Intro to OptiStruct for Composites

EDU Training Session 2020

Greg Delbridge

Starting at 3:35 EDT

Download the class package from the link in the chat

What Is a Composite Material?

A composite material is one in which at least two distinct materials with significantly different material characteristics are joined to act as a single material

Composite materials come in a variety of types, including:

- Particulate Composites (Particles + Matrix)
- Laminated Composites (Layers)
- Fibrous-Matrix Laminated Composites (Layers – "Long fiber + Matrix")
- Core Stiffened Laminated Composites

This training will focus on the *<u>Fibrous-Matrix Laminated Composites</u>*, which are the most commonly used for high performance structural components

Advantages of Composite Design

Why use composites for creating structural components?

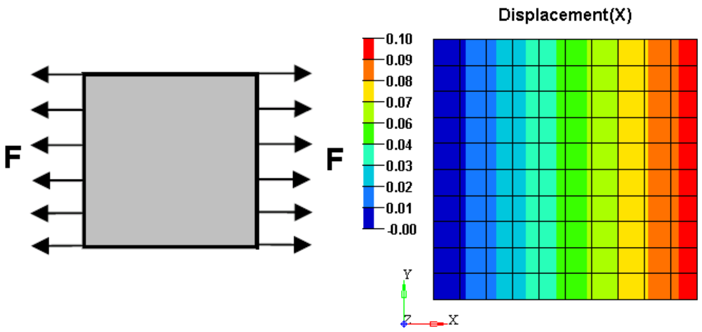
- The material property of the composites can be engineered according to the application requirements.
- The ability to impart the required material property gives them great advantage when compared with traditional homogeneous materials like steel or aluminum.
- Composites have increased strength to weight ratios in use cases against isotropic metals

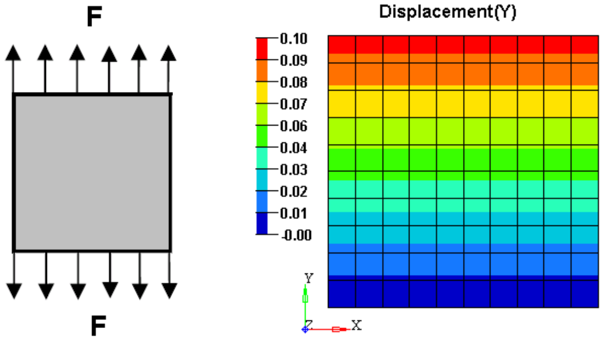
Composite Designable Material Properties

Take the following example:

A simple square steel plate in tension needs to have displacement of 0.1 in x-direction.

- Designing for above requirement is a simple task
- What is the associated displacement of the part for the same loading in the y-direction?





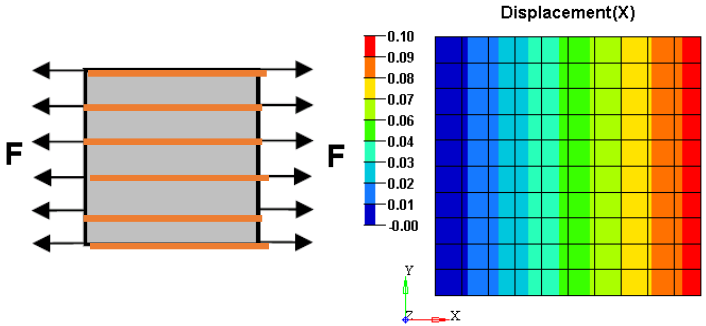
What if the displacement in the y-direction needs to be no more than 0.025 units?

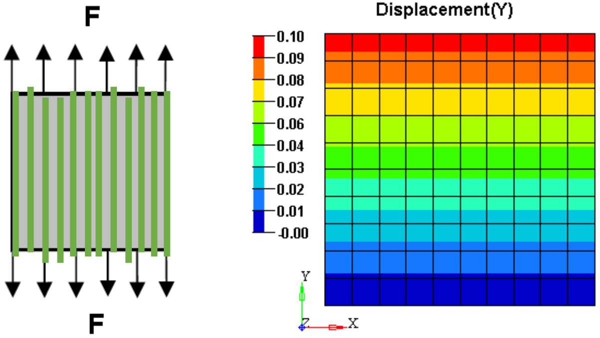
ALTAIR

Composite Designable Material Properties

Using isotropic vs orthotropic materials force different approaches to this design problem

- Steel, being an isotropic material, can not change its properties in different directions. Hence different behavior in different directions needs to be achieved through changing the geometry.
- In case of composites, achieving the above is as simple as determining the correct number of plies in x and y directions.





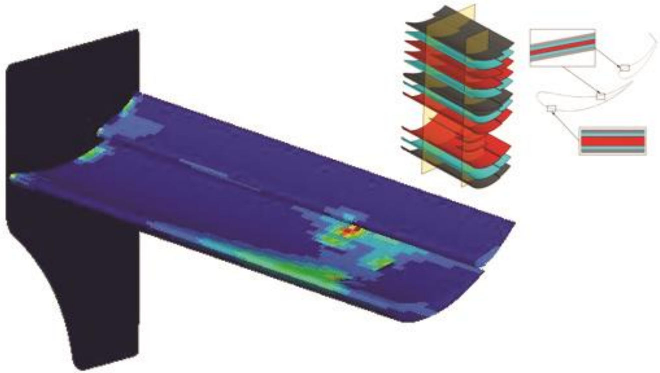
- Ability to design the material property gives lot of freedom to the designers but increases the complexity of the design task.
- Orthotropic designs must take into account undesirable behaviors like extensional–shear coupling, bending-twist coupling, etc

ALTAIR

Finite Element Simulation: Metals vs. Composites

FE of Metal Structures

- Geometry
- Material Properties (Isotropic)
- Loads and BC's
- Visualization of Results on the Geometry
- Failure based on Invariants



FE of Composite Structures

- Geometry
- Material Properties (Non-Isotropic)
- Ply Orientations
- Constituent Properties
- Loads and Boundary Conditions
- Visualization of Results on Geometry, Thru-Laminate, and Constituent Level
- Failure is based on 3D Stress State, is Directional, and Dependent on Constituent Properties

ALTAIR

Composite Pre-Processing FEA Review

How are laminate composites generally modeled in a FEA simulation environment?

- Composites can be modeled using both shells and solids.
- In case of solids:
  - each ply needs to be modeled with at least one solid element. This requires a huge number of solid elements to model a simple plate.
  - OS supports Layered Solid Shell Property for Composite Elements
- Majority of the real life parts are modeled with shell elements.
- Analysis of composite shells is very similar to the solution of standard shell elements.
- An element is modeled as composite by assigning a composite property (e.g. PCOMP, PCOMPG, PCOMPP or PCOMPLS) to it.
- Composite material properties in general are modeled with an orthotropic material model (e.g. MAT8).

Modeling Composites for FE Analysis

Modelling laminate composites for FEA requires more information than isotropic parts:

- Part Geometry
- Mesh Data
- Material Alignment Information
- Ply Geometry
- Lay-up sequence
- Material Data
- Z-Offset Information

Composite Material and Element Orientation and Ply Alignment

Why is material orientation is very important for composite models?

Consider a sample material property which is defined as:

$E_1 = 1.3e^5$ MPa

$E_2 = E_3 = 9650$ MPa

$\upsilon_{12} = \upsilon_{13} = 0.3$

$\upsilon_{23} = 0.6$

$G_{12} = G_{13} = 3450$ MPa

$G_{23} = 3100$ MPa

$a_1 = 1.0e^{-7 mm/mm}/^oC$

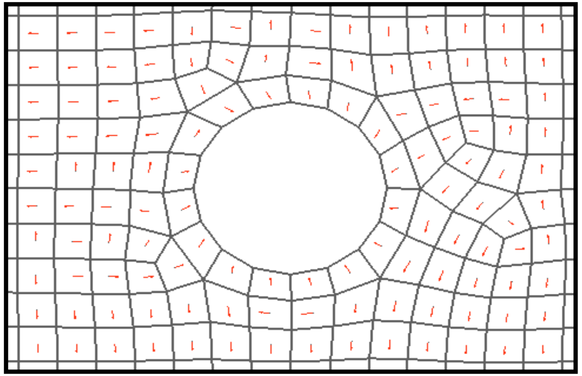
$a_2 = a_3 = 18.0e^{-6\ mm/mm}/^oC$

E_1 is much stiffer than E_2. But, in which directions are E_1 and E_2 measured?

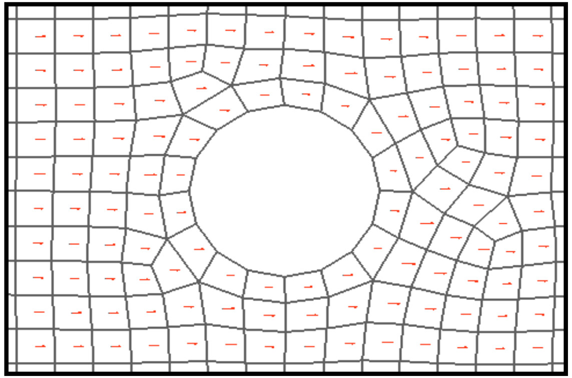
Composite Material and Element Orientation and Ply Alignment

The foundation for developing a properly-oriented composite model is material alignment

- Material and/or element orientation systems provide the point of reference for ply theta definition
- Since element coordinate system is strongly dependent upon the node numbering in individual elements, it is advisable to prescribe a material coordinate system for composite elements and specify ply angles relative to this system.
    - The direction for E_1 is considered the x-axis of the element's material coordinate system
    - The direction for E_2 corresponds to the y-axis of the element's material coordinate system



Material Orientation by default
(based on element node numbering)



Material Orientation by specifying
material orientation angle

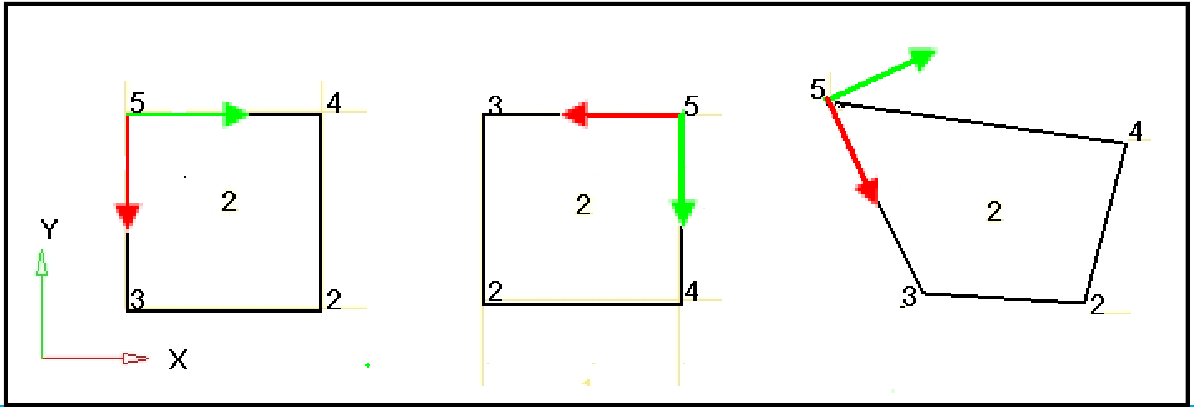
ALTAIR

Composite Material and Element Orientation and Ply Alignment

For shell elements using anisotropic materials, the x-axis of the material system defaults to the vector from G1-G2, parallel to the first and second nodes of the element definition

- Note that the element coordinate system and the material coordinate system are not the same concept
- The element coordinate system is always defined by the bi-section of vectors from G1-G3 and G2-G4

	EID	PID	G1	G2	G3	G4
CQUAD4	2	1	5	3	2	4

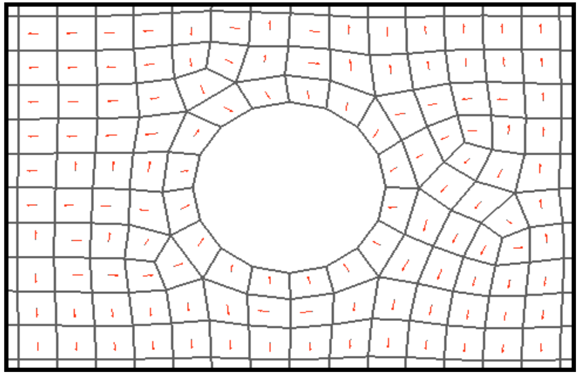


ALTAIR

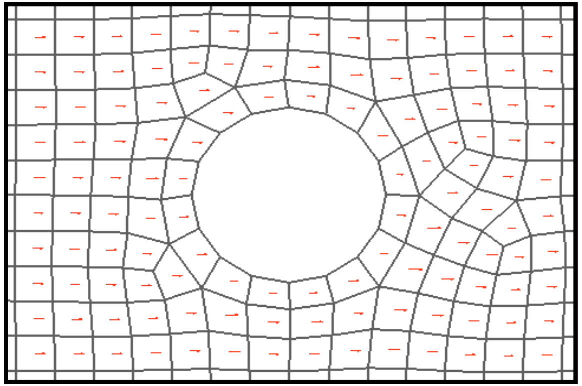
Composite Material and Element Orientation and Ply Alignment

Element material orientations for each element of an entire mesh for a composite part would <u>only</u> be aligned to each other if each element's G1 and G2 nodes were parallel

- Alignment by default is only possible in rectilinear meshes
- Even for rectilinear meshes, the vectors may not align due to what order the nodes were written into the element cards during meshing



TIP: <u>Always</u> review mesh orientation before analysis



Material coordinate system should be defined to align the E_1 and E_2 to the desired direction

Material coordinate system can be defined by defining an angle or a co-ordinate system

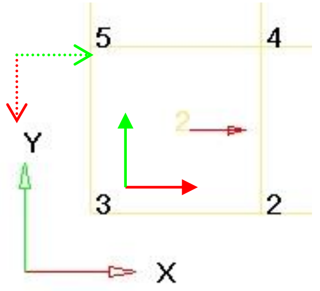
Composite Material and Element Orientation and Ply Alignment

Element material orientation can be defined independently within each element as an angle rotated by THETA degrees from the x-axis of the element coordinate system

THETA = 90 degree

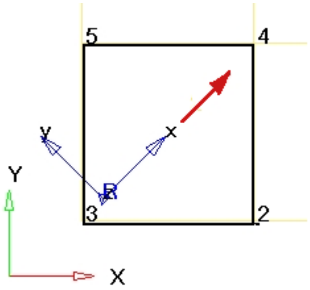
- X (G1 → G2) Rotated by THETA
- Z = Element Normal

	EID	PID	G1	G2	G3	G4	THETA
CQUAD4	2	1	5	3	2	4	90.00(



- MCID: X is defined by the local coordinate system.

