4. Change name if necessary
5. Change the material type if needed
6. Change the appropriate properties
7. Push the **Finish** button

How to Create a Copy of a Group Property

1. From the **Analysis** category chooser, select **Group Properties**
2. Select the group property you want to copy
3. Push the **Copy Group Property** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+C
4. Push the **Paste Group Property** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+V

How to Delete a Group Property

1. From the **Analysis** category chooser, select **Group Properties**
2. Select the group property to be deleted
3. Push the **Delete Group Property** button in the Edit Menu Tool bar

How to Create a Composite Failure Equation

1. From the **Analysis** category chooser, select **Composite Failure Equations**
2. Push the **New Findex** button in the Edit Menu Toolbar
3. Select a Type from the radio buttons
4. Modify the name of the equation parameters
5. Enter an equation using the equation parameters names
6. Push the **Finish** button

How to Create a Copy of a Composite Failure Equation

1. From the **Analysis** category chooser, select **Composite Failure Equations**
2. Select the Composite Failure Equation you want to copy
3. Push the **Copy Findex** button in the Edit Menu Tool bar
4. Push the **Paste Findex** button in the Edit Menu Tool bar

You can also use the shortcut Ctrl+C

You can also use the shortcut Ctrl+V

How to Delete a Composite Failure Equation

1. From the **Analysis** category chooser, select **Composite Failure Equations**
2. Select the Composite Failure Equation to be deleted
3. Push the **Delete Findex** button in the Edit Menu Tool bar

How to Create Multiple Elements

1. Create a New Material using **How to Create a New Material**
2. Define a Group Property using **How to Define Group Properties**
3. From the **Analysis** category chooser, select **Elements**
4. Push the **New Elements** button in the Edit Menu Toolbar



5. Check the **Define New Original Elements** radio button
6. Push **Next>**
7. Select the group you have just created from the list
8. Push **Next>**
9. 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.
10. Select a type of region icon in **Region Definition option**
11. Enter the points coordinates or pick in the Viewport to create the region
12. In the list, select the region you just created
13. Push the **Subdivide** button
14. Enter the number of Elements in each Dimension

The first dimension is displayed by a red line on the shape domain in the Viewport.
The second dimension is displayed by a green line.
The third dimension is displayed by a blue line.
15. Push **Next>**
16. Push the **Finish** button

How to Delete Elements

1. Hide the Elements you do not want to delete using **How to Show/Hide Elements**
2. From the **Analysis** category chooser, select **Elements**
3. Select Elements to be deleted using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps

If you want to delete all the displayed elements, go to the main menu bar, select **Edit** → **Select All**
4. Push the **Delete Elements** button in the Edit Menu Tool bar

How to Create Mirror Copies of Elements

1. From the **Analysis** category chooser, select **Elements**
2. Push the **New Elements** button in the Edit Menu Toolbar
3. Check the **Duplicate Selected Elements (use new mirrored grids)** radio button
4. Push **Next>**

5. Select Elements to be mirrored using **How to Select Element(s) by picking in the Viewport or How to Select Element(s) by Element ID**
6. Push the **Next** button
7. Push the **Change** button to set the Ref. Coord. System
8. Select the desired Coordinate system
9. Push the **Next** button
10. Enter values for the **Center X1, Center X2, Center X3** fields
Alternatively you can select a grid using **How to Select Grid(s) by picking in the Viewport/ How to Select Grid(s) by Grid ID**
11. Set the mirror plane by selecting from the category chooser
12. Push **Next>**
13. Select the **Group** for the new elements from the existing groups
14. Push the **Finish** button

How to Create Multiple Elements with Optional Bias for Nonuniform Mesh Generation

1. Create a New Material using **How to Create a New Material**
2. Define a Group Property using **How to Define Group Properties**
3. From the **Analysis** category chooser, select **Elements**
4. Push the **New Elements** button in the Edit Menu Toolbar
5. Check the **Define New Original Elements** radio button
6. Push **Next>**
7. Select the group you have just created from the list
8. Push **Next>**
9. 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.
10. Select a type of region icon in **Region Definition option**
11. Enter the points coordinates or pick in the Viewport to create the region
12. In the list, select the region you just created
13. Push the **Subdivide** button

14. Enter the number of Elements in each Dimension
15. Enter the desired bias value in the **Bias** field
16. Check the desired Bias Type (**Center/Edge**) radio button

The first dimension is displayed by a red line on the shape domain in the Viewport.
The second dimension is displayed by a green line.
The third dimension is displayed by a blue line.
17. Push **Next>**
18. Push the **Finish** button

How to Change Elements' Group

1. From the **Analysis** category chooser, select **Elements**
2. Select the elements to be moved, using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps
3. Push the **Modify Elements** button in the Edit Menu Tool bar
4. Select the **Change Element's Group** radio button
5. Push **Next>**
6. Select the New Group for the Elements

If you want to have a new Group for these elements only, you need to create a new Group using **How to Define Group Properties**
7. Push the **Finish** button

How to Flip 2-D Elements' Orientation

1. From the **Analysis** category chooser, select **Elements**
2. Select the elements to be modified using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps
3. Push the **Modify Elements** button in the Edit Menu Tool bar
4. Select the **Flip 2-D Element's Orientation** radio button
5. Push the **Finish** button

How to Align 2-D Elements' Orientation

1. From the **Analysis** category chooser, select **Elements**
2. Select the elements to be modified using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps

3. Push the **Modify Elements** button in the Edit Menu Tool bar
4. Select the **Align 2-D Element's Orientation** radio button
5. Push the **Finish** button

How to Change 2-D Elements' Theta

1. From the **Analysis** category chooser, select **Elements**
2. Select the elements to be modified using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps
3. Push the **Modify Elements** button in the Edit Menu Tool bar
4. Select the **Change 2-D Element's Theta** radio button
5. Push **Next>**
6. Select a **Theta Coordinate System**
7. Push the **Finish** button

How to Split Elements into Smaller Elements

1. From the **Analysis** category chooser, select **Elements**
2. Select the elements to be modified using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps
3. Push the **Modify Elements** button in the Edit Menu Tool bar
4. Select the **Split into Smaller Elements** radio button
5. Push the **Finish** button

How to Check the Element Norms

1. From the **Analysis** category chooser, select **Elements**
2. Select the elements to be checked using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps
3. Push the **Generate Orientation Vectors** button

How to Check the Theta Vectors

1. From the **Analysis** category chooser, select **Elements**
2. Select the Elements to be checked using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps



3. Push the **Generate Theta Vectors** button

How to Create a Skin

1. From the **Analysis** category chooser, select **Elements**
2. Push the **New Elements** button in the Edit Menu Toolbar
3. Select the **Surface elements from Selected Solid Elements** option
4. Push **Next>**
5. Select the Solid Elements needed
6. Push **Next>**
7. Select a group
8. Push the **Finish** button

The skin is hidden, as it uses same grids of the solid parts.

How to Create a Grid

You will create a Grid at the following coordinates: (-7.0, 4.0, 0.0)

1. From the **Analysis** category chooser, select **Grids**
2. Push the **New Grids** button in the Edit Menu Toolbar
3. Enter 1 as **Number of New Grids**
4. Accept the default option **Along a Line**
5. Push **Next>**
6. Enter **Base X1** (e.g -7.0), **Base X2** (e.g 4.0) and **Base X3**
7. Enter the **Offset Scale Factor**
8. Push the **Finish** button

How to Create Multiple Grids

1. From the **Analysis** category chooser, select **Grids**
2. Push the **New Grids** button in the Edit Menu Toolbar
3. Enter the **Number of New Grids** to create
4. Select the **Along a line** radio button

You can also create Grids along an arc by selecting the **Along an arc** radio button

5. Enter the **Base X1**, **Base X2** and **Base X3**
For the along a line option, these correspond to the starting grid coordinates.
For the along an arc option, these correspond to the arc center coordinates.
6. Enter an **Offset U1** value, an **Offset U2** value and an **Offset U3** value
These define the direction of the line
7. Enter the **Offset Scale Factor**
8. Push the **Finish** button

How to Translate Grid(s) along a Line

1. From the **Analysis** category chooser, select **Grids**
2. Select the Grid(s) to be translated using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
3. Push the **Modify Grids** button in the Edit Menu Tool bar
4. Select the **Translate Along a Line** radio button
5. Push **Next>**
6. Enter an **Offset U1** value, an **Offset U2** value and an **Offset U3** value
These define the direction of the translation.
7. Enter a **Offset Scale Factor** value
This defines the magnitude translation
8. Push the **Finish** button

How to Rotate Grid(s) About an Axis

1. From the **Analysis** category chooser, select **Grids**
2. Select the Grid(s) to be rotated using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
3. Push the **Modify Grids** button in the Edit Menu Tool bar
4. Select the **Rotate About an Axis** radio button
5. Push **Next>**
6. Enter a **Center X1** value, a **Center X2** value and a **Center X3** value
These define the center coordinates of the rotation.
7. Select the Rotation Axis from the category chooser
8. Enter an **Angle Increment Value** in degrees



9. Push the **Finish** button

How to Delete Grid(s)

You can only delete free grids

1. From the **Analysis** category chooser, select **Grids**
2. Pick in the Viewport the grid(s) you want to delete

If you want to delete all the free grids, go to the main menu bar, select **Edit** → **Select All**
It is easier to view the free grids by hiding all the existing groups using **How to Show/Hide Groups** and by displaying the free grids using **How to Show/Hide Free Grids**.

3. Push the **Delete Grids** button in the Edit Menu Tool bar

2.4 How to use the functions of the Topology tab

How to Quickly Set Up a Common Topology Problem

1. From the **Topology** category chooser, select **Quick Setup Trails**
2. Push the **Quick Topology Setup** button
3. Select the Group to be designed
4. Push **Next>**
5. Select an objective radio button
6. Enter a mass fraction bound value as constraint
7. Push **Next>**
8. Select a loadcase
9. Push the **Finish** button

How to Check a Topology Problem

1. From the **Topology** category chooser, select **Quick Setup Trails**
2. Check whether there are any Topology regions, objectives and constraints are defined.

A summary of the Topology data is displayed in this panel.

How to Define a Topology Region

1. From the **Topology** category chooser, select **Topology Regions**
2. Select the Group(s) from the list
3. Push the **Modify Topology Design** button from the Edit Menu tool bar
4. Enter an Initial Mass Fraction
5. Push **Next>**

You can change the coordinate system if needed. To do this: push the **Change** button, choose the coordinate system and then push the **Next>** button.

6. Select Fabrication Constraint(s) if needed.

You can choose between: Mirror about a plane (XY, YZ or XZ), Extrude along an axis (X, Y or Z), Fill an axis (inside to out), Fill an axis (- to +), Fill an axis (+ to -), Fill an axis (outside to in) and Cyclic about an axis.