	EID	PID	G1	G2	G3	G4	MCID
CQUAD4	2	1	5	3	2	4	1

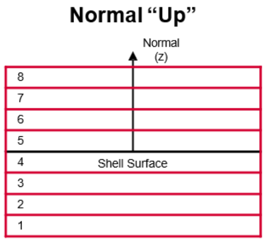


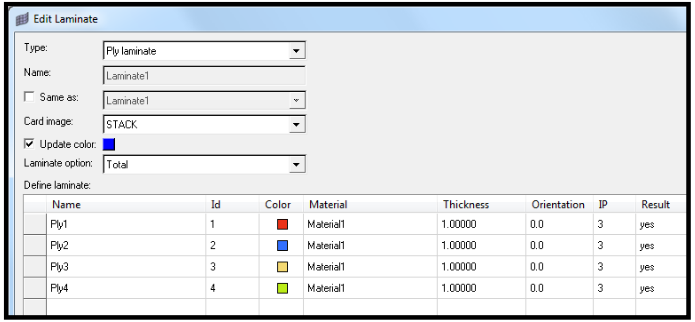
ALTAIR

Ply / Laminate Modelling for OptiStruct

Plies are added to laminate in the normal direction

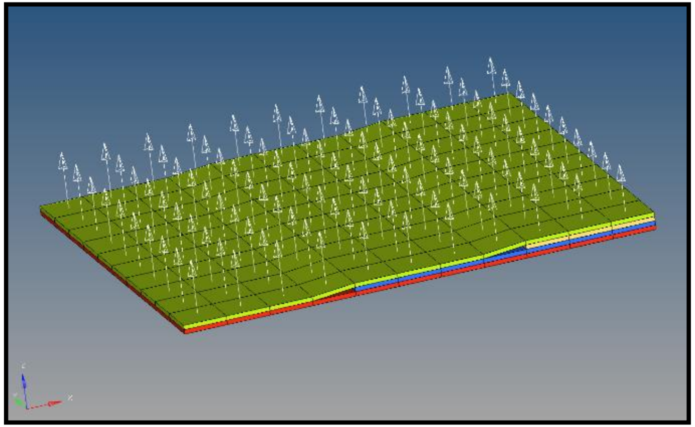
- First ply = bottom surface
- Last ply = top surface





A ply can only appear once in a laminate

- The ply ID must be unique as separate results are written for each ply
- Cannot reuse plies in a laminate currently



Creating the Laminate in OptiStruct

Creating a laminate from shell elements requires creating property cards to define the ply and sequence information

There are 3 main types of property cards to choose between for creating shell laminate properties:

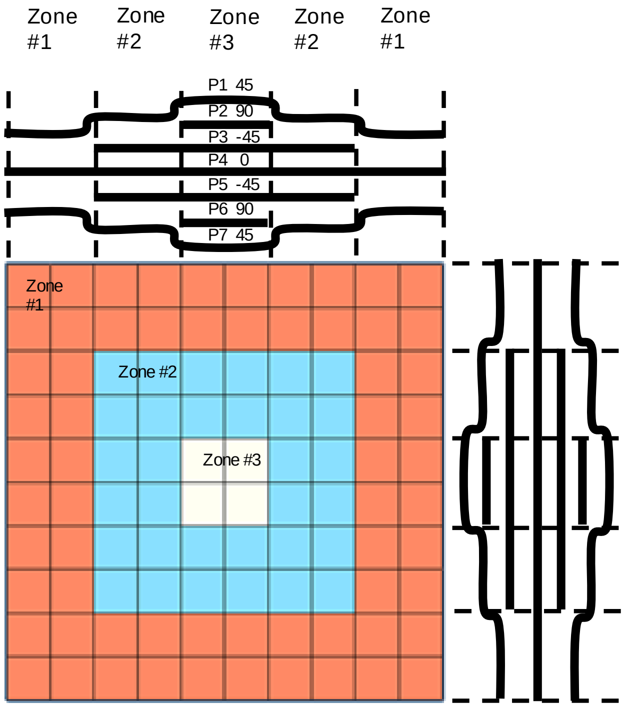
- PCOMPP: Composite Laminate Property for Ply-Based Composite Definition
- PCOMPG: Composite Laminate Property allowing for global ply identification
- PCOMP: Composite Laminate Property

Using PCOMPP (ply-based modeling) is recommended over PCOMPG and PCOMP methods

- Modern ply-based modeling is more flexible than and more synchronous with manufacturing methods than PCOMP or PCOMPG (zone-based)

Traditional Zone-Based Composite Modeling

Requires one property for each laminate zone



Zone-based modeling limitations include:

1. Data duplication
2. Difficult to interpret ply shape
3. No relationship to the manufacturing process
4. Model updates require multiple steps

Zone #1 Property Table			
Ply	Mat	Thk	Theta
P7	M1	0.01	45
P1	M1	0.01	45

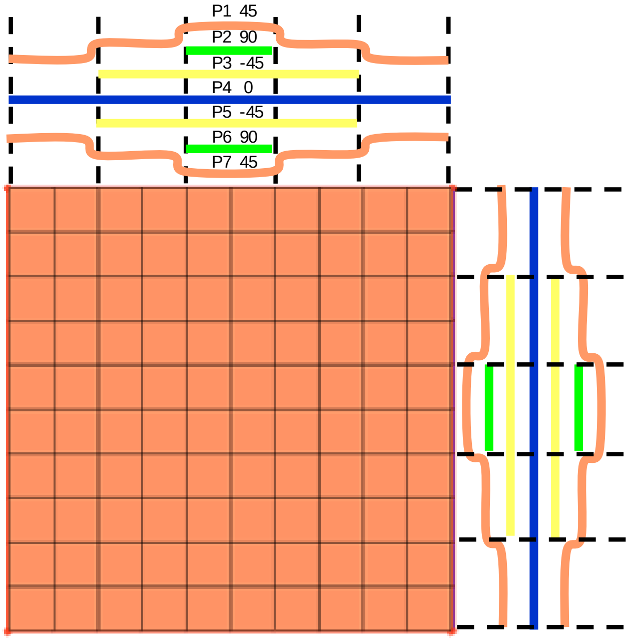
Zone #2 Property Table			
Ply	Mat	Thk	Theta
P7	M1	0.01	45
P5	M1	0.01	-45
P3	M1	0.01	-45
P1	M1	0.01	45

Zone #3 Property Table			
Ply	Mat	Thk	Theta
P7	M1	0.01	45
P6	M1	0.01	90
P5	M1	0.01	-45
P3	M1	0.01	-45
P2	M1	0.01	90
P1	M1	0.01	45

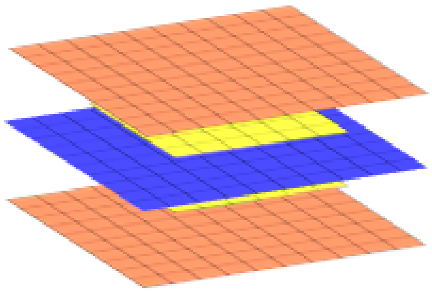
OptiStruct Offers Modern Ply-Based Composite Modeling

An individual composite part is one laminate made up of plies which are stacked in a given sequence!

- No data duplication
- Plies defined as “physical objects” w/ shape
- Direct relationship to the manufacturing process
- Model updates require single step



Stack Table			
Ply	Mat	Thk	Theta
P7	M1	0.01	45
P6	M1	0.01	90
P5	M1	0.01	-45
P3	M1	0.01	-45
P2	M1	0.01	90
P1	M1	0.01	45



OptiStruct Offers PCOMPP for Ply-based Modeling

PCOMPP is a ply-based modeling approach for modern composite analysis

- PCOMPP adopts ply based composite definition which is similar to existing laminate tools and to the Ply-Book used in typical manufacturing set up.
- Ply definition, stacking and the property are defined separately through independent cards (PLY, STACK, and PCOMPP).
  - PLY card defines fiber orientation and layout (element sets)
  - STACK card sets the sequence of the PLYs into a laminate

Note that element properties are set implicitly through STACK and PLY, replacing PCOMP and PCOMPG explicit laminate definitions

- This provides additional flexibility in manipulating laminates in both analysis and optimization

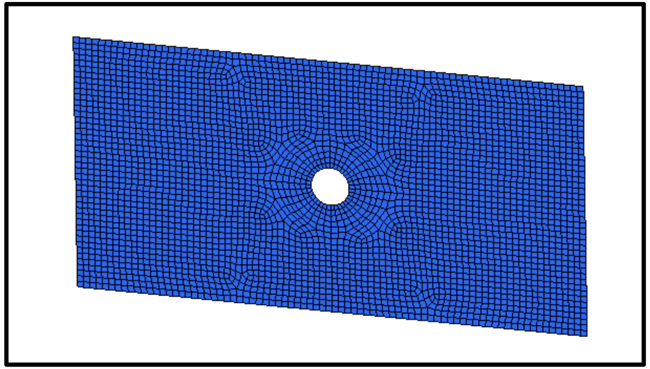
Exercise 1: Simulating a Plate With Hole Test Coupon (20 min)

Exercise 1:

Simulating a Plate With Hole Test Coupon

Objectives:

- Open the model in HyperMesh
- Align the element orientations with the global x-axis
- Create a new PCOMPP & assign to the component
- Create six plies with all elements as the ply shapes
- Create a new laminate
- Set up Loads and Loadstep
- Run in Optistruct



Understanding Composite Material Properties

The strain-stress relationship for isotropic linear elastic materials is given by:

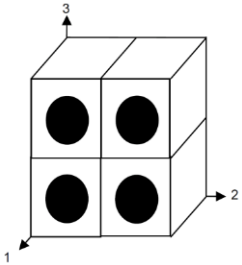
$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{13} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E} & \frac{-\upsilon}{E} & \frac{-\upsilon}{E} & 0 & 0 & 0 \\ \frac{-\upsilon}{E} & \frac{1}{E} & \frac{-\upsilon}{E} & 0 & 0 & 0 \\ \frac{-\upsilon}{E} & \frac{-\upsilon}{E} & \frac{1}{E} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G} \end{bmatrix} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \tau_{12} \\ \tau_{23} \\ \tau_{13} \end{Bmatrix}$$

As shown, isotropic linear elastic materials have only two independent engineering constants. Any two of E, G, or ν which are related by the equation:

$$G = \frac{E}{2(1+\upsilon)}$$

Understanding Composite Material Properties

Laminated composite material properties are generally modeled as orthotropic materials



Thus, the strain-stress relationship can be rewritten as the following:

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{13} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_1} & \frac{-\upsilon_{21}}{E_2} & \frac{-\upsilon_{31}}{E_3} & 0 & 0 & 0 \\ \frac{-\upsilon_{12}}{E_1} & \frac{1}{E_2} & \frac{-\upsilon_{32}}{E_3} & 0 & 0 & 0 \\ \frac{-\upsilon_{13}}{E_1} & \frac{-\upsilon_{23}}{E_2} & \frac{1}{E_3} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{12}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{23}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{13}} \end{bmatrix} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \tau_{12} \\ \tau_{23} \\ \tau_{13} \end{Bmatrix}$$

Understanding Composite Material Properties

The orthotropic strain-stress relationship for plane stress conditions can be further reduced, as:

$$\sigma_3 = \tau_{13} = \tau_{23} = 0 \qquad \frac{\nu_{ij}}{E_i} = \frac{\nu_{ji}}{E_j} \qquad i, j = 1,2,3. \qquad i \neq j$$

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_1} & \frac{-\upsilon_{12}}{E_1} & 0 \\ \frac{-\upsilon_{12}}{E_1} & \frac{1}{E_2} & 0 \\ 0 & 0 & \frac{1}{G_{12}} \end{bmatrix} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix}$$

with: $\varepsilon_3 = -\left(\frac{\upsilon_{13}}{E_1}\sigma_1 + \frac{\upsilon_{23}}{E_2}\sigma_2 \right)$

Composite Material Types

OptiStruct supports three types of material definitions for 2D composite modeling

MAT8:

- Linear Temperature-Independent Orthotropic Material
- The typical material model used for composites
- Used for 2-dimensional elements

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_1} & \frac{-\upsilon_{12}}{E_1} & 0 \\ \frac{-\upsilon_{12}}{E_1} & \frac{1}{E_2} & 0 \\ 0 & 0 & \frac{1}{G_{12}} \end{bmatrix} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix}$$

MAT2:

- Linear Temperature-Independent Anisotropic Material
- Used for 2-dimensional elements

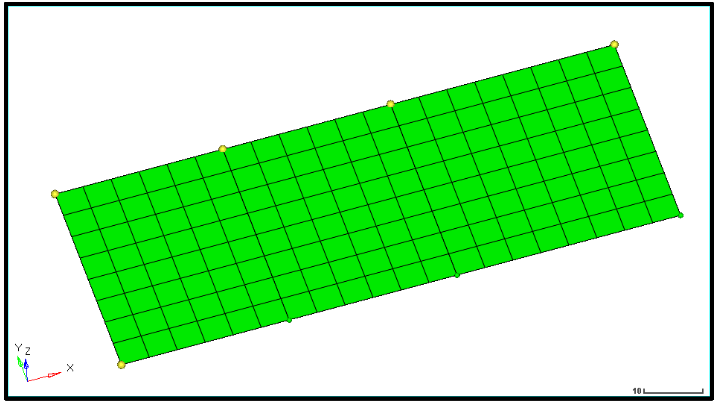
$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} \right.$$

MAT1:

- Linear Temperature-Independent Isotropic Material
- Used with shell elements under plane stress formulation

$$\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{13} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E} & \frac{-\upsilon}{E} & \frac{-\upsilon}{E} & 0 & 0 & 0 \\ \frac{-\upsilon}{E} & \frac{1}{E} & \frac{-\upsilon}{E} & 0 & 0 & 0 \\ \frac{-\upsilon}{E} & \frac{-\upsilon}{E} & \frac{1}{E} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G} \end{bmatrix} \begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \tau_{12} \\ \tau_{23} \\ \tau_{13} \end{Bmatrix}$$

Exercise 2: Creating Ply-based Laminates using Geometry (15 min)



Exercise 2:

Creating Ply-based Laminates using Geometry

Objectives:

- Open File: Geometry.hm
- Steps 1-3 have been completed
- Create a Mat8 material
- Create a new PCOMPP property with default parameters and assign to the component
- Create four plies using the geometric lines as boundaries
- Create the laminate with the four plies in sequence
- Create the 2d mesh
- Realize the plies

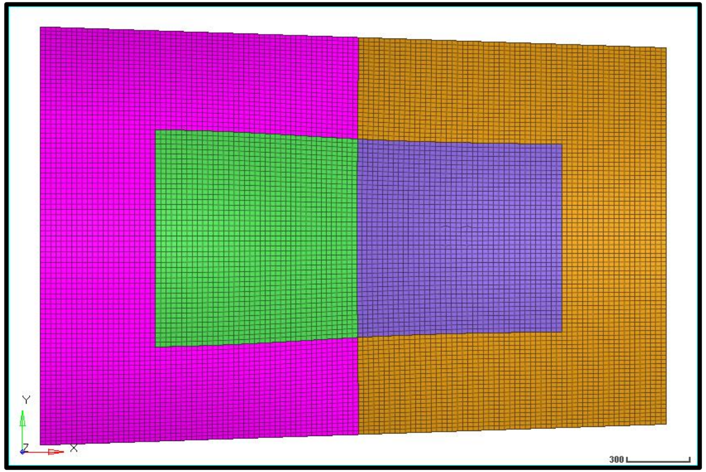
Exercise 3: Analysis of a Composite Underbelly Fairing

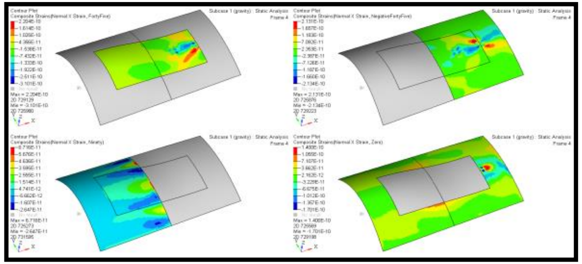
Exercise 3:

Analysis of a Composite Underbelly Fairing

Objectives:

- Open the model in HyperMesh Desktop
- Create plies for the model
- Create laminate with ply order
- Run the analysis and post-process the model results using **HyperView**

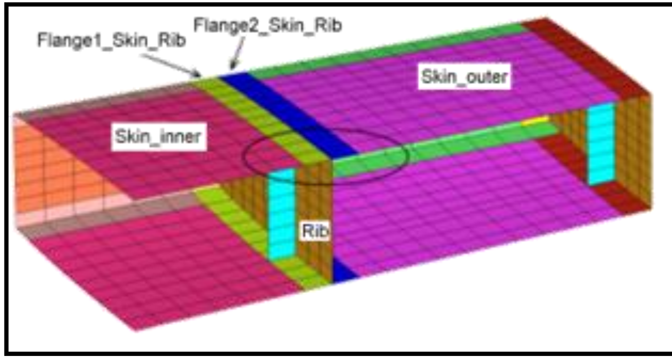


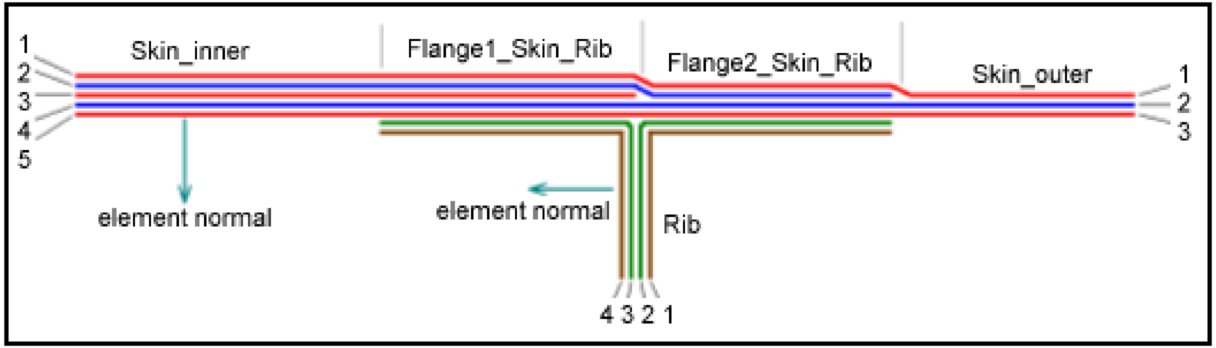


Choosing Between PCOMPG and PCOMP Property Definition

Generally, PCOMPG is preferred to PCOMP for modeling zone-based composite parts

- PCOMP definition contains no information on plies that are also part of other regions
  During post-processing, this requires lot of book keeping to track ply and stacking information for each PCOMP

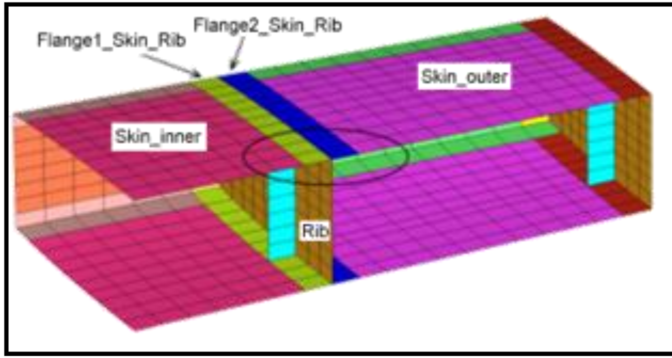


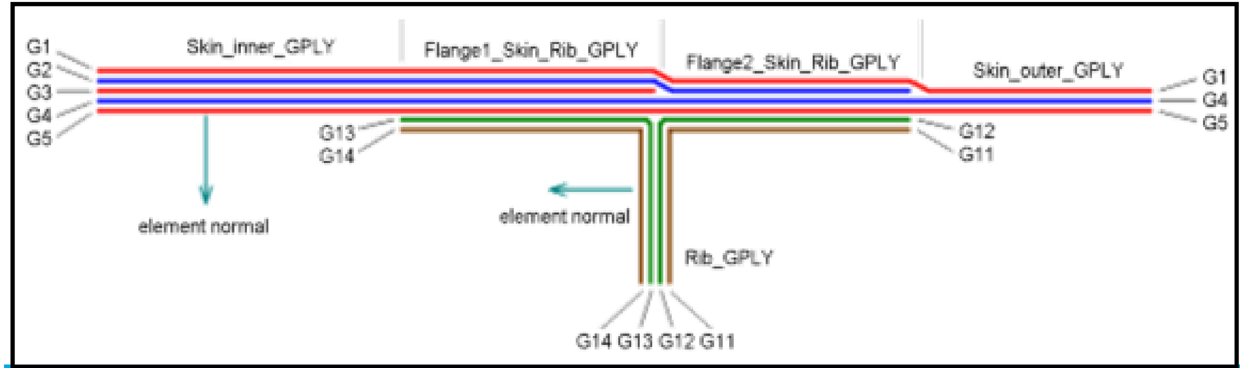


Choosing Between PCOMPG and PCOMP Property Definition

Generally, PCOMPG is preferred to PCOMP for modeling zone-based composite parts

- Through the global ply identification number, PCOMPG plies that are part of many regions can be tracked across the regions, reducing the effort for keeping track of the ply properties and stacking information





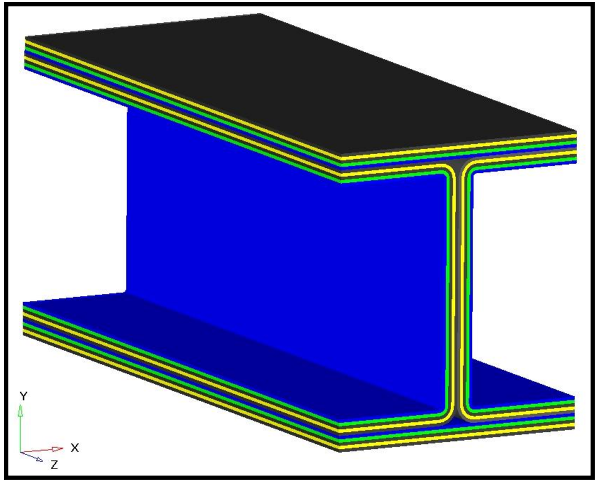
Exercise 4: Analysis of a PCOMPG Clamped I-Beam (20 min)

Exercise 4:

Analysis of a PCOMPG Clamped I-Beam

Objectives:

- Open the model in HyperMesh Desktop
- Create (two) PCOMPG properties
- Assign appropriate prop to the elements it represents
- Run the analysis
- Post-process the model results using **HyperView**



ALTAIR

Macromechanical Behavior of a Ply

The constitutive stress/strain relationship is written in the principal material 1-, 2-, 3- coordinate system:

$$\sigma_i = Q_{ij}\left(\varepsilon_j\right)$$

In order to determine the global behavior of a ply, this relationship is transformed to the global x-, y-, z- coordinate system, using the 2D plane stress transformations:

$$\{\sigma_x\} = [TS]^{-1}[Q][TS]^{-T}\left(\{\varepsilon_x\}\right)$$

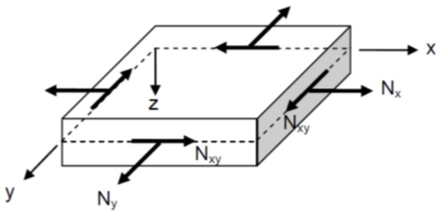
OR

$$\{\sigma_x\} = \left[\overline{Q}\right]\left(\{\varepsilon_x\}\right)$$

Where $\left[\overline{Q}\right] = [TS]^{-1}[Q][TS]^{-T}$ is the stiffness matrix in the global coordinate system

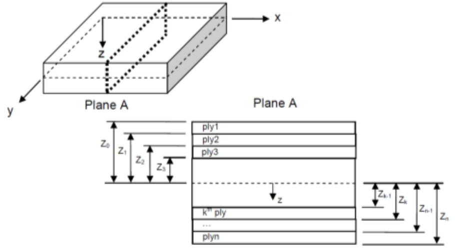
Classical Lamination Theory: Mid-Plane Forces

For a homogenous single ply plate of constant thickness the mid-plane forces can be written in terms of stress variation through the thickness of the plate as:



$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \int_{-t/2}^{t/2} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} dz$$

For a laminated plate made up of ‘n’ constant thickness plies the mid-plane forces can be written in terms of the sum of the stress variation through the thickness of each ply as:

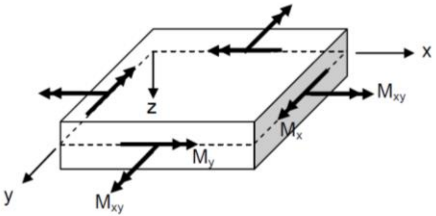


$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \sum_{k=1}^{n} \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} dz$$

ALTAIR

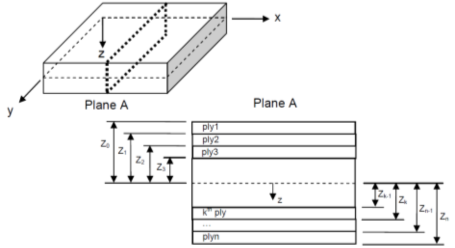
Classical Lamination Theory: Mid-Plane Moments