7. Enter a value of the **Minimum Size** to define the minimum member size for the member
8. Enter a value for the **Spread Fraction** to define the spread
9. Push the **Finish** button

How to Remove Topology Design Data

1. From the **Topology** category chooser, select **Topology Regions**
2. Select the Group(s) from the list
3. Push **Delete Topology Design** button

Verify that there is no hammer icon next to the removed group after you push the button.

How to Define a Topology Objective(s)

1. From the **Topology** category chooser, select **Topology Objectives**
2. Push the **New Topology Objective** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the response type you want to optimize
5. Select the Objective Definition Switch you need
6. Push **Next>**
7. Select the loadcase(s)
8. Push the **Finish** button

How to Delete a Topology Objective

1. From the **Topology** category chooser, select **Topology Objectives**
2. Select the objective to be deleted
3. Push the **Delete Topology Objective** button in the Edit Menu Tool bar

How to Define a Topology Constraint(s)

1. From the **Topology** category chooser, select **Topology Constraints**
2. Push the **New Topology constraint** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the Response Type you want to Constrain

5. Enter the Constraint Bounds
6. Push the **Finish** button

How to Create a Copy of a Topology Constraint

1. From the **Topology** category chooser, select **Topology Constraints**
2. Select the Constraints you want to copy
3. Push the **Copy Topology Constraint** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+C
4. Push the **Paste Topology Constraint** button in the Edit Menu Tool bar
You can also use the shortcut Ctrl+V

How to Define a Topology Buckling Constraint

1. From the **Topology** category chooser, select **Topology Constraints**
2. Push the **New Topology Constraint** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Buckling Load Factor** response
5. On the Buckling Load Factor Box, enter a value
6. Enter a **Lower Bound** value and/or a **Upper Bound** value
7. Push **Next>**
8. Select a Buckling loadcase
You can create a Buckling loadcase using **How to Create a Buckling Loadcase**
9. Push the **Finish** button

How to Delete a Topology Constraint

1. From the **Topology** category chooser, select **Topology Constraints**
2. Select the constraint to be deleted
3. Push the **Delete Topology Constraint** button in the Edit Menu Tool bar
4. Push the **Finish** button

How to Create an Extra Design Variable

1. From the **Topology** category chooser, select **Extra Variables**

2. Push the **New Topology Extra Variable** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Independent Topology Variable (TVAR)** radio button
5. Push **Next>**
6. Enter an **Initial Value**, a **Lower Bound**, and an **Upper Bound**
7. 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 **Independent** design variable.

How to Create a Constant Value (DTABLE)

1. From the **Topology** category chooser, select **Extra Variables**
2. Push the **New Topology Extra Variable** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Constant Value (DTABLE)** radio button
5. Enter an integer value in the **Label** field
6. Enter a value in the **Value** field
7. Push the **Finish** button

How to Create a Topology Synthetic Response

For the example, you can define following synthetic equation: $F = (3 * Arg1 + Arg2) / 4$

1. From the **Topology** category chooser, select **Synthetic Responses**
2. Push the **New Topology Synthetic Response** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Leave **User Function (TRESP2)** as the type
5. Push **Next>**
6. Push the + button to create the first argument of the function
7. Select the type of the argument

Design variables, grid locations, fundamental responses can be used as arguments:

For Design variable, select the Design Variable radio button, select the Design Variable from the category chooser and push **Next>**.

Or for Grid locations, select the Grid point using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps, select the **Grid X Coordinate**, **Grid Y Coordinate** or **Grid Z Coordinate** radio button and push **Next>**.

Or for Fundamental Response, select the **Fundamental Response...** radio button, choose the

Response, choose the option if needed (loadcase, groups,...) and push **Next>**.

8. Push the + button to add the argument of the function
9. Once all the arguments are added, write the synthetic equation in terms of the arguments (e.g: $F = (3 * Arg1 + Arg2) / 4$)
10. Push the **Finish** button

How to Define a Topology Synthetic Constraint

1. From the **Topology** category chooser, select **Topology Constraints**
2. Push the **New Topology Constraint** button in the Edit Menu Toolbar
3. Select **Synthetic Response** as the response type
4. Enter a **Lower Bound** value and/or an **Upper Bound** value
5. Push **Next>**

Note that at this point if there are no synthetic responses in the list., you will need to create a synthetic response with a new trail and then come back to where you are now.
6. Create a Synthetic Response using **How to Create a Topology Synthetic Response**
7. From the **Topology** category chooser, select **Topology Constraints**

Design Studio allows you to return the same position in the constraints trail. Notice that the synthetic response created is listed.
8. Select the newly created synthetic response
9. Push **Next>**
10. Select loadcase
11. Push the **Finish** button

2.5 How to use the functions of the Design tab

How to Use the Quick Setup Trails

1. From the **Design** category chooser, select **Quick Setup Trails**

A summary of the design data is shown in the panel.

2. Push the **Quick Sizing Setup** button
3. Select the Group(s) to be designed

Groups that are already being designed will not be displayed in the list.

4. 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.

5. For **C1** and **M1**: Enter values
6. For **C2** and **M2**: Enter values
7. For **C3** and **M3**: Enter values
8. Push the **Finish** button

How to Create a Synthetic Response based on an equation

For the example, you can define following synthetic equation: $F=(3*Arg1+Arg2)/4$

1. From the **Design** category chooser, select **Synthetic Responses**
2. Push the **New Synthetic Response** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Leave **User Function (DRESP2)** as the type
5. Push **Next>**
6. Push the + button to create the first argument of the function
7. Select the type of the argument

Design variables, grid locations, fundamental responses can be used as arguments:

For Design variable, select the Design Variable radio button, select the Design Variable from the category chooser and push **Next>**.

Or for Grid locations, select the Grid point using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps, select the **Grid X Coordinate**, **Grid Y**

Coordinate or **Grid Z Coordinate** radio button and push **Next>**.

Or for Fundamental Response, select the **Fundamental Response...** radio button, choose the Response, choose the option if needed (loadcase, groups,...) and push **Next>**.

8. Push the + button to add the argument of the function
9. Once all the arguments are added, write the synthetic equation in terms of the arguments (e.g: $F=(3*Arg1+Arg2)/4$)
10. Push the **Finish** button

How to Create a Design Variables Selection Constraint (DSELECT)

1. From the **Design** category chooser, select **Synthetic Responses**
2. Push the **New Synthetic Response** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Design Variable Selection (DSELECT)**
5. Push **Next>**
6. Push the + button to add an argument
7. Select a Design Variable
8. Push **Next>**
9. Repeat the steps 6-8 to add more arguments
10. Enter a **Fraction** number
11. Push the **Finish** button

How to Define a Matching Design Objective (DMATCH)

1. From the **Design** category chooser, select **Objectives**
2. Push the **New Objective** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the Response Type, which is your objective
5. Select the **Target** radio button as Objective Definition Switch
6. Enter the Target Value in the text box adjoining the **Target** radio button.
7. Push **Next>** and setup the response type if needed
8. Push the **Finish** button



How to Define a Design Objective

1. From the **Design** category chooser, select **Objectives**
2. Push the **New Objective** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the Response Type, which is your objective
5. Select the desired Objective Definition Switch
6. Push **Next>** and setup the response type if needed
7. Push the **Finish** button

How to Delete the Objective

1. From the **Design** category chooser, select **Objectives**
2. Select the objective to be deleted
3. Push the **Delete Objective** button in the Edit Menu Tool bar

How to Define a Stress Constraint for Group(s)

1. From the **Design** category chooser, select **Constraints**
2. Push the **New Constraint** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Stress** radio button as Response Type
5. Select **Selected Groups** from **Stress** option
6. Enter a **Lower Bound** and/or an **Upper Bound**
7. Push **Next>**
8. Select the group(s) to be constrained
9. Push **Next>**
10. Set up the **Stress Components** option as you need
11. Push **Next>**
12. Select static loadcase
13. Push the **Finish** button

How to Define a Stress Constraint for Element(s)

1. From the **Design** category chooser, select **Constraints**
2. Push the **New Constraint** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Stress** radio button as Response Type
5. Select **Selected Elements** from **Stress** option
6. Enter a **Lower Bound** value and/or a **Upper Bound** value
7. Push **Next>**
8. Select the element(s) using **How to Select Element(s) by Element ID** or **How to Select Element(s) by picking in the Viewport** steps
9. Push **Next>**
10. Set up the **Stress Components** option as you need
11. Push **Next>**
12. Select static loadcase
13. Push the **Finish** button

How to Define a Frequency Mode Number Constraint

1. From the **Design** category chooser, select **Constraints**
2. Push the **New Constraint** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Frequency Mode Number** radio button as the Response Type
5. Enter a value in the Frequency Mode Number field
6. Enter a **Lower Bound** value or an **Upper Bound** value
7. Push **Next>**
8. Select a Normal Modes analysis loadcase
9. Push the **Finish** button

How to Define a Displacement Constraint

1. From the **Design** category chooser, select **Constraints**
2. Push the **New Constraint** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Displacement** radio button as Response Type



5. Enter a **Lower Bound** value and/or an **Upper Bound** value
6. Push **Next>**
7. Select the grid(s) using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
8. Select an option radio button for **Select Component**
9. Push **Next>**
10. Select static loadcase
11. Push the **Finish** button

How to Define a Synthetic Constraint

1. From the **Design** category chooser, select **Constraints**
2. Push the **New Constraint** button in the Edit Menu Toolbar
3. Select **Synthetic Response** as the response type
4. Enter a **Lower Bound** value and/or an **Upper Bound** value
5. Push **Next>**

Note that at this point if there are no synthetic responses in the list., you will need to create a synthetic response with a new trail and then come back to where you are now.
6. Create a Synthetic Response using **How to Create a Synthetic Response based on an equation**
7. From the **Design** category chooser, select **Constraints**

Design Studio allows you to return the same position in the constraints trail. Notice that the synthetic response created is listed.
8. Select the newly created synthetic response
9. Push **Next>**
10. Select loadcase
11. Push the **Finish** button

How to Define a Geometric Constraints (DRESPG)

1. From the **Design** category chooser, select **Constraints**
2. Push the **New Constraint** button in the Edit Menu Toolbar
3. Select **More Response Types..** as the response type
4. Enter a **Lower Bound** value and/or an **Upper Bound** value

5. Push **Next>**
6. Select the **Geometric** radio button
7. Push **Next>**
8. Select the grid(s) using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
9. Select a Geometric Responses type
10. Push the **Finish** button

How to Define Grid Stress Constraints

1. From the **Design** category chooser, select **Constraints**
2. Push the **New Constraint** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **More Response Types..** radio button as the Response Type
5. Enter a **Lower Bound** value and/or an **Upper Bound** value
6. Push **Next>**
7. Select the **Grid Stress** radio button in Additional Responses
8. Select the stress type in the options for **Grid Stress**
9. Push **Next>**
10. Select the grid(s) using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
11. Push **Next>**
12. Select static loadcases
13. Push the **Finish** button

How to Define Temperature Constraints

1. From the **Design** category chooser, select **Constraints**
2. Push the **New Constraint** button in the Edit Menu Toolbar
3. Select the **More Response Types..** radio button as the Response Type
4. Enter a **Lower Bound** and an **Upper Bound**
5. Push **Next>**
6. Select the **Temperature** radio button as the Additional Responses



7. Select the grid(s) using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
8. Push **Next>**
9. Select loadcase (heat transfer)
10. Push the **Finish** button

How to Delete a Constraint

1. From the **Design** category chooser, select **Constraints**
2. Select the constraint to be deleted

If you want to delete all the constraints: from the main menu bar, select **Edit** → **Select All**
3. Push the **Delete Constraint** button in the Edit Menu Tool bar

How to Create an Independent Design Variable

1. From the **Design** category chooser, select **Design Variables**
2. Push the **New Design Variable** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Independent Design Variable** radio button
5. Push **Next>**
6. Enter an **Initial Value**, a **Lower Bound**, and an **Upper Bound**
7. 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 “**T**” indicates this is an **Independent** design variable.