For a homogenous single ply plate of constant thickness the mid-plane moments can be written in terms of stress variation through the thickness of the plate as:



$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \int_{-t/2}^{t/2} z \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} dz$$

For a laminated plate made up of 'n' constant thickness plies the mid-plane moments can be written in terms of the sum of the stress variation through the thickness of each ply as:



$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \sum_{k=1}^{n} \int_{z_{k-1}}^{z_k} z \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} dz$$

ALTAIR

Classical Lamination Theory: Stress-Strain Relationship

By adding the subscript "*k*" to designate the equation on the laminated coordinates for each ply the general stress-strain relationship,

$$\{\sigma_x\} = [\overline{Q}](\{\varepsilon_x\})$$

can be represented as:

$$\{\sigma_x\}_k = [\overline{Q}]_k(\{\varepsilon_x\}_k)$$

Substituting the above into the equation for mid-plane forces moments, it is shown:

$$\{N_x\} = \sum_{k=1}^{n} \int_{z_{k-1}}^{z_k} ([\overline{Q}]_k(\{\varepsilon_{x^o}\}_k + z_k\{k\}_k))dz$$

OR

$$\{N_x\} = [A]\{\varepsilon_{x^o}\} + [B]\{k_x\}$$

$$\{M_x\} = \sum_{k=1}^{n} \int_{z_{k-1}}^{z_k} ([\overline{Q}]_k(\{\varepsilon_{x^o}\}_k + z_k\{k\}_k))zdz$$

OR

$$\{M_x\} = [B]\{\varepsilon_{x^o}\} + [D]\{k_x\}$$

The [A] [B] [D] Matrix

The definition of the relationship between the mid-plane generalized forces and strain can be written as:

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix} \begin{Bmatrix} \varepsilon_0 \\ k \end{Bmatrix}$$

The [A], [B] and [D] matrices in the above relation have a lot of significance in designing the laminates of a composite structure

By looking at these matrices the designer can determine:

- Various behaviors, such as if the laminate is balanced
- The nature of any coupling(s) between extension, shear, bending, twisting etc.

ALTAIR

The [A] [B] [D] Matrix

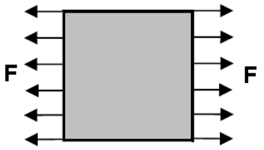
The [B] matrix relates mid-plane forces to plate curvatures and mid-plane moments to mid-plane strains.

The [B] matrix is zero for symmetric laminates

The [A] matrix relates mid-plane forces to mid-plane strains defining the extensional behavior of the laminate

The [A] matrix is stacking sequence independent

$$
\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \\ M_x \\ M_y \\ M_{xy} \end{Bmatrix} =
\left[\begin{array}{ccc|ccc}
A_{11} & A_{12} & A_{14} & B_{11} & B_{12} & B_{14} \\
A_{12} & A_{22} & A_{24} & B_{12} & B_{22} & B_{24} \\
A_{14} & A_{24} & A_{44} & B_{14} & B_{24} & B_{44} \\
\hline
B_{11} & B_{12} & B_{14} & D_{11} & D_{12} & D_{14} \\
B_{12} & B_{22} & B_{24} & D_{12} & D_{22} & D_{24} \\
B_{14} & B_{24} & B_{44} & D_{14} & D_{24} & D_{44}
\end{array}\right]
\begin{Bmatrix} \varepsilon_{x^o} \\ \varepsilon_{y^o} \\ \gamma_{xy^o} \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}
$$



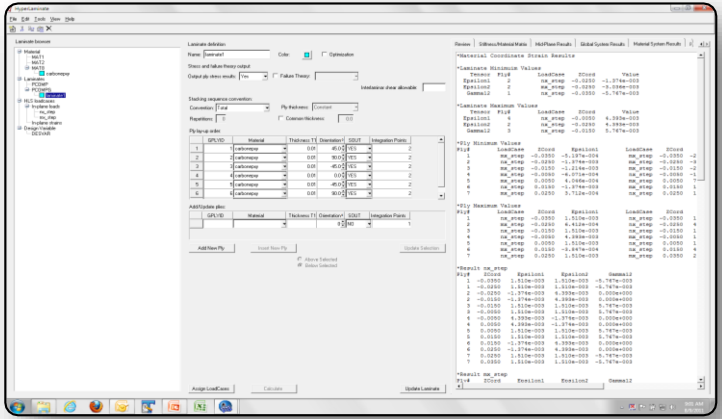
The [D] matrix relates mid-plane moments to plate curvatures defining the bending behavior of the laminate

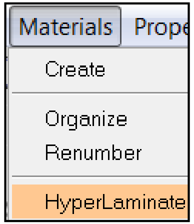
The [D] matrix is stacking sequence dependent and is most affected by the location of zero degree plies in the stacking sequence.

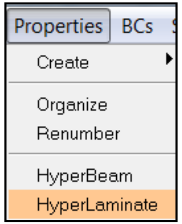
Introduction to HyperLaminate

HyperLaminate is a HyperMesh module that facilitates the creation, review and edition of composite laminates (Zone based PCOMP and PCOMPG)

- The **HyperLaminate Solver** (HLS) uses classical laminated plate theory for simple in-plane analysis of composite laminates.
- Calculation of Homogenized Laminate Stiffness's
  - Classical Lamination Theory Matrices [A], [B], [D]
  - Equivalent Material Matrices [G1], [G2], [G3], [G4]
  - Homogenized Membrane/Bending Laminate Properties E_1, E_2, ν_{12}, G_{12}, α_1, α_2
- **HyperLaminate** is launched from the **Materials** or **Properties** pull-down menus.





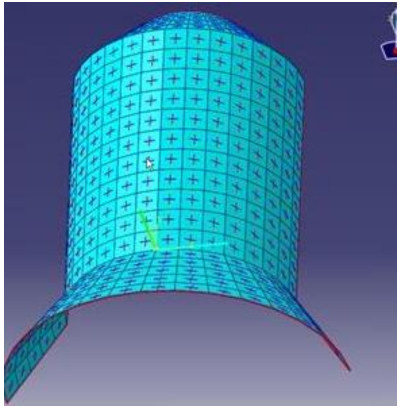


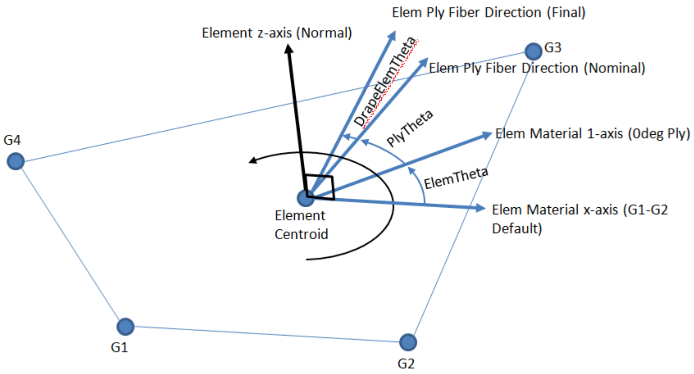
ALTAIR

Incorporating Draping Data Into Composite Simulation

Draping data corrects the thickness and fiber direction for every element within a ply

- Draping information is geometrically and mechanically based
  - With flat & singly curved surfaces, orientation of the ply stays unchanged over the whole application area, but for doubly curved surfaces, a ply can follow the surface only by deforming.
- Draping information can be imported from FiberSim or calculated through HyperWorks





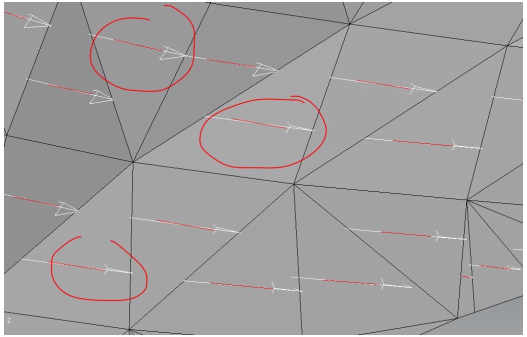
HyperWorks Drape Estimator

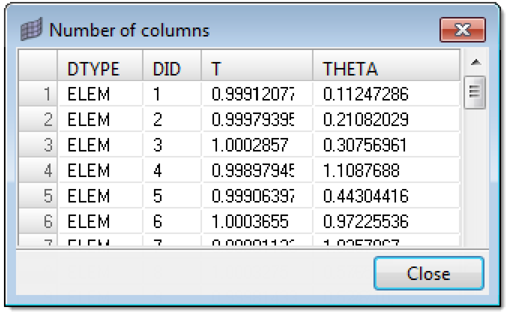
The HyperWorks Drape Estimator can calculate, based upon ply geometry:

- Draping angles
- Thickness variation

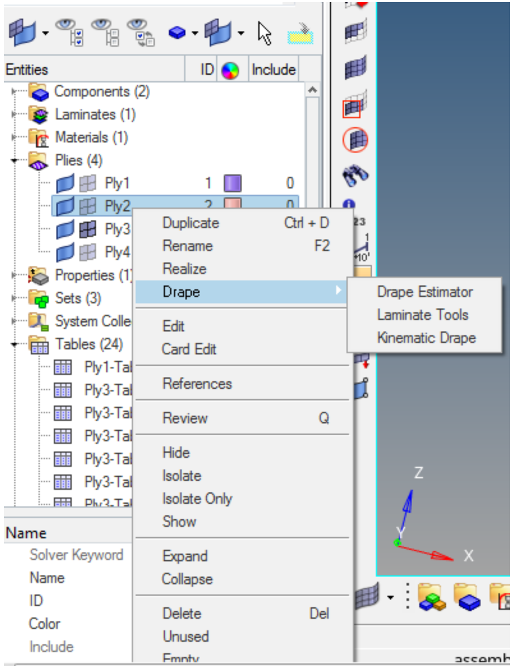
Draping data is stored in tables, containing drape data for the elements

- Change in ply thickness due to draping is given as a scale factor
- The change in fiber angle due to draping is noted in degrees





	DTYPE	DID	T	THETA
1	ELEM	1	0.9991207	0.11247286
2	ELEM	2	0.9997939	0.21082029
3	ELEM	3	1.0002857	0.30756961
4	ELEM	4	0.9989794	1.1087688
5	ELEM	5	0.9990639	0.44304416
6	ELEM	6	1.0003655	0.97225536
7	ELEM	7	[illegible]	[illegible]

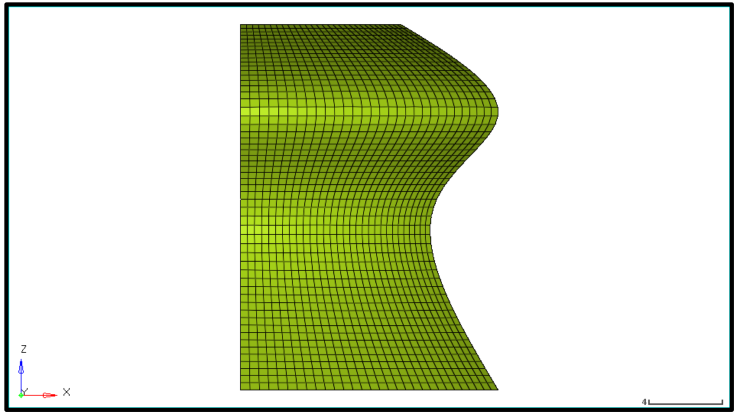


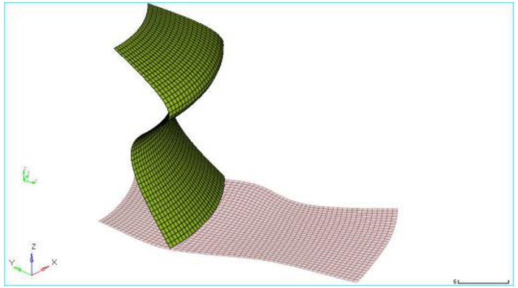
ALTAIR

Exercise 5: Kinematic Draping (10 min)

Exercise 5:

Using the HyperWorks Drape Estimator





Objectives:

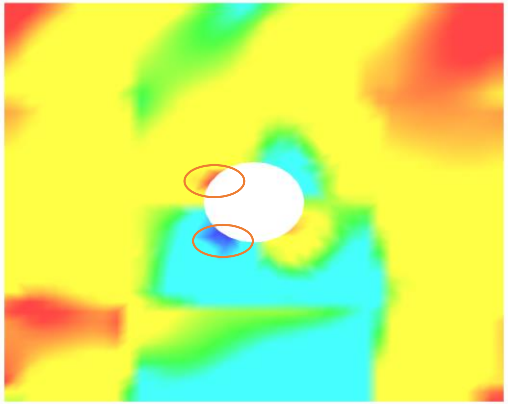
- Open the model
- Review the model's material orientation and confirm the orientation of each ply
- Use the drape estimator to calculate the change in orientation and thickness of each element in every ply
- Review the drape tables
- Review the drape results visually using the Drape Estimator

ALTAIR

FE Failure Simulation Factors: Metals vs. Composites

Isotropic Materials

- Von Mises Stress determines failure
- Geometry changes (shape or thickness) will prevent failure.



Black Aluminum Design - Size and Shape

Laminate Composites

- In what ply layer is the failure?
- Is it a fiber failure or matrix failure?
- Will a crack form?
- Is it a delamination failure?
- If I change the ply angle, will it prevent failure?
- If I change the stacking sequence, will it prevent failure?
- Will the failure propagate?

ALTAIR

Laminate Failure Theories in OptiStruct

A range of standard failure theories are available in OptiStruct

- Compare ply stresses or strains to material allowables
- Produce contours to identify where failure will occur

Failure theories available:

- Hill
- Hoffman
- Tsai-Wu
- Maximum Stress / Maximum Strain
- Hashin
- Puck

$$F_{Hill} = \left(\frac{\sigma_1^2}{X^2}\right)+\left(\frac{\sigma_2^2}{Y^2}\right)-\left(\frac{\sigma_1\sigma_2}{X^2}\right)+\left(\frac{\tau_{12}^2}{S^2}\right)$$

Additional results for laminate composite failure are available from OptiStruct

- Bonding failure
- Failure index

ALTAIR

Producing Failure Results Using OptiStruct

1. Add material allowables into the ply materials

- Orthotropic materials (MAT8)
  - Xt - Allowable tensile stress or strain in the longitudinal direction
  - Xc - Allowable compressive stress or strain in the longitudinal direction
  - Yt - Allowable tensile stress or strain in the lateral direction
  - Yc - Allowable compressive stress or strain in the lateral direction
  - S – Allowable in-plane shear stress or strain
  - STRN – Option to choose if allowables are for stresses (blank) or strains (1.0)
- Isotropic materials (MAT1), anisotropic materials (MAT2)
  - ST - Stress limit in tension
  - SC - Stress limit in compression
  - SS - Stress limit in shear

2. Set the stress output (SOUT) option to YES on the plies to include in the failure calculations (PLY, PCOMP, PCOMPG)

Producing Failure Results Using OptiStruct

3. Choose the failure theory (FT) to use on the composite property (PCOMPP, PCOMP or PCOMPG):

- HILL for Hill theory
- TSAI for Tsai-Wu theory
- HASH for Hashin
- HOFF for Hoffman theory
- STRN for Maximum Strain Theory
- PUCK for Puck

4. Create a MATF card to specify the failure criteria parameters (req. for PUCK)

5. For bonding failure calculations, add bonding shear stress allowable into composite property (optional)

- SB - Allowable inter-laminar shear stress in the bonding material

6. Add output request to write ply stress and failure index output:

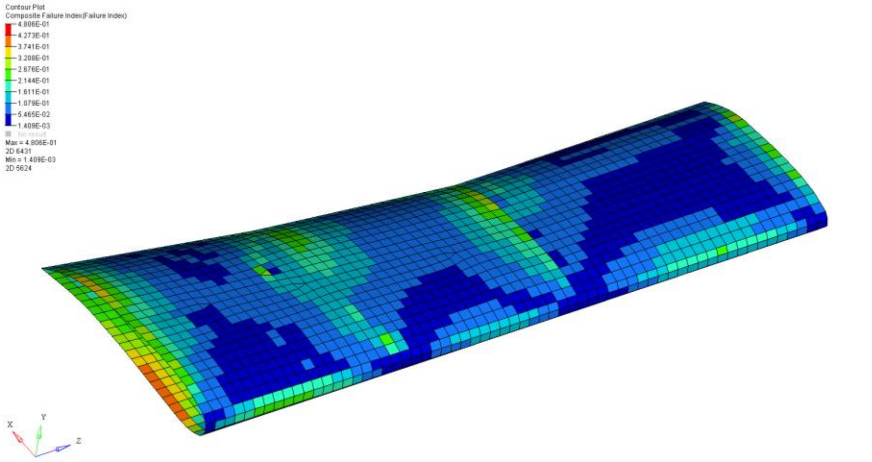
- CFAILURE=ALL
- CSTRESS = ALL

Final Failure Index for Composite Elements

The failure index for the composite shell element is taken from the worst failure values across all the individual plies and bonding layers

- Failure of a single layer qualifies as failure of the composite

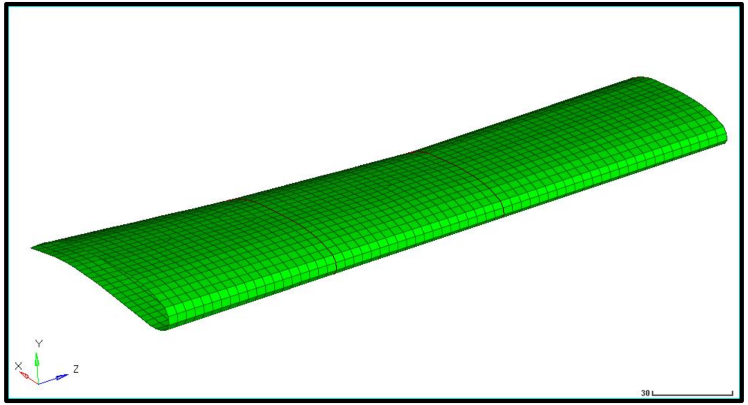
Only plies with requested stress output are taken into account in the failure index calculation



Exercise 6: Generating Failure Results from PCOMP

Exercise 6 :

Generating Failure Results from PCOMP



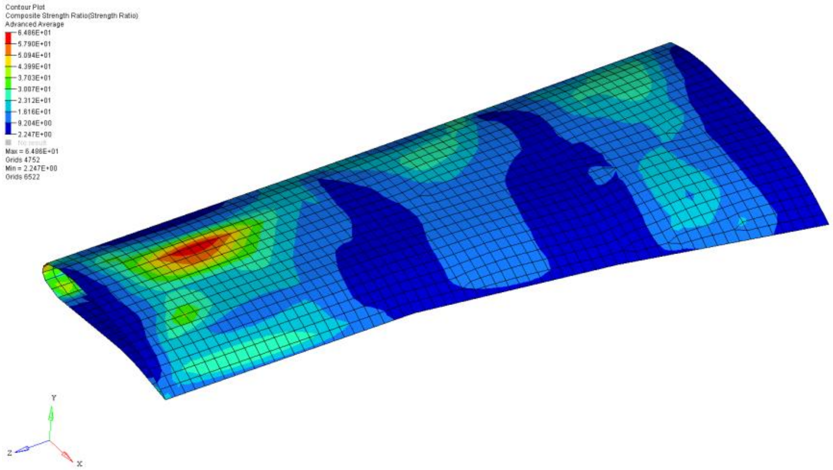
Objectives:

- Open the model in HyperMesh
- Update the model to include failure values
- Set the stress output (SOUT) option to **`YES`** on the PCOMP property
- Set the failure theory (FT) of the Laminate property to **`HILL`** and the SB value to **`11.6`**
- Add output request control cards and parameters
- Save the model as **`wing_failure_output.hm`**
- Run the model in OptiStruct
- Use HyperView to contour the **`Composite Failure Index (s)`** results

Strength Ratio Calculations for Composite Elements

Strength Ratio (SR) is a direct failure indicator whereas failure index only indicates if failure has occurred

- SR = 1.2 indicates that the applied loads can be increased by 20% before failure occurs
- FI = 0.8 indicates that failure has not occurred but does not indicate 20% safety margin



ALTAIR

Strength Ratio Calculations

Calculated by rearranging the failure theory equation

- Replace applied stress by SR times the applied stress
- Equate to a failure value of 1.0
- Rearrange and solve for SR

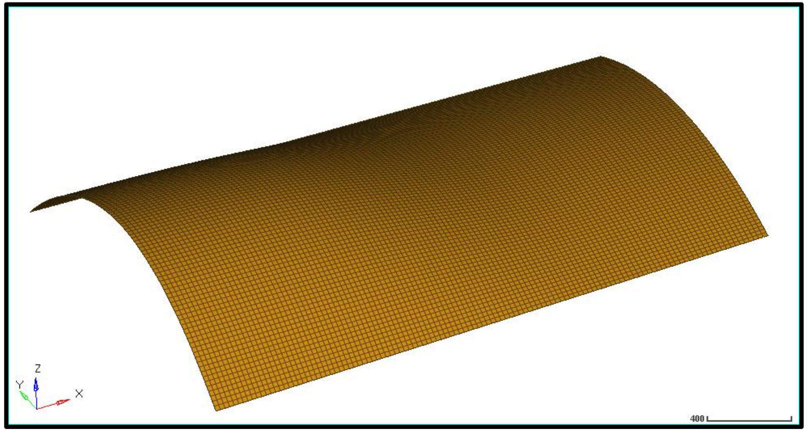
Example for Hill Failure Theory

$$1.0 = \left(\frac{SR^2\sigma_1^2}{X^2}\right)+\left(\frac{SR^2\sigma_2^2}{Y^2}\right)-\left(\frac{SR^2\sigma_1\sigma_2}{X^2}\right)+\left(\frac{SR^2\tau_{12}^2}{S^2}\right)$$

Activated with OptiStruct control card **PARAM,SRCOMPS,YES**

ALTAIR

Exercise 7 : Contouring Strength Ratios in HyperView



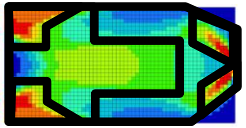
Exercise 7:

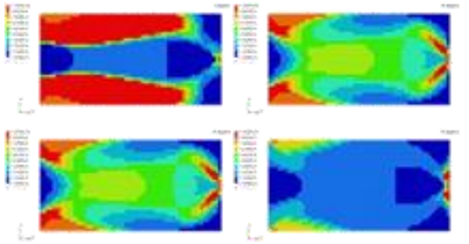
Contouring Strength Ratios in HyperView

Objectives:

- Alter the contour plot to only show results with `SR <= 1.2`
- Create a new window with a copy of the existing plot
- Create a new derived loadstep for use with an envelope plot
- Use the new derived results to envelope plot the composite strength ratio by subcase
  - Steps 5 and 6 are reversed

Composite Optimization: Three Steps From Concept To Final Design

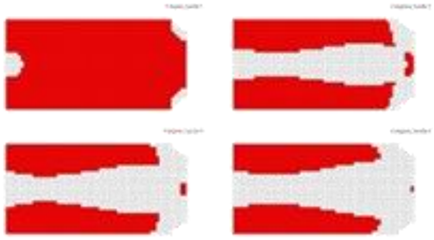




Phase 1: Free Size Optimization

“What are potential efficient ply shapes I could use?”

- Determine composite patch size, shape & location
- Super Ply – Total designable thickness of a particular ply orientation
- Tailoring – Cutting the ply patch shapes for super plies into ply bundles



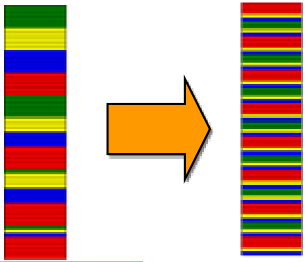
Phase 2: Ply Bundle Size Optimization

“How many of each ply shape do I need?”

- Update ply patches in HM if necessary
- Determine optimum ply bundle thicknesses
- Ply Bundle – A consecutive stack of plies of the same patch shape & angle

Phase 3: Stacking Sequence Optimization

“How should plies be stacked such that design meets engineering constraints?”



- Meet ply book rules
- Improve performance
- Improve manufacturability
- Stacking Sequence – Order of ply layup in the laminate
- Shuffling – Rearranging the stacking sequence of plies

Phase 1: Free Size Optimization Results

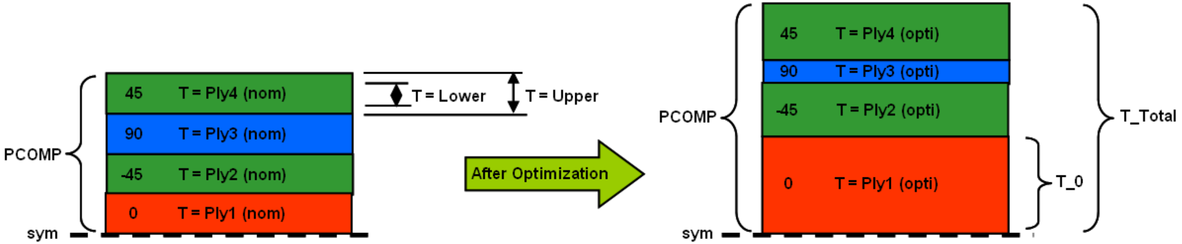
Free-Size Optimization is a concept-level optimization that optimizes the thickness of each ply on an element-by-element basis

To determine the optimum laminate OptiStruct uses the SMEAR technology that captures the stacking sequence effects:

- A = Stacking Sequence independent
- B = 0 (Symmetric)
- D = At^2/12 - Stacking Sequence Independent

The results of this optimization illustrates the optimized geometric ply boundaries

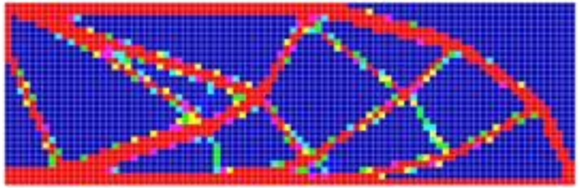
- Elements are grouped into sets according to these geometries
- This process is known as ‘ply tailoring’

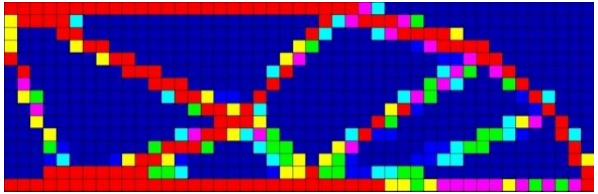


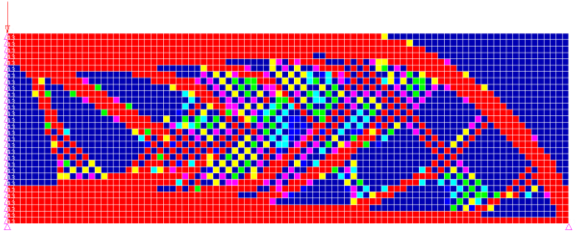
Using Manufacturing Constraints for Practical Design Concepts