How to Create a Discrete Design Variable

1. From the **Design** category chooser, select **Design Variables**
2. Push the **New Design Variable** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Discrete Design Variable** radio button
5. Push **Next>**
6. Enter a **Lower Bound**, and an **Upper Bound**
7. Enter a **Discrete Value**

8. Push the **+** button to add an additional discrete value

Alternatively, a list of discrete values can be generated using first discrete value, increment and number of values

To remove values from the discrete set, use the **-** button adjacent to the value

9. 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 **Discrete** design variable.

How to Create a Linked Design Variable (DLINK)

1. From the **Design** category chooser, select **Design Variables**
2. Push the **New Design Variable** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Linked Design Variable** radio button
5. Push **Next>**
6. Enter a **Lower Bound**, and an **Upper Bound**
7. Select your Master Variable from the **Master Variable** list
8. Enter a coefficient in the **Coefficient** field
9. Push the **+** button to add new arguments if needed
10. Push the **Finish** button

Notice that an asterisk (*) is in front of the linked design variable, which indicates that this design variable is not being used yet. The “**L**” next to the asterisk indicates this is a **Linked** design variable.

How to Create a Synthetic Design Variable

1. From the **Design** category chooser, select **Design Variables**
2. Push the **New Design Variable** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Synthetic Design Variable** radio button
5. Push **Next>**
6. Select your Master Variable from the Master Variable list
7. Enter a coefficient in the Coefficient field
8. Push the **Finish** button

Notice that an asterisk (*) is in front of the synthetic design variable, which indicates that this



design variable is not being used yet. The “S” next to the asterisk indicates this is a Synthetic design variable.

How to Create an Equation Design Variable

1. From the **Design** category chooser, select **Design Variables**
2. Push the **New Design Variable** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Equation Design Variable** radio button
5. Push **Next>**
6. Select a Design Variable from the **Master Variable** category chooser
7. Push the + button to add new arguments
8. Enter your equation as a function of all arguments

In the **Equation Parameter** list, replace Arg1 by X, then push the keyboard key Return or Enter

Notice that if you replace Arg1 by X, Design Studio will automatically update the equation.

9. Push the **Finish** button

Notice that an asterisk (*) is in front of the equation design variable, which indicates that this design variable is not being used yet. The “E” next to the asterisk indicates this is a **Equation** design variable.

How to Create a Constant Value (DTABLE)

1. From the **Design** category chooser, select **Design Variables**
2. Push the **New Design Variable** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Constant Value (DTABLE)** radio button
5. Enter an integer value in the **Label** field
6. Enter a value in the **Value** field
7. Push the **Finish** button

How to Modify Created Design Variables

1. From the **Design** category chooser, select **Design Variables**
2. Select the design variable to be checked
3. Push the **Modify Design Variable** button in the Edit Menu Tool bar

4. Push **Next>**
5. Change any parameters if necessary
6. Push the **Finish** button

How to Change Lower and Upper Bounds of all Design Variables

1. From the **Design** category chooser, select **Design Variables**
2. Select all the design variables
3. Push the **Modify Design Variable** button in the Edit Menu Tool bar
4. Push **Next>**

You can change only the same type of variables in the same time. If you select many type (Independent and Synthetic for example) you will need to push **Next>** between each type.

5. Put a checkmark in the **Lower bound** checkbox
6. For **Lower Bound** enter a value
7. Put a checkmark in the **Upper bound** checkbox
8. For **Upper Bound** enter a value
9. Push the **Finish>** button

How to Delete Existing Design Variables

1. From the **Design** category chooser, select **Design Variables**
2. Select the Design Variables to be deleted

If you want to delete all the Design Variables: from the main menu bar, select **Edit** → **Select All**

3. Push the **Delete Design Variable** button in the Edit Menu Tool bar

How to Define a Sizing Optimization

1. From the **Design** category chooser, select **Sizing**
2. Select the Group(s) to be optimized
3. Push the **Modify Sizing Design** button in the Edit Menu Tool bar
4. Setup the properties for sizing optimization by assigning design variables

If you do not have a design variable, you need to create one using **How to Create an Independent Design Variable**

5. Push the **Finish** button

How to Modify Created Sizing Data

1. From the **Design** category chooser, select **Sizing**
2. Select the Group to be modified
3. Push the **Modify Sizing Design** button in the Edit Menu Tool bar
4. Change any of the properties for sizing optimization
5. Push the **Finish** button

How to Assign a Design Variable for Designing the Location of the Reference Plane, Z0 (PCOMP sizing)

1. From the **Design** category chooser, select **Sizing**
2. Select a PCOMP Group
3. Push the **Modify Sizing Design** button in the Edit Menu Tool bar
4. For **Z0**, select the design variable you have just created

If you do not have a design variable, you need to create one using **How to Create an Independent Design Variable**
Usually, Z0 is created using a non independent variable.
5. Push the **Finish** button

How to Add Design Variables to Design the Angles/Thickness of a PCOMP Group

1. From the **Design** category chooser, select **Sizing**
2. Select a PCOMP Group
3. Push the **Modify Sizing Design** button in the Edit Menu Tool bar
4. For each designable **Angle** of each **Layer**, select a design variable from the category chooser

If you do not have a design variable, you need to create one using **How to Create an Independent Design Variable**
5. For each designable **Thickness** of each **Layer**, select the design variable from the category chooser

If you do not have a design variable, you need to create one using **How to Create an Independent Design Variable**
6. Push the **Finish** button

How to Define a Topometry Optimization

1. Define a Sizing Optimization for the group(s) you want to run Topometry using **How to Define a Sizing Optimization**
2. From the **Design** category chooser, select **Topometry**
3. Select the Group(s) to be optimized
4. Push the **Modify Topometry Design** button in the Edit Menu Tool bar
5. Push the **Finish** button

How to Define Symmetry Conditions for Topometry Optimization

1. From the **Design** category chooser, select **Topometry**
2. Select the Group to be defined
3. Push the **Modify Topometry Design** button in the Edit Menu Tool bar

Set up Symmetry Conditions:

You will now set up symmetry conditions in the selected topometry regions. For example, the desired constraint is mirror symmetry with respect to the YZ plane of the local coordinate system 1.

4. Push the **Change** button to change to a different coordinate system
5. Select the symmetry constraints needed (e.g **MYZ: Mirror about YZ plane**)
6. Push **Next>**
7. Push the **Finish** button

How to Set the Coarse Conditions in Topometry

1. From the **Design** category chooser, select **Topometry**
2. Select a group
3. Push the **Modify Topometry Design** button in the Edit Menu Tool bar
4. For **Coarse Method**: select **Max elements per design variable**
5. For **Coarse Parameter**, enter a value
6. Push the **Finish** button

How to Set a Topography Region

1. From the **Design** category chooser, select **Topography**
2. Select a group

3. Push the **Modify Topography Design** button in the Edit Menu Tool bar
4. Select a **Shape Type**
5. Enter values for the parameters needed for the selected **Shape Type**
The pictures on the bottom of the panel show what each parameter means.
6. Push **Next>**
7. Select the grid(s) to be designed using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
8. Push **Next>**
9. Define the direction of the perturbation
10. Push the **Finish** button

How to Set up a Topography Optimization with Bead Fraction

1. From the **Design** category chooser, select **Topography**
2. Select a group
3. Push the **Modify Topography Design** button in the Edit Menu Tool bar
4. Select a **Shape Type**
5. Enter values for the parameters needed for the selected **Shape Type**
The pictures on the bottom of the panel show what each parameter means.
6. Enter a number for the **Bead Fraction**
7. Push the **Finish** button

How to Create a Shape Domain

If you want to create an axisymmetric bar shape domain, use the following steps:

How to Create an Axisymmetric Bar Shape Domain

If you want to create an axisymmetric quad shape domain, use the following steps:

How to Create an Axisymmetric Quad Shape Domain

If you want to create a quad shape domain, use the following steps:

How to Create a Quad Shape Domain

If you want to create multiple quad shape domain, use the following steps:

How to Create Multiple Quad Shape Domains

If you want to create multiple hexa shape domain, use the following steps:

How to Create Multiple Hexa Shape Domains

How to Create an Axisymmetric Bar Shape Domain

1. From the **Design** category chooser, select **Shape Domains**
2. Push the **New Domain** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Check the **Define A New Original Domain** radio button
5. Push **Next>**
6. Select **Create New Domain Group** item
7. Push **Next>**
8. Select the **Axisymmetric Bar** radio button
9. Push **Next>**
10. Select 2 grids to define the bar using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
11. Select 2 grids to define the symmetry axis using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
12. Push **Next>**
13. Enter radius value in the **Radius for non-3D Domains** field
14. Push the **Select Interior Grids** button
15. Push the **Finish** button

How to Create an Axisymmetric Quad Shape Domain

1. From the **Design** category chooser, select **Shape Domains**
2. Push the **New Domain** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Check the **Define A New Original Domain** radio button
5. Push **Next>**
6. Select **Create New Domain Group** item
7. Push **Next>**
8. Select the **Axisymmetric Quad** radio button

9. Push **Next>**
10. Select 4 grids to define the quad using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
11. Select 2 grids to define the symmetry axis using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
12. Push **Next>**
13. Enter radius value in the **Radius for non-3D Domains** field
14. Push the **Select Interior Grids** button
15. Push the **Finish** button

How to Create a Quad Shape Domain

1. From the **Design** category chooser, select **Shape Domains**
2. Push the **New Domain** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Check the **Define A New Original Domain** radio button
5. Push **Next>**
6. Select **Create New Domain Group** item
7. Push **Next>**
8. Select the **Quad** radio button
9. Push **Next>**
10. Select 4 grids to define the quad using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
11. Push **Next>**
12. Enter radius value in the **Radius for non-3D Domains** field
13. Push the **Select Interior Grids** button
14. Push the **Finish** button

How to Create Multiple Quad Shape Domains

1. From the **Design** category chooser, select **Shape Domains**
2. Push the **New Domain** button in the Edit Menu Toolbar
3. Enter a name in the name field

4. Select a type of the domain
5. Check the **New Domain Quick Setup** radio button
6. Push **Next>**
7. Select **Create New Domain Group** item
8. Push **Next>**
9. Check the **Drag out size** radio button

You can also pick point by point
10. Check the **Pick Points on Workplane** radio button
11. Select a **Plane of view** icon
12. Select the **Quads** icon as **Region Definition Option**
13. Create a Domain by dragging in the main window
14. Select the Quad region you just created
15. Push the **Subdivide** button
16. Enter a number of Elements in 1st Dimension
17. Enter a number of 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.
18. Push **Next>**
19. Push the **Finish** button

How to Create Multiple Hexa Shape Domains

1. From the **Design** category chooser, select **Shape Domains**
2. Push the **New Domain** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Check the **New Domain Quick Setup** radio button
5. Push **Next>**
6. Select **Create New Domain Group** item
7. Push **Next>**
8. Check the **Drag out sizes** radio button
9. Check the **Pick Points on Workplane** radio button
10. Select a **Plane of view** icon

11. Select the **Hexas** icon as **Region Definition Option**
12. Create a Domain by dragging
13. Select the Hexa region you just created
14. Push the **Subdivide** button
15. Enter a number of Elements in 1st Dimension
16. Enter a number of Elements in 2nd Dimension
17. Enter a number of 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.

18. Push **Next>**
19. Push the **Finish** button

How to Create Copies of Domains by Translation

1. From the **Design** category chooser, select **Shape Domains**
2. Push the **New Domain** button in the Edit Menu Toolbar
3. Select the **Duplicate Selected Domains (use new translated grids)** radio button
4. Enter a name in the **Name** field
5. Push **Next>**
6. Select the Domain(s) to be copied from the existing set of Domains
7. Push **Next>**
8. Push the **Change** button to set the desired Ref. Coord. System
9. Select the desired Coordinate system
10. Push **Next>**
11. Enter values for the **Offset U1, Offset U2, Offset U3** fields

Alternatively you can select a grid using **How to Select Grid(s) by picking in the Viewport/ How to Select Grid(s) by Grid ID**

12. Enter a value for the **Offset Scale Factor**
13. Enter a value in the **Radius for non-3D Domains** field for selecting the interior grids which are part of the domain
14. Push **Next>**
15. Select the **Group** for the new domains from the existing groups
16. Push the **Finish** button

How to Create Copies of Domains by Rotation

1. From the **Design** category chooser, select **Shape Domains**
2. Push the **New Domain** button in the Edit Menu Toolbar
3. Select the **Duplicate Selected Domains (use new rotated grids)** radio button
4. Enter a name in the **Name** field
5. Push **Next>**
6. Select the Domain(s) to be copied from the existing set of Domains
7. Push **Next>**
8. Push the **Change** button to set the desired Ref. Coord. System
9. Select the desired Coordinate system
10. Push **Next>**
11. Enter values for the **Center X1, Center X2, Center X3** fields

Alternatively you can select a grid using **How to Select Grid(s) by picking in the Viewport/**
How to Select Grid(s) by Grid ID
12. Select an axis for the **Rotational Axis**
13. Enter a value for the **Angle Increment**
14. Enter a value in the **Radius for non-3D Domains** field for selecting the interior grids which are part of the domain
15. Push **Next>**
16. Select the **Group** for the new domains from the existing groups
17. Push the **Finish** button

How to Create Mirror Copies of Domains

1. From the **Design** category chooser, select **Shape Domains**
2. Push the **New Domain** button in the Edit Menu Toolbar
3. Check the **Duplicate Selected Domains (use new mirrored grids)** radio button
4. Enter a name in the **Name** field
5. Push **Next>**
6. Select the Domain(s) to be mirrored from the existing set of Domains
7. Push **Next>**
8. Push the **Change** button to set the desired Ref. Coord. System

9. Select the desired Coordinate system
10. Push **Next>**
11. Enter values for the **Center X1, Center X2, Center X3** fields
Alternatively you can select a grid using **How to Select Grid(s) by picking in the Viewport/ How to Select Grid(s) by Grid ID**
12. Set the mirror plane by selecting from the category chooser
13. Enter a value in the **Radius for non-3D Domains** field for selecting the interior grids which are part of the domain
14. Push **Next>**
15. Select the **Group** for the new domains from the existing groups
16. Push the **Finish** button

How to Create Multiple Shape Domains with Bias Options

1. From the **Design** category chooser, select **Shape Domains**
2. Push the **New Domain** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select a type of the domain
5. Check the **New Domain Quick Setup** radio button
6. Push **Next>**
7. Select **Create New Domain Group** item
This will create a new domain group
8. Push **Next>**
9. Check the **Drag out size** radio button
You can also pick point by point
10. Check the **Pick Points on Workplane** radio button
11. Select a **Plane of view** icon
12. Select the **Quads** icon as **Region Definition Option**
13. Create a Domain by dragging in the main windows
14. Select the Quad region you just created
15. Push the **Subdivide** button
16. Enter a number of Elements in each Dimension
17. Enter the desired bias value in the **Bias** field

18. Check the desired Bias Type (Center/Edge) radio button

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.

19. Push **Next**>
20. Push the **Finish** button

How to Create a Perturbation associated to the set of Domains

1. Create a Shape Domain using **How to Create a Shape Domain**
2. From the **Design** category chooser, select **Shape Morphing Sets**
3. Push the **New Shape Set** button in the Edit Menu Toolbar
4. Enter a name in the name field
5. Select the **Domain Morphing Set** radio button
6. Create a design variable using **How to Create an Independent Design Variable** or use the created by default one
7. Select the created design variable in the Design Variable menu or use the created by default one
8. Push **Next**>
9. Select the shape domain(s) to be perturbed from the **Select Domain to Act Upon** list.
10. Push **Next**>
11. Push the **Select None** button
12. Select the grid(s) where the perturbation must be applied using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
13. Define the direction of the perturbation
 - by entering numbers in the **X**, **Y** and **Z** direction fields
 - or
 - by pushing the **By 2 Grids...** button, then by selecting 2 grids in the Viewport and by pushing **Next**>
 - or
 - by pushing the **By 3 Grids...** button, then by selecting 3 grids in the Viewport and by pushing **Next**>
14. Enter a **Magnitude** value
15. Push the **Add Perturbation** button
16. Push the **Finish** button



How to Create a Perturbation on individual Grids

1. From the **Design** category chooser, select **Shape Morphing Sets**
2. Push the **New Shape Set** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Raw Morphing Set** radio button
5. Push **Next>**
6. Select the grid(s) where the perturbation must be applied using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
7. Define the direction of the perturbation
by entering numbers in the **X, Y** and **Z** direction fields
or
by pushing the **By 2 Grids...** button, then by selecting 2 grids in the Viewport and by pushing **Next>**
or
by pushing the **By 3 Grids...** button, then by selecting 3 grids in the Viewport and by pushing **Next>**
8. Enter a **Magnitude** value
9. Push the **Add Perturbation** button
10. Push the **Finish** button

How to Create a Perturbation Normal to Surfaces or Edges

1. Create a Shape Domain using **How to Create a Shape Domain**
2. From the **Design** category chooser, select **Shape Morphing Sets**
3. Push the **New Shape Set** button in the Edit Menu Toolbar
4. Enter a name in the name field
5. Select the **Raw Morphing Set** radio button
6. Select a design variable in the Design Variable menu or specify the creation of a new design variable
7. Push **Next>**
8. Push the **Select None** button
9. Select the grid(s) where the perturbation must be applied using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps

10. Select the **Normal to Surfaces /Normal to Edges** option from the category chooser in the **Change perturbations on Selected grids** section
11. Enter a **Magnitude** value
12. Push the **Add Perturbation** button
13. Push the **Finish** button

How to Create a Natural Perturbation from the displacement result

1. From the **Design** category chooser, select **Shape Morphing Sets**
2. Push the **New Shape Set** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Raw Morphing Set** radio button
5. Push **Next>**
6. Select the grid(s) where the perturbation must be applied using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
7. Push the **Convert Deform Results...** button
8. Select a deform results from the list
9. Enter a value in the **Scale Factor** field
10. Push **Next>**
11. Push the **Finish** button

How to Create Raw Freeform Morphing Set Data

1. From the **Design** category chooser, select **Shape Morphing Sets**
2. Push the **New Shape Set** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Raw Morphing Set** radio button
5. Select a design variable in the Design Variable menu or specify the creation of a new design variable
6. For the **Freeform**, check the **Yes** radio button
7. Enter a value for **Maximum Perturbation** to scale the applied perturbations
8. Enter a value for **Sensitivity Scaling** to scale the calculated sensitivities

9. Enter a value for **Initial Randomness** to assign small random values for the shape change in the initial design.
10. Push **Next>**
11. Select the grid(s) where the perturbation must be applied using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
12. Define the direction of the perturbation
by entering numbers in the **X, Y** and **Z** direction fields
or
by pushing the **By 2 Grids...** button, then by selecting 2 grids in the Viewport and by pushing **Next>**
or
by pushing the **By 3 Grids...** button, then by selecting 3 grids in the Viewport and by pushing **Next>**
13. Enter a **Magnitude** value
14. Push the **Add Perturbation** button
15. Push the **Next** button
16. Select the **Coarse method** from the category chooser and enter a value for the **Real Coarse Parameter**
17. Push the **Change** button to set the Symmetry Coord. Sys.
18. Select the desired Coordinate system
19. Push the **Next** button
20. Set the **Symmetry 1, Symmetry 2, Symmetry 3** values by selecting from the category chooser
21. Enter the value for the **Grid Fraction** field
22. Push the **Finish** button

How to Create Domain Freeform Morphing Set Data

1. From the **Design** category chooser, select **Shape Morphing Sets**
2. Push the **New Shape Set** button in the Edit Menu Toolbar
3. Enter a name in the name field
4. Select the **Domain Morphing Set** radio button
5. Select a design variable in the Design Variable menu or specify the creation of a new design variable
6. For the **Freeform**, check the **Yes** radio button