Minimum member size control (mindim) specifies the smallest dimension to be retained in concept-level optimization designs

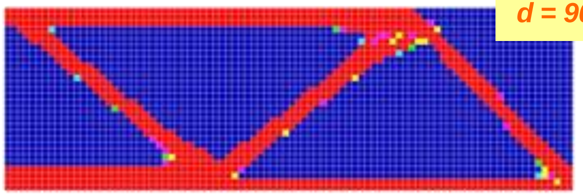
- Controls checker board effect and discreteness
- Min Member Size > 3 x mesh size
- The smallest mindim available in a run is dependent on average mesh size







d = 60



d = 90

Without min member size

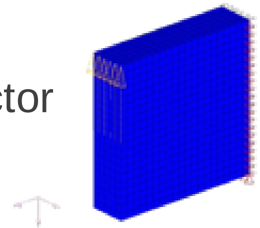
- **Difficult to manufacture due to micro structures**
- **Results are mesh dependent**

ALTAIR

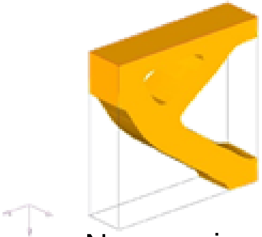
Using Manufacturing Constraints for Practical Design Concepts

Pattern grouping provides model symmetry control during optimization

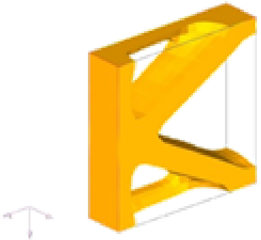
- The amount of control is indicated by how many planes of symmetry are needed
- Each plane of symmetry is specified by a normal vector
- 1-plane symmetry has one anchor node which serves as the base of the plane and a first node which orients the vector
- 2- and 3-plane symmetries add second and third nodes, respectively, for orthogonal planes



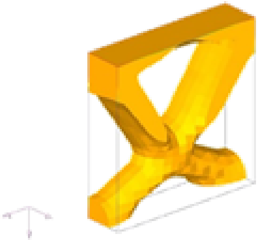
Original Model



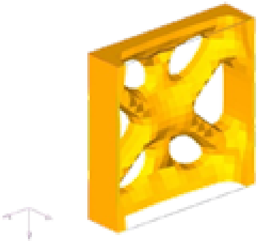
No grouping



1-pln symmetry (YZ)



1-pln symmetry (XZ)



2-pln symmetry (XZ & YZ)

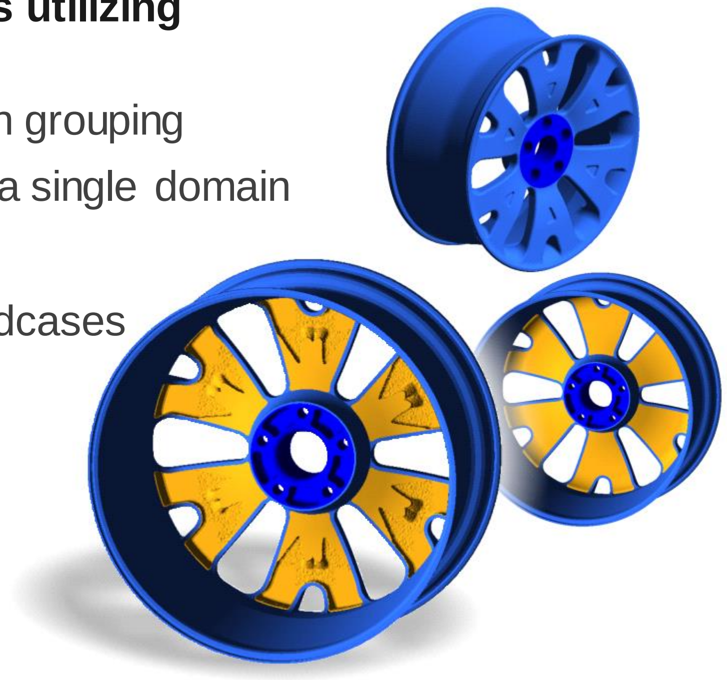
ALTAIR

Using Manufacturing Constraints for Practical Design Concepts

Pattern grouping provides model symmetry control during optimization

Cyclic Repetition is pattern grouping for structures utilizing axial rotational symmetry

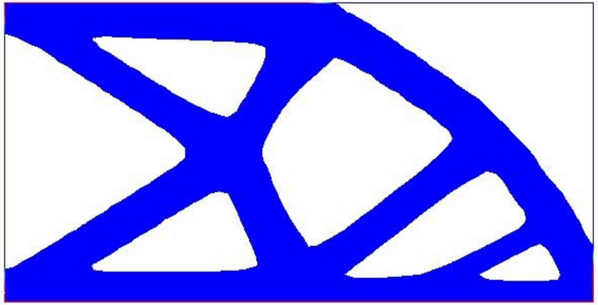
- Symmetry definitions are similar to planar pattern grouping
- Allows cyclic repetition of design features within a single domain
- User enters # of wedges and specifies an axis
- Use case: cyclic structures & non-symmetric loadcases



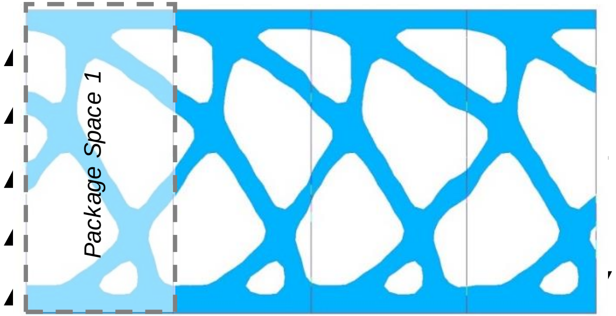
Using Manufacturing Constraints for Practical Design Concepts

Pattern repetition reproduces topological results in different structural components

- Package spaces can be of:
  - different sizes
  - different meshes
  - different components
- Scale patterns to different design regions
- Applied results may also be spatially reoriented according to user control



Topology without pattern repetition



Beam with 4 design areas:

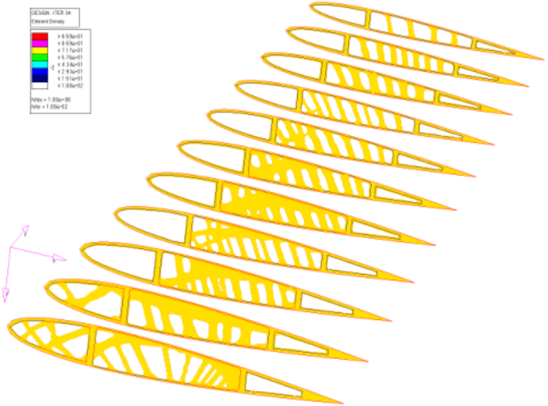
Topology with pattern repetition: 3 repetitions

All package spaces shall have similar topology

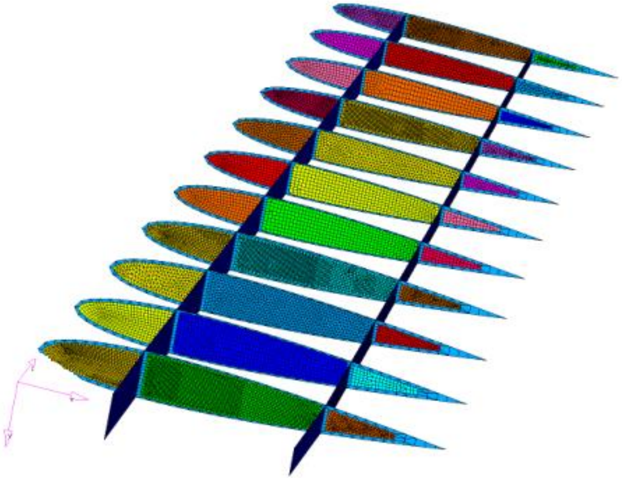
Using Manufacturing Constraints for Practical Design Concepts

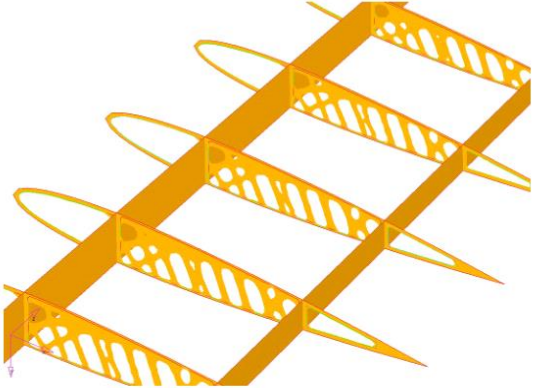
A practical application example of pattern repetition is airplane wing ribs

- Same topology on every rib
- Scaling factor to account for different sizes of design space



Without pattern repetition



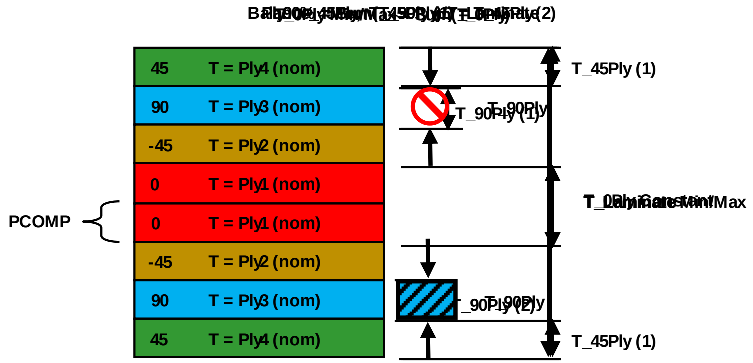


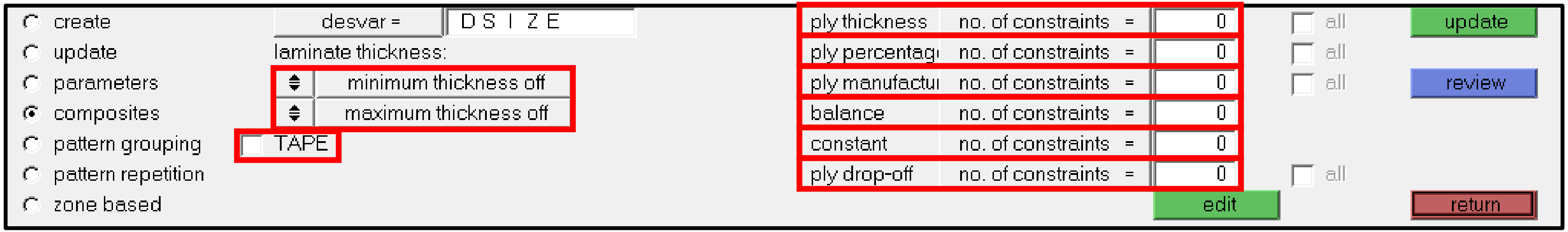
With pattern repetition

Composite-Specific Manufacturing Constraints for Optimization

The free size optimization panel composites subpanel allows the creation of composite-specific manufacturing constraints for optimization

- Min/Max Total Laminate Thickness (LAMTHK)
- Min/Max Individual Ply Thickness (PLYTHK)
- Min/Max Individual Ply Angle Percentage (i.e. %90…) (PLYPCT)
- Min Manufacturable Ply Thickness Constraints (PLYMAN)
- Balanced Ply Angles (i.e. Balance +/- 45's) (BALANCE)
- Constant Individual Ply Thickness (CONST)
- Ply Drop-Off Constraints (PLYDRP)
- Tape Laying Constraints (TAPE)



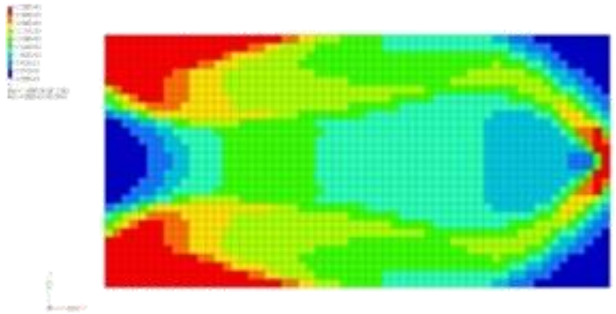


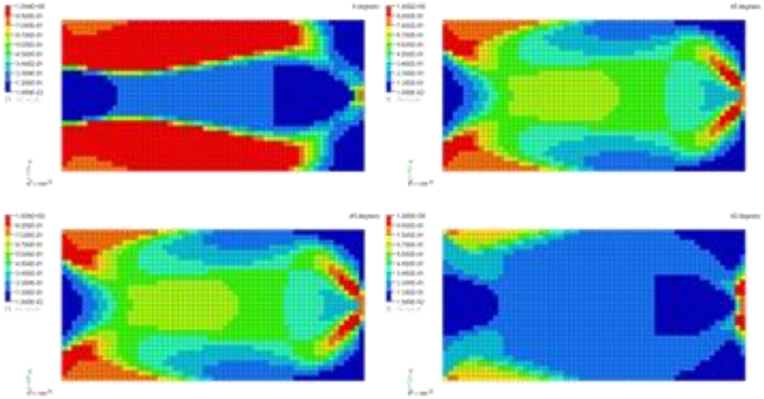
Phase 1: Free Size Optimization Results

When post-processing Free Size Optimization, results should be considered on several levels:

- Global: was the optimization able to achieve objectives and meet constraints?
- Tailored Patch: What are the size, shape, and ply depth of the optimized regions?
- Response: Is the analysis of the optimized structure reasonable?