7. Select **Split Only Control Perts.** or **Split All Generated Perts.** option for **Freeform Level**
 - Split Only Control Perts.** option would perform freeform only by splitting the perturbations on the DOMAINS
 - Split All Generated Perts.** option would perform freeform by splitting the all perturbations created by the DOMAINS
8. Enter a value for **Maximum Perturbation** to scale the applied perturbations
9. Enter a value for **Sensitivity Scaling** to scale the calculated sensitivities
10. Enter a value for **Initial Randomness** to assign small random values for the shape change in the initial design.
11. Push **Next>**
12. Select the grid(s) where the perturbation must be applied using **How to Select Grid(s) by Grid ID** or **How to Select Grid(s) by picking in the Viewport** steps
13. Define the direction of the perturbation
 - by entering numbers in the **X, Y** and **Z** direction fields
 - or
 - by pushing the **By 2 Grids...** button, then by selecting 2 grids in the Viewport and by pushing **Next>**
 - or
 - by pushing the **By 3 Grids...** button, then by selecting 3 grids in the Viewport and by pushing **Next>**
14. Enter a **Magnitude** value
15. Push the **Add Perturbation** button
16. Push the **Next** button
17. Select the **Coarse method** from the category chooser and enter a value for the **Real Coarse Parameter**
18. Push the **Change** button to set the Symmetry Coord. Sys.
19. Select the desired Coordinate system
20. Push the **Next** button
21. Set the **Symmetry 1, Symmetry 2, Symmetry 3** values by selecting from the category chooser
22. Enter the value for the **Grid Fraction** field
23. Push the **Finish** button



2.6 How to use the functions of the Post tab

How to Delete some Post-Processing Results

1. From the **Post** tab, push the **Manage Result Datasets** button
2. Select the data-sets to delete
3. Push the **Delete Result Set** button in the Edit Menu Tool bar
4. Push the **Up** button

How to Post-Process Stresses Results

1. Import the Post-Processing file using **How to Import Analysis Post-Processing Files**
2. Select the **Post** tab
3. Push the **Deform/Mesh Color Mesh** button
4. Select a Shell Stress Result for a design cycle
5. Push the **Options...** button. Slide the bar to hide elements with low values
6. Push the **Draw No Element Edges** radio button
7. Push the **Draw All Elements Edges** radio button
8. Push the **Draw Features Edges Only** radio button
9. Push the **Close** button
10. Push the **Up** button

How to Post-Process Displacements Results

1. Import the Post-Processing file using **How to Import Analysis Post-Processing Files**
2. Select the **Post** tab
3. Push the **Deform/Mesh Color Mesh** button
4. Push the **Clear** button in the Color Mesh
5. Select a Displacement Result for a design cycle
6. Push the **Oscillate** button
7. Push the **Filled Contour** button in the **Color Mesh**
8. In the Color Mesh list, select a Displacement Result for a design cycle

9. From the category chooser located at the bottom center of the **Color Mesh** section, select the option: **Normal to Surface**
Also applicable for **Shape Change** Results
10. Right-click in the viewport, select **List Top Ten**
11. Right-click in the viewport, select **List Bottom Ten**
12. Push the **Up** button

How to Post-Process Sizing Results

1. Import the Post-Processing file using **How to Import Design Post-Processing Files**
2. Select the **Post** tab
3. Push the **Deform/Mesh Color Mesh** button
4. Select a Sizing Result for a design cycle
5. Change the **Group Display Style** if needed from the panel below
6. Right-click in the viewport, select **List Top Ten**
The list is printed in the Messages window.
7. Right-click in the viewport, select **List Bottom Ten**
8. Push the **Up** button

How to Check the Perturbations of a Shape Domain Morphing Set

It is recommended to check each perturbation vector after creating them because it helps to prevent bad element deformations.

1. Select the **Post** tab
2. Push the **Deform Mesh/Color Mesh**
3. In the **Deform Mesh** window, select **Ramp** (try also **Static** and **Oscillate**)
4. In the **Color Mesh** window, select **Filled Contours**
5. Select each **Perturbation** from the **Color Mesh** window and study it
6. Push **Up** when finished

How to Post-Process Topology Density Isosurfaces Results

7. Import the `filename_dsgDENSxx.pch` file using **How to Import Design Post-Processing Files**
8. Select the **Post** tab

9. Push the **Density Isosurface** button
10. Select a Topology Result for any design cycle
11. Select a Topology Result for the last design cycle
12. Push the **Options...** button located in the left bottom corner of the **Isosurface Mesh** section of the **Post** Tab
13. Check the **Show Topology Region** check box

Additionally if you do not wish to view those elements with no value in your model, then check the **Hide Elements With No Value** Check Box.

The Isosurface Value slider bar allows you to set different Isosurface values to aid in interpretation of results.

14. Push the **Close** button
15. Push the **Up** button

How to Create a Movie file

1. Select the **Post** tab
2. Push the **Animation** button
3. Choose the **Deform Result Type** and the **Color Result Type** responses to animate from the category chooser.
4. Push **Next>**
5. Select all the results you want to animate
6. Push **Next>**
7. Push the **Options...** button; slide the **Lower Cutoff** slide bar to mask out element with low density values
8. Push the **Close** button
9. Push the **Save Animation...** button
10. Enter a name
11. Push the **Save** button
12. Push the **Up** button

How to Animate Results

1. Select the **Post** tab
2. Push the **Animation** button

3. Choose the **Deform Result Type** and the **Color Result Type** responses to animate from the category chooser.
4. Push **Next>**
5. Select all the results you want to animate
6. Push **Next>**
7. Push the **Options...** button; slide the **Lower Cutoff** slide bar to mask out element with low density values
8. Change the **Group Display Style** if needed from the panel below
9. Push the **Close** button
10. Push the **Up** button

How to Post-Process the Histories

1. Import the History file using **How to Import the History File**
2. Select the **Post** tab
3. Push the **Design History Plots** button
4. Select the Design History `filename_dsg.HIS`
5. Push the **New Plot** button
6. Select all quantities to plot
7. Push the **Finish** button
8. Push the **Up** button

How to Create a Frequency Responses Plot

1. Select the **Post** tab
2. Push the **Freq. Resp. Plot** button
3. Push the **New Freq. Resp. Plot** button in the Edit Menu Toolbar
4. Push the **Magnitude + Phase** radio button
5. Push **Next>**
6. Push the **+** button
7. Choose the cycle(s) to be plotted
8. Push **Next>**
9. Choose the element(s) to plot



10. Push **Next>**
11. Push the **Finish** button
12. Push the **Up** button

How to Save a Frequency Responses Plot

1. Create a frequency responses plot using **How to Create a Frequency Responses Plot**
2. Right click on the graph
3. Push the **Save Image...** button
4. Enter a file name
5. Push the **Save** button
6. Push the **Finish** button
7. Push the **Up** button

How to Find Values for Results

1. Select the **Post** tab
2. Select an entity on the structure

In the **Design Studio Messages** windows you will be able to see the value selected.

How to Animate Sequences of Isosurface Plots

1. Select the **Post** tab
2. Push the **Animation** button
3. Set the **Deform Result Type** Value remain as **None**
4. Choose the **Color Result Type** to be **Topology Result**
5. Check the **Isosurface** radio button
6. Push **Next>**
7. Select all the results you want to animate
8. Push **Next>**
9. Push the **Options...** button
10. Check the **Show Topology Region** check box
11. Push the **Close** button

12. Push the **Up** button

Display style Options for Color Mesh and Isosurface

1. Import the Post-Processing file using **How to Import Analysis Post-Processing Files**
2. Select the **Post** tab
3. Push the **Deform/Mesh Color Mesh** button
4. Select a Result for a design cycle
5. 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 tooltip will reveal the names/function of the buttons.

Alternatively this can be done by repeating steps 5 under **Group Display Style** section in **Display** tab

Also applicable to results in **Isosurface Mesh** section in the **Density Isosurface** option under **Post** tab

6. Push the **Up** button

2.7 How to use the 'Find' Filter

How to do a Basic Search

You can use this option in most of the features in Design Studio. The table below shows the various features in Design Studio along with the availability of the find filter:

Tab	Feature	Available / Not Available
Display	Show / Hide Groups	Available
Display	Show / Hide Elements	Not Available
Display	Identify Grids	Not Available
Display	Identify Elements	Not Available
Display	Manage Groups	Available
Display	Manage Assemblies	Available
Display	Manage Color Palette	Available
Display	Set View Coord. Sys	Available
Analysis	Grid-Component Sets	Available
Analysis	Element Sets	Available
Analysis	Grid Sets	Available
Analysis	Static Loads	Available
Analysis	Eigenvalue Methods	Available
Analysis	Loadcases	Available
Analysis	Coordinate Systems	Available
Analysis	Materials	Available
Analysis	Composite Failure Equations	Available
Analysis	Group properties	Available
Analysis	Elements	Not Available
Analysis	Grids	Not Available
Topology	Quick Setup Trails	Not Available
Topology	Topology Regions	Available
Topology	Topology Objectives	Available
Topology	Topology Constraints	Available

Design	Quick Setup Trails	Not Available
Design	Synthetic Responses	Available
Design	Objectives	Available
Design	Constraints	Available
Design	Design Variables	Available
Design	Sizing	Available
Design	Topometry	Available
Design	Topography	Available
Design	Shape Domains	Available
Design	Shape Morphing Sets	Available
Post	Manage Result Datasets	Available
Post	Deform Mesh / Color Mesh	Available
Post	Density Isosurface	Available
Post	Animation	Not Available for the first step
Post	Design History Plots	Available
Post	Freq. Resp. Plot	Not Available
Post	Sum ESE Results	Available

1. In any of the tabs, select a **feature**
2. Enter the text to be searched (a word or a number) in the **Find** option field

If you want to search a string which contains a space, you need to use “”. For example you can type: “Left Part”

3. Press the enter key

How to Clear a Search

1. In any of the tabs, select a **feature**
2. Push the **Clear Find** button

The **Clear Find** button is the cross at the right of the Find option field

You can also delete the word or the number using the backspace key and press the enter key

How to Use Boolean Operators

1. In any of the tabs, select a **feature**

2. Enter a sequence of words and/or numbers separated by Boolean operator

For 'and' you can enter: word1 and word2. You can also enter: word1 word2.

For 'or' you can enter: word1 or word2

For 'not' you can enter: !word1

You can also combine multiple operators: !word1 and number1 or word2

('and' has the priority on 'or')

3. Press the enter key

How to search based on keywords

1. In any of the tabs, select a **feature**
2. Enter an appropriate keyword format as shown in the table below:

Keyword Formats	Description
<code>desc:string</code>	To search in the description
<code>name:string</code>	To search in the name
<code>id:number</code>	To search in the id
<code>design:</code>	To search the properties which are designed
<code>sizing:</code>	To search the properties which are designed using sizing
<code>topology:</code>	To search the properties which are designed using topology
<code>topometry:</code>	To search the properties which are designed using topometry
<code>topography:</code>	To search the properties which are designed using topography
<code>mat:string</code>	To search in the material name
<code>matid:number</code>	To search in the material id

Additional criteria can be added using any of the above keyword formats by including a space.

Example: If you are looking for all the PSHELL which are designed you can type:

Pshell and design:

For the same results you can also enter:

design: Pshell

3. Press the enter key

How to use 'Find' in Show / Hide Groups

1. From the **Display** tab, push the **Show/Hide Groups** button
2. Enter search criteria in the find field

For example if you are looking for the PCOMP which are not using the material id=1 you can enter:

PCOMP and !matid=1

3. Press the enter key

How to use 'Find' in Loadcases

1. From the **Analysis** tab, select **Loadcases**
2. Enter search criteria in the find field

For example if you are looking for the Loadcases which do not have an Id=5 you can enter:

`!id:5`

For example if you are looking for the Loadcases which have an Id between 5 and 10 you can enter:

`id:5-10`

3. Press the enter key

How to use 'Find' in Topology Regions

1. From the **Topology** tab, select **Topology Regions**
2. Enter search criteria in the find field

For example if you are looking for the Topology Regions which do not have an Id=5 you can enter:

`!id:5`

For example if you are looking for the Topology Regions which have an Id between 5 and 10 you can enter:

`id:5-10`

3. Press the enter key

How to use 'Find' in Constraints

1. From the **Design** tab, select **Constraints**
2. Enter search criteria in the find field

For example if you are looking for the Constraints which have an upper bound = 0.5 you can enter:

`UB=0.5`

3. Press the enter key

How to use 'Find' in Manage Result Datasets

1. From the **Post** tab, select **Manage Result Datasets**
2. Enter search criteria in the find field

For example if you are looking for the results of the Loadcases 101 during the Cycle 2 you can

enter:

`"Cycle 2" "Loadcases 101"`

For the same results you can also enter:

`"Cycle 2" and "Loadcases 101"`

If you want to search a string which contains a space, you need to use `"`.

3. Press the enter key

How to use 'Find' in Deform Mesh / Color Mesh

1. From the **Post** tab, select **Deform Mesh / Color Mesh**
2. Enter search criteria in the find field

For example if you are looking for the results of the Cycle 2 and the Cycle 10 you can enter:

`"Cycle 2" or "Cycle 10"`

If you want to search a string which contains a space, you need to use `"`.

3. Press the enter key

CHAPTER 3

How to enter Genesis Analysis Parameters (PARAM) and Design Optimization Parameters (DOPT) in Design Studio

- Analysis Parameters: PARAM Bulk Data Statement
- Optimization Parameters for Topology: DOPT Bulk Data Statement
- Optimization Parameters for Shape, Sizing, Topometry and Topography: DOPT Bulk Data Statement
- Optimization Parameters For Discrete Design Variables: DOPT Bulk data Statement

3.1 Analysis Parameters: PARAM Bulk Data Statement

How to Change AUTOSPC

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Check **Automatic SPC (AUTOSPC)**
4. From the category chooser select the value

If AUTOSPC = YES, then perform automatic constraining of degrees of freedom with little or no stiffness.
If AUTOSPC = NO, then automatic constraining of degrees of freedom is not performed.
5. Push the **Apply** button

How to Change BAILOUT

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Check **Stop on Singularity (BAILOUT)**
4. From the category chooser select the value

If BAILOUT = YES, then stop if the factor ratio of the triangularized matrix is greater than MAXRATIO.
If BAILOUT = NO, the program will solve the problem with singularities.
5. Push the **Apply** button

How to Change EOF

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Check **Element-on-the-Fly (EOF)**
4. From the category chooser select the value

If EOF = YES, finite element matrices are calculated as they are needed, rather than stored. This option may reduce elapsed run time on computers with fast CPU's. This parameter can only be used if the analysis parameter SOLVER=1. If SOLVER=2, then EOF=NO.
EOF = NO, means finite element matrices are computed and stored for future use.
5. Push the **Apply** button

How to Change Structural Damping (G)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Check **Structural Damping (G)**
4. Enter the value in the **Structural Damping** field

G is the structural damping coefficient applied to the global stiffness matrix.
5. Push the **Apply** button

How to Change RESVEC

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Check **Residual Vectors (RESVEC)**
4. From the category chooser select the value

If RESVEC = YES, then residual vectors are used with the normal mode shapes to increase the accuracy of modal dynamic analysis.
To not use residual vectors set RESVEC = NO.
5. Push the **Apply** button

How to Change THREADS

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Check **Processors (THREADS)**
4. Enter the value in the **Processors** field

THREADS selects number of parallel threads.
5. Push the **Apply** button

How to Change COUPMASS

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Check **Consistent Mass (COUPMASS)**
4. From the category chooser select the value

A coupled mass matrix is used if COUPMASS = YES.
A lumped (diagonal) mass matrix is used if COUPMASS = NO.
A consistent mass matrix is used if COUPMASS = FULL.

5. Push the **Apply** button

How to Change WTMASS

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Check **Mass Scaling (WTMASS)**
4. Enter the value in the **Mass Scaling** field

The conversion factor from weight units to mass units.
If WTMASS is not 1.0, then all mass-related entries in the model (e.g., density) are assumed to be entered in weight units, and are scaled by WTMASS to get mass values.
5. Push the **Apply** button

How to Change EPZERO

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **EPZERO**
5. Enter the value in the **EPZERO** field

The AUTOSPC threshold value. Degrees of freedom with less than EPZERO stiffness will be constrained.
6. Push the **Close** button
7. Push the **Apply** button

How to Change IRTOL

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **IRTOL**
5. Enter the value in the **IRTOL** field

The maximum energy error ratio. Used to determine sufficient support on inertia relief load-cases.
6. Push the **Close** button
7. Push the **Apply** button

How to Change KDAMP

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **KDAMP**
5. Enter the value in the **KDAMP** field

When $KDAMP = 1$, the modal damping is added to the viscous damping matrix. When $KDAMP = -1$, the modal damping is added to the complex stiffness matrix.
6. Push the **Close** button
7. Push the **Apply** button

How to Change MIDSIDE

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **MIDSIDE**
5. Enter the value in the **MIDSIDE** field

If $MIDSIDE=1$, then each midside node of all second order elements is moved to the physical midpoint of its edge.

If $MIDSIDE=2$, then each midside node of all second order elements is moved to the physical midpoint of its edge and perturbations applied to corner nodes of elements are averaged and applied to the corresponding midside nodes.

If $MIDSIDE=0$, then no changes are made to midside nodes.
6. Push the **Close** button
7. Push the **Apply** button

How to Change SHAPECK

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **SHAPECK**
5. Enter the value in the **SHAPECK** field

Controls the checking of the shape of regular 2D and 3D finite elements and of the DOMAIN

elements.

SHAPECK=0 Element shapes are not checked (replacement of diag=192)

SHAPECK=1 Count shape distortion errors as non-fatal. Errors are printed, but warnings are not. (replacement of diag=194)

SHAPECK=2 Count shape distortion errors as non-fatal. Error and warning messages are printed. (replacement of diag=191)

SHAPECK=3 Do not print shape distortion warning messages. (replacement of diag=195)

SHAPECK=4 Perform normal checking. This is the default.

SHAPECK=5 Perform normal checking and print element shape characteristics. (replacement of diag=193)

6. Push the **Close** button
7. Push the **Apply** button

How to Change SOLVER=1

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **SOLVER**
5. Enter the value in the **SOLVER** field

Determines the type of linear equation solver to use.

If SOLVER = 1, use the sparse matrix solver.

If SOLVER = 2, use the skyline solver.

SOLVER must be 1 if the Lanczos method or the SMS method is used for eigenvalue calculations.

6. Push the **Close** button
7. Push the **Apply** button

How to Request REDUCE Mode

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **REDUCE Mode**

In **REDUCE** mode, Genesis will only reduce matrices and loads to a specified boundary set of degrees of freedom

5. Push the **Close** button
6. Push the **Apply** button

How to Change FINDEXCK

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **FINDEXCK**
5. From the category chooser select the value

When FINDEXCK = YES, the program checks if the material properties needed for failure index calculations are available or not. If they are not available and FINDEX is requested then Genesis will stop with an error message.

If FINDEXCK = NO, the program will not stop for the above reason but will turn off the FINDEX calculation request.

6. Push the **Close** button
7. Push the **Apply** button

How to Change ITMXSS

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **ITMXSS**
5. Enter the value in the **ITMXSS** field

ITMXSS is the maximum number of subspace iteration cycles allowed.

6. Push the **Close** button
7. Push the **Apply** button

How to Change MAXRATIO

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **MAXRATIO**
5. Enter the value in the **MAXRATIO** field

The maximum factor ratio is used to determine singularities in the stiffness matrix during decomposition.

6. Push the **Close** button
7. Push the **Apply** button

How to Change PLOADM

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **PLOADM**
5. Enter the value in the **PLOADM** field

Controls the way vector pressures are treated. A value of 1, the default, will use the full vector. A value of 0 will use the normal component only. This parameter affects PLOAD4 and PLOAD5.

6. Push the **Close** button
7. Push the **Apply** button

How to Change SMSMAX

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **SMSMAX**
5. Enter the value in the **SMSMAX** field

Maximum number of degrees of freedom in a supernode in SMS. Influences the incore memory requirements of SMS.