When contour plotting thickness results, the total thickness of the composite layup may be broken down by ply angle



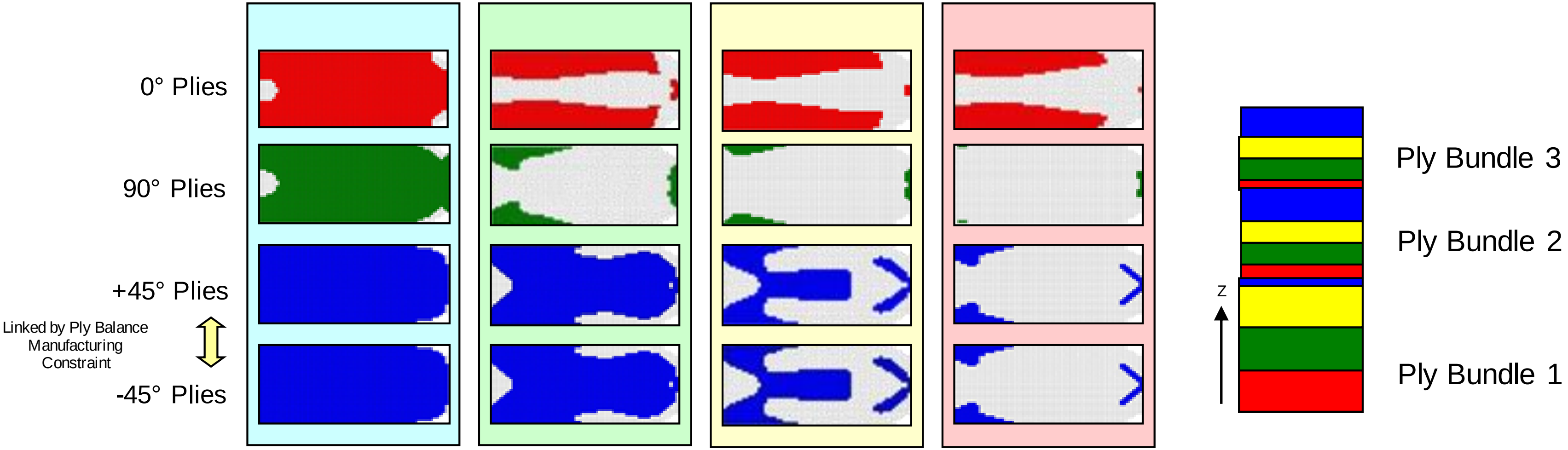


ALTAIR

Phase 1: Free Size Optimization Results

For each ply angle, the tailored patch shapes represent superply bundles which are generally split into four levels of resolution

- Users can define a different number of ply bundles per orientation to tune the complexity of the design
- The super ply bundles can be visualized using the Contour panel or the isosurface display
- The elements within each tailored patch ply bundle can be automatically exported by OptiStruct

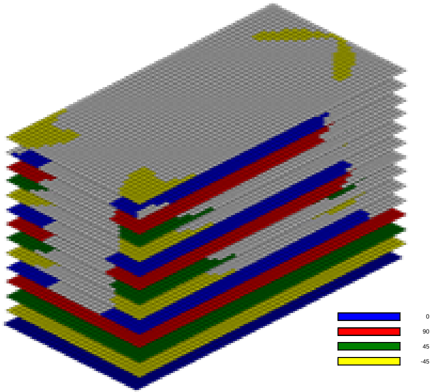


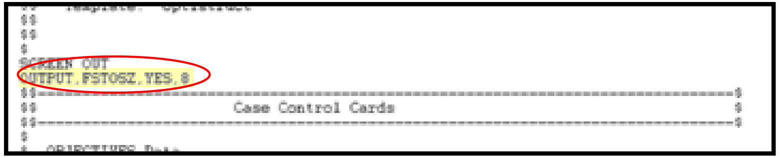
ALTAIR

Free-Size to Size Optimization Output Automation

Using the control card output parameter FSTOSZ instructs OptiStruct to a generate element sets and new property cards from ply bundles at the end of Phase 1 optimization

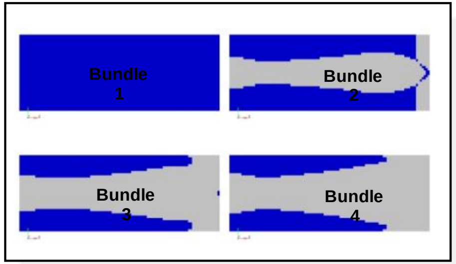
- Useful for transitioning automatically from FreeSize to Size Optimization
- Creates a new `*.sizing.fem` deck
- Number of ply bundles can be specified in the output



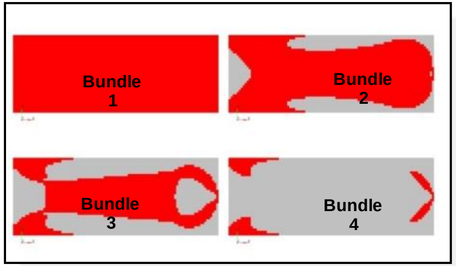


```
$$
$$
$
SCREEN OUT
OUTPUT,FSTOSZ,YES,8
$$------------------------------------------------------------------$
$$                    Case Control Cards                            $
$$------------------------------------------------------------------$
$
```

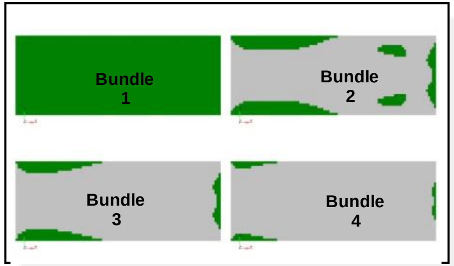
Level setting Ply-Bundles: 0° plies



Level setting Ply-Bundles: ±45° plies



Level setting Ply-Bundles: 90° plies



Exercise 8: Composite Optimization of a Plate with Hole Coupon

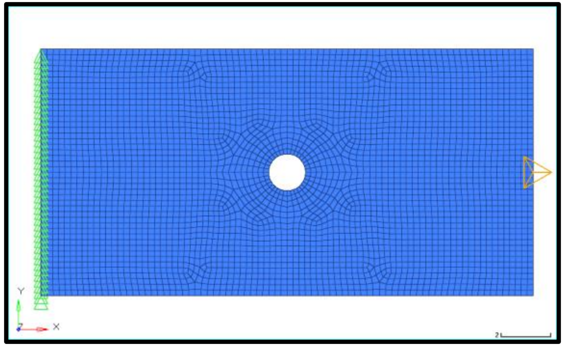
Exercise 8:

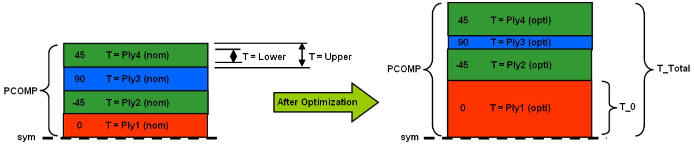
Composite Optimization of a Plate with Hole Coupon

Phase 1: Free Size Optimization : Steps 1-10

Objectives:

- Open the model in HyperMesh Desktop
- Create the design variable for free sizing optimization
- Add composite manufacturing constraints to the design var
- Add symmetry constraints to the design variable
- Add responses to the free size optimization setup
- Create a constraint on the displacement response
- Create a minimize volume objective
- Set output control cards to automatically output size optimization

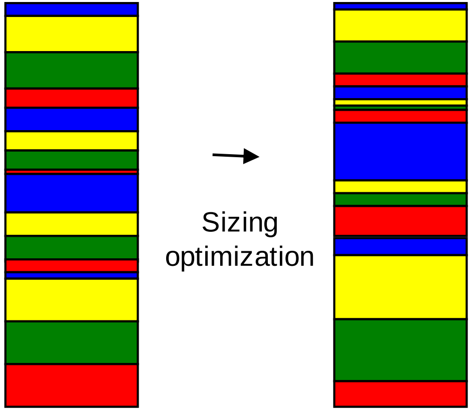




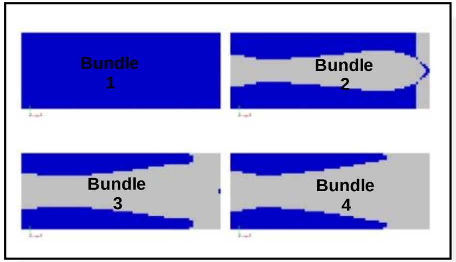
Phase 2: Size Optimization Setup

Size Optimization is a fine-tuning-level optimization that optimizes the thickness of the tailored plies

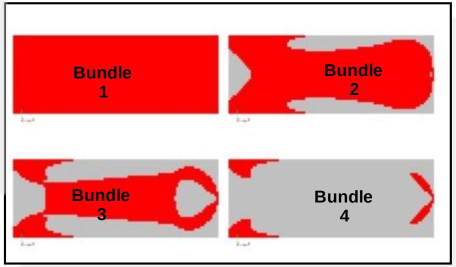
- All behavior constraints and manufacturing constraints carry over from free-size model using FSTOSZ
- Each ply bundle has a design variable (DESVAR) and design variable property relationship (DVPREL)
- Following a ply bundle sizing optimization, the number of plies required per orientation can be established simply by dividing each ply thickness by the thickness of the basic manfuacturable ply



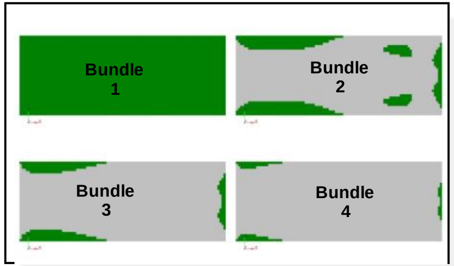
Level setting Ply-Bundles: 0° plies



Level setting Ply-Bundles: ±45° plies



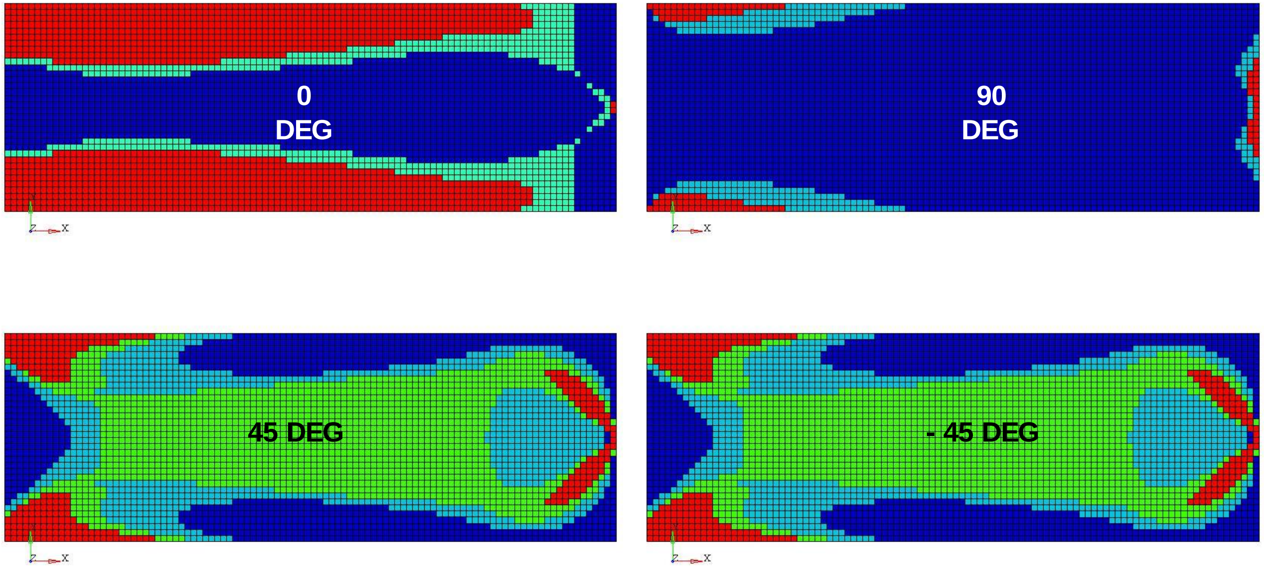
Level setting Ply-Bundles: 90° plies



Phase 2: Size Optimization Results

The results of size optimization give the optimum thickness for each ply bundle

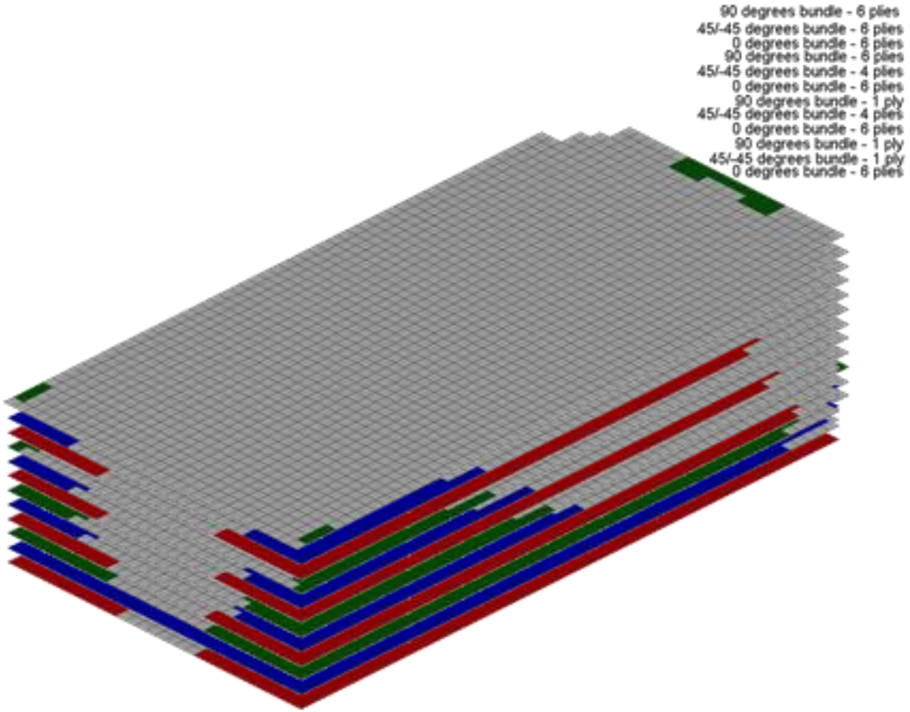
- When ply thickness is known, this thickness can be converted into number of plies
- Results are output under the final iteration listed in the `*.prop` file following optimization