6. Push the **Close** button
7. Push the **Apply** button

How to Change SPCFTOL

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check **SPCFTOL**

5. Enter the value in the **SPCF'TOL** field
SPCFORCE filter parameter. Reaction forces with a norm less than SPCFTOL will not be printed in the output file.
6. Push the **Close** button
7. Push the **Apply** button

How To Change SEMPC

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check the **SEMP'C** check box
5. Enter the value in the **SEMP'C** field
SEMP'C = 0 will not create MPC's to recover displacements of eliminated degrees of freedom in the superelement reduction
SEMP'C = integer will create MPC's, with the defined integer as set ID, to recover the eliminated translational degrees of freedom that are requested as outputs
6. Push the **Close** button
7. Push the **Apply** button

How To Change PRGPST

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check the **PRGPST** check box
5. From the category chooser, select the value
PRGPST = Yes will print the AUTOSPC table
PRGPST = No will not print the AUTOSPC table
6. Push the **Close** button
7. Push the **Apply** button

How to Change MSMOOTH

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab

3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **Mesh Smoothing**
6. From the category chooser, select the value
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.
7. Push the **Close** button
8. Push the **Apply** button

How To Change RANDOM

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check the **RANDOM** check box
5. Enter an integer value in the **RANDOM** field
The integer value controls the random seed used for the random number generator in genesis.
6. Push the **Close** button
7. Push the **Apply** button

How To Change RBE3SPC

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Analysis Control** tab
3. Push the **Advanced...** button
4. Check the **RBE3SPC** check box
5. From the category chooser, select the value
RBE3SPC = Yes will automatically constraint degrees of freedom of any independent grids in the RBE3 if the grid is not attached to any other elements
RBE3SPC = No will not constraint degrees of freedom of any independent grids in the RBE3 if the grid is not attached to any other elements
6. Push the **Close** button
7. Push the **Apply** button

3.2 Analysis Parameters not listed in Design Studio

The following analysis parameters can NOT be activated or modified in Design Studio:

CK2, CM2, CP2, EPSEIG, GRDPNT, INREL, LIMITLSF, LOADCK, MODSNS, MODTRK, OPPTHK, OPPTHO, PCH2PST, PRTMAXIM, PRTRESLT, PSMOOTH, SHELLCK, T3SRM, T6TOT3, TAPELBL, THETA.

However, these parameters can be read in and written out by Design Studio if you modify them in the input data file.

3.3 Optimization Parameters for Topology: DOPT Bulk Data Statement

How to Change the Number of Maximum Design Cycle (DESMAX) (for topology)

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Check **Maximum Design Cycles**
4. Enter an integer value in the **Maximum Design Cycles** field
5. Push the **Apply** button

How to Change OPTM (for topology)

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Check **Optimizer**
4. From the category chooser, select the value
 Switch to select optimizer. If **Optimizer= DOT**, Genesis will use the DOT optimizer. If **Optimizer = BIGDOT**, Genesis will use the BIGDOT optimizer. (Default=DOT for Shape and Sizing and BIGDOT for Topology).
5. Push the **Apply** button

How to Change TINDEXM (for topology)

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Methods** tab
5. Check **Topology Index**
6. From the category chooser, select the option
 TINDEXM is the compliance index formulation.
 If **Normalized Reciprocal** is selected then Genesis will use the reciprocal contribution for responses with negative weighting factors. Responses are normalized.
 If **Normalized Direct** is selected then Genesis will use the direct contribution for responses

with negative weighting factors. Responses are normalized

If **Reciprocal** is selected then Genesis will use the reciprocal contribution for responses with negative weighting factors.

If **Direct** is selected then Genesis will use the direct contribution for responses with negative weighting factors.

7. Push the **Close** button
8. Push the **Apply** button

How to Change METHOD (for topology)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Methods** tab
5. Check **DOT Method**
6. From the category chooser, select a method
 - Optimization method to be used by DOT.
 - If **MMFD** is selected, the modified method of feasible directions is used.
 - If **SLP** is selected, then sequential linear programming is used.
 - If **SQP** is selected, then sequential quadratic programming is used.
7. Push the **Close** button
8. Push the **Apply** button

How to Change CONV1 (for topology)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Hard Relative (CONV1)**
6. Enter the value in the **Hard Relative (CONV1)** field
 - CONV1 is the relative change criterion to detect hard convergence of the overall optimization process.
7. Push the **Close** button
8. Push the **Apply** button

How to Change CONV2 (for topology)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Hard Absolute (CONV2)**
6. Enter the value in the **Hard Absolute (CONV2)** field
7. Push the **Close** button
8. Push the **Apply** button

CONV2 is the absolute change criterion to detect hard convergence of the overall optimization process.

How to Change GMAX (for topology)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Hard Max. Violation (GMAX)**
6. Enter the value in the **Hard Max. Violation (GMAX)** field
7. Push the **Close** button
8. Push the **Apply** button

Maximum constraint violation allowed at the optimum. Constraints are normalized so a value of 0.01 represents a one percent constraint violation, which is normally considered acceptable.

How to Change CONVCN (for topology)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Soft Constraint (CONVCN)**

6. Enter the value in the **Soft Constraint (CONVCN)** field

Allowable change in the maximum constraint value for soft convergence. If the change in the maximum constraint value is less than CONVCN, and CONVDV is satisfied, then terminate the design process with soft convergence.

7. Push the **Close** button
8. Push the **Apply** button

How to Change CONVDV (for topology)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Soft Variable (CONVDV)**
6. Enter the value in the **Soft Variable (CONVDV)** field

Relative change criterion to detect soft convergence of the overall optimization process. Terminate with soft convergence if the maximum relative change in the design variables is less than CONVDV during the approximate optimization and CONVCN is satisfied.

7. Push the **Close** button
8. Push the **Apply** button

How to Change DELT

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Move Limits** tab
5. Check **Fractional Topology (DELT)**
6. Enter the value in the **Fractional Variable (DELT)** field

Fractional change allowed for each designable topology variable during the approximate optimization. This provides move limits.

7. Push the **Close** button
8. Push the **Apply** button

How to Change DTMIN

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Move Limits** tab
5. Check **Minimum Topology (DTMIN)**
6. Enter the value in the **Minimum Topology (DTMIN)** field
Minimum move limit fraction imposed for topology design variables.
7. Push the **Close** button
8. Push the **Apply** button

How to Change ITMAX (for topology)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **Max DOT Iterations/Cycle (ITMAX)**
6. Enter the value in the **Max DOT Iterations/Cycle (ITMAX)** field
Maximum number of iterations in the approximate optimization.
7. Push the **Close** button
8. Push the **Apply** button

How to Change DNSHIS

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check ***.DNS File (DNSHIS)**
6. From the category chooser, select the option
The “**Create**” option will cause the program to create and print the density *.DNS” file.
The “**Do Not Create**” option will cause the program to not print the density *.DNS” file.
7. Push the **Close** button

8. Push the **Apply** button

How to Change ITRMOP (for topology)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **DOT Conv. Iterations (ITRMOP)**
6. Enter the value in the **DOT Conv. Iterations (ITRMOP)** field

The number of consecutive iterations that DOT or BIGDOT must satisfy the absolute or relative convergence criteria before the optimization is terminated. Usually ITRMOP should be at least 2 because it is common to make little progress on one iteration, only to make major progress on the next. Therefore, ITRMOP = 2 will allow a second try before terminating. If progress toward the optimum seems slow, but consistent, and function evaluations are not too expensive, increasing ITRMOP to a value of 3 to 10 may improve the solution.

7. Push the **Close** button
8. Push the **Apply** button

How to Change FILTER

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **Anticheckerboard Filter**
6. From the category chooser, select the option

Anti-checkerboard filtering ON (FILTER = 1) or OFF (FILTER = 0) parameter.

7. Push the **Close** button
8. Push the **Apply** button

How to Change IPRINT (for topology)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button

4. Select the **Misc.** tab
5. Check **DOT Debug Output**
6. From the category chooser, select the option

Print control for optimizer output during approximate optimization. Larger value gives more print. IPRINT=0 gives no print (select **None**).
To have IPRINT=1, from the category chooser, select **Minimal**
To have IPRINT=4, from the category chooser, select **More**
To have IPRINT=7, from the category chooser, select **All**
7. Push the **Close** button
8. Push the **Apply** button

3.4 Optimization Parameters not listed in Design Studio (Topology)

The following optimization parameters for topology can NOT be activated or modified in Design Studio:

TMIN, DABOBJ, DELOBJ, BDMEM.

However, these parameters can be read in and written out by Design Studio if you modify them in the input data file.

3.5 Optimization Parameters for Shape, Sizing, Topometry and Topography: DOPT Bulk Data Statement

How to Change the Number of Maximum Design Cycle (DESMAX)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Check **Maximum Design Cycles**
4. Enter an integer value in the **Maximum Design Cycles** field
5. Push the **Apply** button

How to Change OPTM

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Check **Optimizer**
4. From the category chooser, select the value

Switch to select optimizer. If **OPTM=DOT**, Genesis will use the DOT optimizer. If **OPTM=BIGDOT**, Genesis will use the BIGDOT optimizer. (Default=DOT for Shape and Sizing and BIGDOT for Topology).
5. Push the **Apply** button

How to Change DINDEXM

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Methods** tab
5. Check **Multi-objectives Index**
6. From the category chooser, select the value

Design multi-objective index formulation.
If **Normalized Reciprocal** is selected then Genesis will use the reciprocal contribution for responses with negative weighting factors. Responses are normalized.
If **Normalized Direct** is selected then Genesis will use the direct contribution for responses with negative weighting factors. Responses are normalized
If **Reciprocal** is selected then Genesis will use the reciprocal contribution for responses with

negative weighting factors.

If **Direct** is selected then Genesis will use the direct contribution for responses with negative weighting factors.

7. Push the **Close** button
8. Push the **Apply** button

How to Change IMATCH

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Methods** tab
5. Check **Matching Method**
6. From the category chooser, select the value

This parameter is used when DMATCH and DMATCH2 are used to match analysis results to specific target values.

If **Matching Method** = **Least Squares**, then the least square method is used and if **Matching Method** = **Beta**, the Beta method is used.

7. Push the **Close** button
8. Push the **Apply** button

How to Change ICOMPAPP

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Methods** tab
5. Check **Composite Approximation**
6. From the category chooser, select the value

Switch for composite element approximation. If ICOMPAPP = **More Complex**, the program will use special approximations that can use a large amount of disk space in some cases.

If ICOMPAPP = **Faster**, the program will use direct approximations and usually will use less disk space.

7. Push the **Close** button
8. Push the **Apply** button

How to Change RCOMPAPP

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Methods** tab
5. Check **Composite Force Appr. (RCOMPAPP)**
6. From the category chooser, select the value

Switch for composite element forces approximations. If RCOMPAPP = **More Accurate**, the program will use the standard force approximations. If RCOMPAPP = **Faster**, the program will use a special fast approximation. The default is **More Accurate** except when the STRDOT optimizer is used in which case the default is **Faster**. The standard force approximation is more accurate but uses more computation resources (time/disk and memory).

7. Push the **Close** button
8. Push the **Apply** button

How to Change ISHLAP

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Methods** tab
5. Check **Shell Approximation**
6. From the category chooser, select the value

Switch for shell element approximation and DVPROP3. If ISHLAP = **More Complex**, the program will use 2 intermediate design variable per PSHELL+DVPROP3 property. If ISHLAP = **Faster**, the default for topometry optimization, the program will use direct approximations and usually will use half the disk space for the sensitivity matrix.

7. Push the **Close** button
8. Push the **Apply** button

How to Change RSHLAP

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button

4. Select the **Methods** tab
5. Check **Shell Force Appr. (RSHLAP)**
6. From the category chooser, select the value

Switch for shell element forces approximations. If RSHLAP = **More Accurate**, the program will use the standard force approximations. If RSHLAP = **Faster**, the program will use a special fast approximation. The default is **More Accurate** except when the STRDOT optimizer is used in which case the default is **Faster**. The standard force approximation is more accurate but uses more computation resources (time/disk and memory).

7. Push the **Close** button
8. Push the **Apply** button

How to Change MODAPP

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Methods** tab
5. Check **MDISP,MVELO,MACCE Approx.:**
6. From the category chooser, select the value

Controls the type of approximation used in calculating the MDISP, MVELO and MACCE response types of DRESP1. If MODAPP = **Faster**, the default, the program will use direct approximations.

If MODAPP = **More Complex**, the program will use a special approximation better suited to capturing resonance. For most problems, the direct method is much faster and requires less memory.

7. Push the **Close** button
8. Push the **Apply** button

How to Change ADJOINT

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Methods** tab
5. Check **Sensitivity Method**
6. From the category chooser, select the value

Switch to select the sensitivity calculation method. If ADJOINT= **Adjoint**, Genesis will use

the adjoint method for response sensitivities.

If ADJOINT = **Direct**, Genesis will use the direct method. The adjoint method is generally faster for problems with many design variables and few constraints.

If ADJOINT = **Automatic** (default), Genesis will automatically select direct or adjoint.

7. Push the **Close** button
8. Push the **Apply** button

How to Change CONV1

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Hard Relative (CONV1)**
6. Enter the value in the **Hard Relative (CONV1)** field

Relative change criterion to detect hard convergence of the overall optimization process. Terminate if the relative change in the objective function is less than CONV1 for two consecutive design cycles and all constraints are satisfied within a tolerance of GMAX.

7. Push the **Close** button
8. Push the **Apply** button

How to Change CONV2

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Hard Absolute (CONV2)**
6. Enter the value in the **Hard Absolute (CONV2)** field

Absolute change criterion to detect hard convergence of the overall optimization process. Terminate if the absolute change in the objective function is less than CONV2 for two consecutive design cycles and all constraints are satisfied within a tolerance of GMAX.

7. Push the **Close** button
8. Push the **Apply** button

How to Change GMAX

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Hard Max. Violation (GMAX)**
6. Enter the value in the **Hard Max. Violation (GMAX)** field

Maximum constraint violation allowed at the optimum. Constraints are normalized so a value of 0.01 represents a one percent constraint violation, which is normally considered acceptable.
7. Push the **Close** button
8. Push the **Apply** button

How to Change CONVCN

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Soft Constraint (CONVCN)**
6. Enter the value in the **Soft Constraint (CONVCN)** field

Allowable change in the maximum constraint value for soft convergence. If the change in the maximum constraint value is less than CONVCN, and CONVDV, CONVPR and CONVLC are satisfied, then terminate the design process with soft convergence.
7. Push the **Close** button
8. Push the **Apply** button

How to Change CONVLC

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Soft Grid (CONVLC)**
6. Enter the value in the **Soft Grid (CONVLC)** field

Maximum allowable grid location movement for soft convergence. Terminate with soft convergence if the maximum grid location movement is less than CONVLC, and CONVDV,

CONVPR and CONVCN are satisfied.

7. Push the **Close** button
8. Push the **Apply** button

How to Change CONVDV

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Soft Variable (CONVDV)**
6. Enter the value in the **Soft Variable (CONVDV)** field

Relative change criterion to detect soft convergence of the overall optimization process. Terminate with soft convergence if the maximum relative change in the design variables is less than CONVDV during the approximate optimization and CONVPR, CONVCN, and CON-VLC are satisfied.

7. Push the **Close** button
8. Push the **Apply** button

How to Change CONVPR

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Convergence** tab
5. Check **Soft Property (CONVPR)**
6. Enter the value in the **Soft Property (CONVPR)** field

Relative change criterion to detect soft convergence of the overall optimization process. Terminate if the maximum relative change in the properties is less than CONVPR during the approximate optimization and CONVDV, CONVCN, and CONVLIC are satisfied.

7. Push the **Close** button
8. Push the **Apply** button

How to Change DELP

1. From the main menu bar, select **Genesis → Options...**

2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Move Limits** tab
5. Check **Fractional Property (DELP)**
6. Enter the value in the **Fractional Property (DELP)** field
Fractional change allowed for each designed property during the approximate optimization.
This provides move limits.
7. Push the **Close** button
8. Push the **Apply** button

How to Change DELX

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Move Limits** tab
5. Check **Fractional Variable (DELX)**
6. Enter the value in the **Fractional Variable (DELX)** field
Fractional change allowed for each design variable during the approximate optimization. This provides move limits.
7. Push the **Close** button
8. Push the **Apply** button

How to Change DXMIN

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Move Limits** tab
5. Check **Minimum Variable (DXMIN)**
6. Enter the value in the **Minimum Variable (DXMIN)** field
Minimum move limit factor imposed for design variables.
7. Push the **Close** button
8. Push the **Apply** button



How to Change DPMIN

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Move Limits** tab
5. Check **Minimum Property (DPMIN)**
6. Enter the value in the **Minimum Property (DPMIN)** field
Minimum move limit factor imposed for properties.
7. Push the **Close** button
8. Push the **Apply** button

How to Change ITMAX

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **Max DOT Iterations/Cycle (ITMAX)**
6. Enter the value in the **Max DOT Iterations/Cycle (ITMAX)** field
Maximum number of iterations in the approximate optimization.
7. Push the **Close** button
8. Push the **Apply** button

How to Change OPTHIS

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check ***.OPT File (OPTHIS)**
6. From the category chooser, select the value

If **Create** is selected, this will cause the program to create and print the “*.OPT” file.
If **Do Not Create** is selected this will cause the program to not print the “*.OPT” file.

7. Push the **Close** button
8. Push the **Apply** button

How to Change ITRMOP

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **DOT Conv. Iterations (ITRMOP)**
6. Enter the value in the **DOT Conv. Iterations (ITRMOP)** field

The number of consecutive iterations that DOT must satisfy the absolute or relative convergence criteria before the optimization is terminated. Usually ITRMOP should be at least 2 because it is common to make little progress on one iteration, only to make major progress on the next. The default ITRMOP = 2 will allow a second try before terminating. If progress toward the optimum seems slow, but consistent, and function evaluations are not too expensive, it may improve the solution to increase ITRMOP to a value of 3 to 10.

7. Push the **Close** button
8. Push the **Apply** button

How to Change IPRINT

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **DOT Debug Output**
6. From the category chooser, select the value

Print control for optimizer output during approximate optimization. Larger value gives more print. IPRINT=0 gives no print.(select **None**)

To have IPRINT=1, from the category chooser, select **Minimal**

To have IPRINT=4, from the category chooser, select **More**

To have IPRINT=7, from the category chooser, select **All**

7. Push the **Close** button
8. Push the **Apply** button

How To Change Shape Control (SHAPECN)

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **Shape Control (SHAPECN)** check box
6. From the category chooser, select the value

When **SHAPECN** = **Prevent Negative Jacobian**, the program restricts the shape changes so that a negative jacobian is prevented

When **SHAPECN** = **Prevent DISTOR Errors**, the program restricts the shape changes so that the errors associated with distortion of elements is prevented

When **SHAPECN** = **Prevent DISTOR Warnings**, the program restricts the shape changes so that the errors as well as warnings associated with distortion of elements is prevented

When **SHAPECN** = **No Shape Control**, the program does not put any restrictions on the shape changes resulting from the optimization process

7. Push the **Close** button
8. Push the **Apply** button

How to Change DVGTOL

1. From the main menu bar, select **Genesis** → **Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **DVGRIDC Tol. (DVGTOL)**
6. Enter the value in the **DVGRIDC Tol. (DVGTOL)** field

Parameter used to check whether a grid in a DOMAIN referenced by a perturbation data (DVGRIDC data) is close or not to the midside of the DOMAIN.

7. Push the **Close** button
8. Push the **Apply** button

3.6 Optimization Parameters not listed in Design Studio (Shape, Sizing, Topometry and Topography)

The following optimization parameters can NOT be activated or modified in Design Studio:

BASIS, BDMEM, DR1MV, DVINIT, DVTOL, DXFRAC, FDCHMU, FDCMU, ISRMET, ISRMAX, ISDMAX, IUGRAD, LAMASMS, OPTGRID, PTOL, SGENEL, SK2UU, SM2UU, RMATCH, STRDOT, RPERT1, RPERT2

However, these parameters can be read in and written out by Design Studio if you modify them in the input data file.

3.7 Optimization Parameters For Discrete Design Variables: DOPT Bulk data Statement

How to Change DSTART

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **Discrete Option (DSTART)**
6. Enter the value in the **Discrete Option (DSTART)** field

Parameter to control when discrete optimization starts. A value of -2 will make Genesis ignore discrete variable sets (all DVARs will be continuous). A value of -1 will make Genesis start the discrete optimization process after the continuous optimization process has converged. A value of 0 will make Genesis start the discrete optimization process from the design cycle 0. A value of $N > 0$ will make Genesis start the discrete optimization process after N continuous optimization cycles have been completed. If the continuous optimization process finishes in M design cycles with M less than N , then the discrete optimization process is started in the $M+1$ design cycle.

7. Push the **Close** button
8. Push the **Apply** button

How to Change DVINIT2

1. From the main menu bar, select **Genesis → Options...**
2. Select the **Design Control** tab
3. Push the **Advanced...** button
4. Select the **Misc.** tab
5. Check **Discrete Option (DVINIT2)**
6. From the category chooser, select the value

Parameter used to control the starting design variable values.

If **Stop if not Discrete** is selected, the program to stop if the initial value of the design variable (INIT) does not belong to the associated discrete set.

If **Leave as is** is selected, the program to use the initial value of the design variable (INIT) even if it does not belong to the provided discrete set.

If **Reset to Closest** is selected, Genesis to replace the design variable initial value with the value from the discrete set closest to the provided value.

If **Reset and Freeze** is selected, Genesis to replace the design variable initial value with the

value from the discrete set closest to the provided value and then hold discrete design variables as constants during the optimization. This option allows restart of a problem that has both discrete and continuous variables to keep improving the design using only the continuous variables. This parameter is ignored if DSTART=-2.

7. Push the **Close** button
8. Push the **Apply** button

3.8 Optimization Parameters not listed in Design Studio (Discrete Design Variables)

The following optimization parameters can NOT be activated or modified in Design Studio:

DDAMIN, DDCMIN, DDLMIN, DDPMIN, DDELA, DDELC, DDELL, DDELP, DSCDOT, IPEN, NDSCRT, PENLTD, PMULTD.

However, these parameters can be read in and written out by Design Studio if you modify them in the input data file.