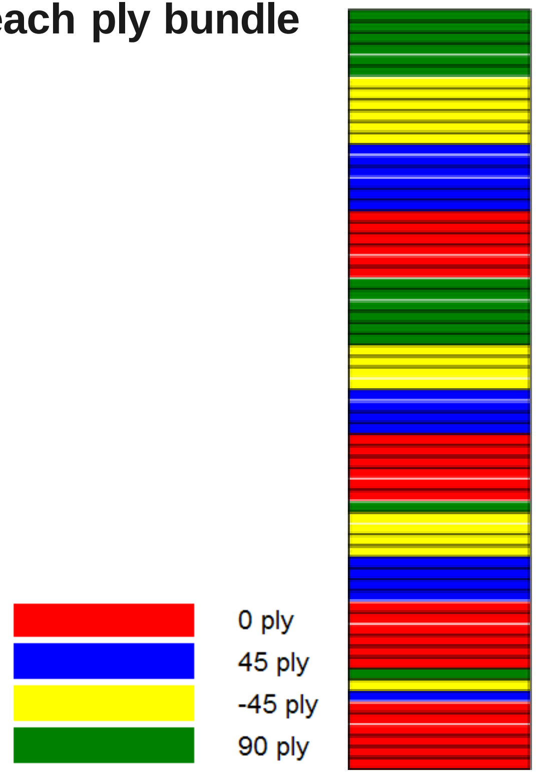


ALTAIR

Phase 2: Size Optimization Results

The results of size optimization give the optimum thickness for each ply bundle





Exercise 8: Composite Optimization of a Plate with Hole Coupon

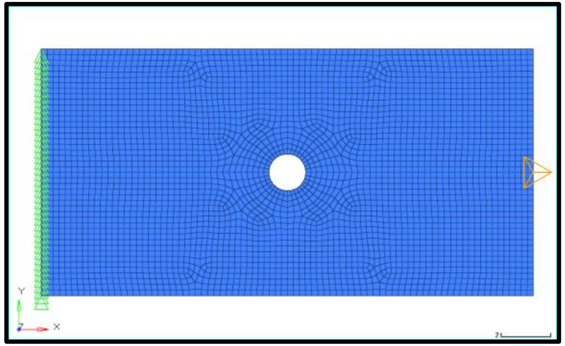
Exercise 8:

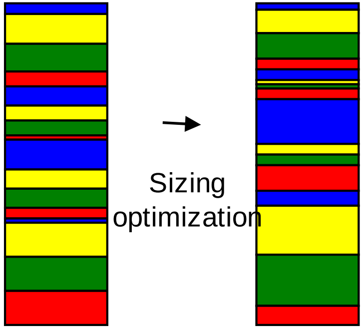
Composite Optimization of a Plate with Hole Coupon

Phase 2: Size Optimization : Steps 11-19

Objectives:

- Import the size optimization into a new session and review the ply bundles
- Review the new optimization entities created by FSTOSZ
- Update each of the desvar upper bounds to `0.1`
- Review and verify the composite manufacturing constraints
- Change the Laminate option from **`SMEAR`** to **`TOTAL`**
- Create a composite stress (normal 1) response on all plies
- Create constraint to bound **`stress`** between **`-25`** and **`25`**
- Set the SZTOSH output parameter
- Save the model as **`plate_with_hole_opti_phase2.hm`**
- Run the composite size optimization



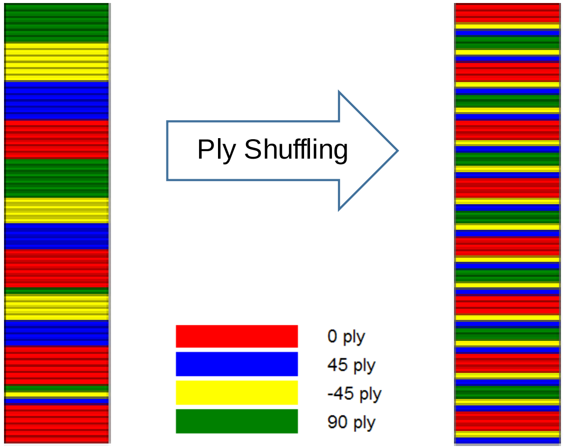


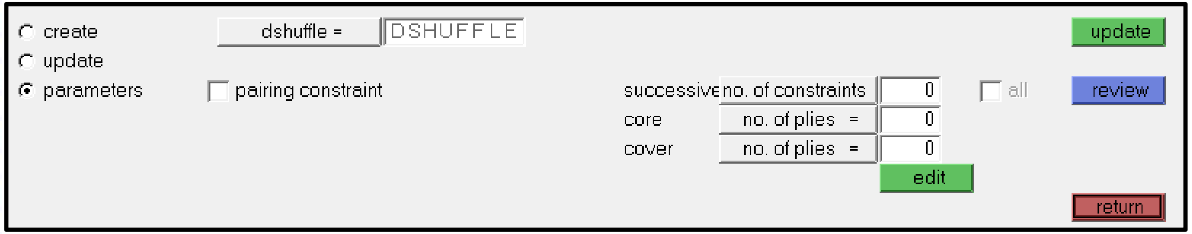
ALTAIR

Phase 3: Composite Shuffling Optimization Setup

In Composite Shuffling Optimization, OptiStruct determines the optimum stacking order for each individual ply, using ply book rules as manufacturing constraints

- Shuffling optimization models can be created from size optimizations through SZTOSH output parameter
- The optimization results determine the final stacking sequence for the model
- Ply book rules can be entered on the **composite shuffle** panel **parameter** subpanel





Phase 3: Shuffling Optimization Results

The results of shuffling optimization give the final stacking sequence for the part

- Results are available visually in an `*.html` file & STACK written in `*.prop` file

Stacking sequence for STACK 1

Iteration 0	Iteration 1	Iteration 2
13101	12301	12301
13201	14301	14301
11301	13101	13101
11302	11301	11301
12301	13201	13201
12302	12302	12302
13301	14302	14302
13302	11302	11302
14301	13301	13301
14302	11401	11401
11401	12401	12401
11402	14401	14401
12401	13302	13302
13401	11402	11402
14401	13401	13401

Legend
90.0 degrees
45.0 degrees
0.0 degrees
-45.0 degrees

Stacking Sequence before Optimization

```
STACK           1     SYM 1030101 1040101 1010201 1010202 1010203 1030201
+                 1040201 1010301 1010302 1010303 1010304 1010305 1020301
+                 1020302 1030301 1040301 1010401 1020401 1020402 1020403
+                 1020404 1020405 1030401 1030402 1040401 1040402
```

Stacking Sequence after Optimization

```
STACK           1SYM     1020302 1010203 1010201 1010202 1020402 1020401
+                 1040402 1040301 1040101 1010303 1010301 1010302 1030402
+                 1020405 1020403 1020404 1030101 1030301 1040401 1030401
+                 1030201 1010401 1010304 1010305 1040201 1020301
```


ALTAIR

Exercise 8: Composite Optimization of a Plate with Hole Coupon

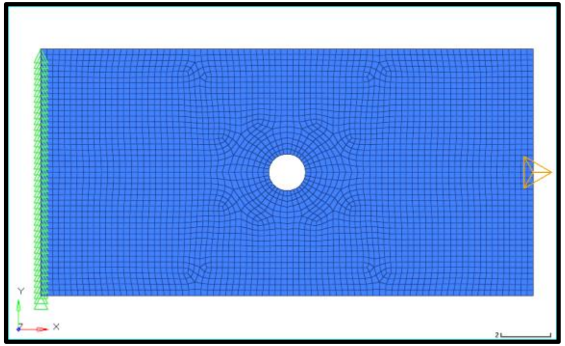
Exercise 8:

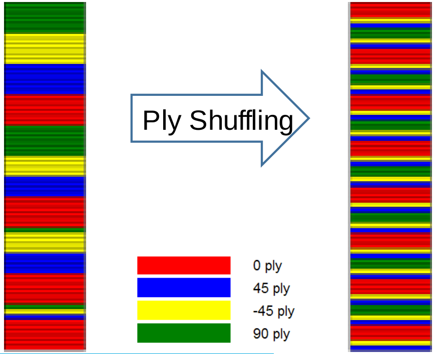
Composite Optimization of a Plate with Hole Coupon

Phase 3 Shuffling Optimization : Steps 20-25

Objectives:

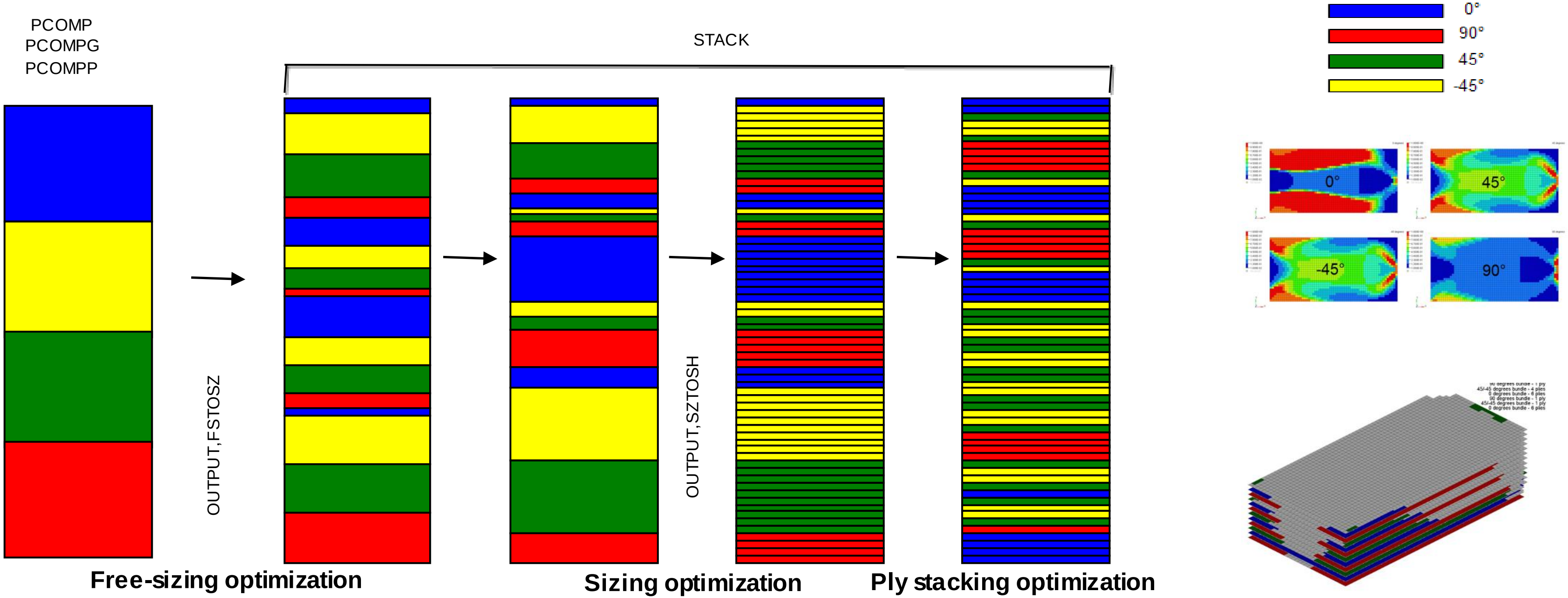
- Import the shuffling optimization file in a new HyperMesh session
- Set the control cards to output property, HTML shuffling report, and H3D results
- Add ply book constraints to the shuffling optimization
- Save the model as **`plate_with_hole_opti_phase3.hm`**
- Run the shuffling optimization
- Review the results using the **`*.prop`** file generated





Composite Optimization-Based-Design Summary

Composite optimization-based-design provides a 3-step highly-automated workflow that transforms FE analysis models into insightful optimized design solutions



ALTAIR